41Hz Audio

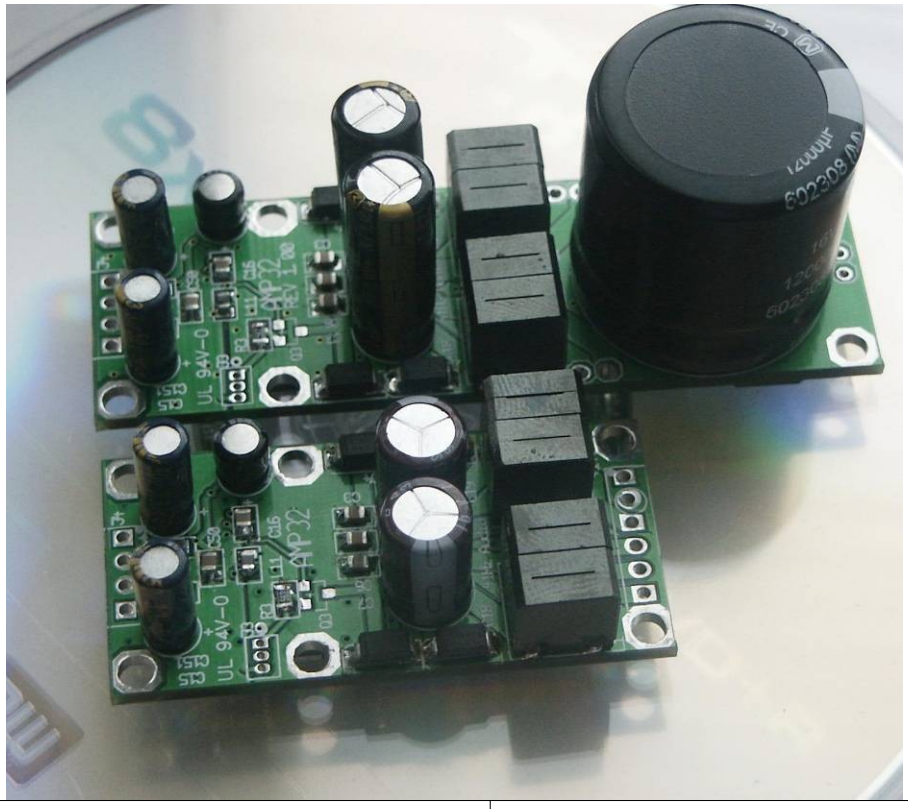
www.41hz.com

Assembly instructions AMP6-BASIC

Assembly instruction **Revision 1.0 December 2021**

Document ID: 25

Written by Jan Fredriksson



AMP32 FEATURES

- Very small footprint
- Mostly surface mounted components
- Read wound output inductors supplied
- Status output LED for sleep / awake and overload
- Low noise quality components supplied
- Chip can be cooled by amplifier casing

AMP32-PS additional FEATURES

- Rectifier with very low voltage drop Schottky diodes
- 10.000 uF bulk capacitor

TA2021B FEATURES

- Class-T architecture
- Single Supply Operation
- “Audiophile” Quality Sound
- 0.05% THD+N @ 13W 4Ω
- 0.1% THD+N @15.5W 4Ω
- 0.1% IHF-IM @ 1W 4Ω
- High Power
- 25W @ 4Ω, 10% THD+N, V_{DD}=14.6V
- 23.5W @ 4Ω, 10% THD+N, V_{DD}=14.2V
- 14W @ 8Ω, 10% THD+N, V_{DD}=14.2V
- High Efficiency
- 88% @ 13.5W 8Ω
- 81% @ 25W 4Ω
- Dynamic Range = 100 dB
- Mute and Sleep inputs
- Turn-on & turn-off pop suppression
- Over-current protection
- Over-temperature protection
- Bridged outputs
- 36-pin PSOP “Slug-Up” package

Table of Contents

Assembly instructions AMP32 and AMP32-PS

Introduction

Thank you for choosing an audio product from 41hz.com! On the download page of the support web site, http://www.support.14hz.com there is additional information available. There may also be later 7 upgraded versions of this assembly instruction.

Check delivery

On delivery, check that all components have been included. We do double-check the component count but mistakes can happen. Work on an area where you can recover any component you drop while unpacking. More than one customer has lost components between the floorboards. You may need a loupe or magnifying glass to identify components. If something is missing, let us know immediately. A bill of material (BOM) is found as APPENDIX 1 in this document.

IMPORTANT

1. **The Tripath chips use MOSFEt outputs which by nature are sensitive to ESD (Electro Static Discharge). Use ESD precautions. Preferably work on a conductive, grounded "ESD mat", and avoid touching the chip leads with your fingers. Discharge yourself before working with the components. Also the little FET in SOT23 package is a FET which is sensitive.**
2. **The chip and AMP32 board is not protected from over voltage or reversed / wrong polarity voltage. Connecting a voltage higher than 14.6V or wrong polarity may damage the Tripath chip permanently.**
   (AMP32-PS has polarity protection, but not over-voltage protection.)

Tools needed

Assembly of the kits requires the usual set of electronics working tools; soldering iron, wire cutter etc. The chip of this amplifier, the TA2021B has a small footprint and thin, closely spaced connectors. You need a fine tipped solder iron for these. A tip diameter of 0.3 – 0.6 mm should work well. Chisel shaped solder iron tips can be used by the experienced builder. The boards for AMP32 are double weight, double sided copper. Even if the PCB and components are small, quite a powerful soldering iron is very helpful. Especially components and pads connected to the ground plane require significant heating. A temperature controlled 50W soldering iron is the minimum recommended. At the same time, applying excessive heat may damage the board, causing the copper leads to come off. Preheating the board to around 100ºC will make work easier and allows using a lower solder iron temperature which decreases the damage risk.

A magnifying glass/lupe of the type that you wear like a pair of glasses or like a cap is recommended, as it increases the precision and quality of your work. Some information on how to solder SMT components is available in the forum on http://www.41hz.com

The schematics used for AMP32 is almost identical to the schematics in the Tripath data sheet for the TA2021B chip. However there are differences. These are pointed out in comments to the BOM in APPENDIX 1.

Additional components

The following will at some stage be needed to complete the amplifier, but is not included in the kit:

- Heat sink. Screws and heat conductive paste to mount the heat sink. In most cases, if you mount the board and chip to an aluminum amplifier casing is sufficient to cool the chip.
- Hookup wire. I recommend soldering connection wires to the board. Optionally you can fit screw/solder terminals with 2.54 mm spacing.
- Mute/un-mute switch or jumper. Either wire this to a switch on your panel or use a 1.24 mm jumper (50 mil). Or if you use a power supply line switch, you can permanently close the mute jumper with a short piece of wire. The board should draw less than 0.3 mA when muted. Note that if you use a stabilized power supply, many voltage regulators will not work (voltage usually drops…) at this low current.

Considerations

1. On the board there are two signal input capacitors, C14 and C15. These are required, as the amplifier is internally biased to about +2.5V. The board provides space for both chip and RM 2.54 (100 mil) through-hole capacitors. In the kit there are provided two 2.2 uF ceramic chip capacitors and two 3.3 uF trough hole electrolyte capacitors. You can use either one or even both types in parallel. Some possible choices include: electrolytic capacitors, “plastic” and tantalum capacitors. Using to large capacitance may produce power up thumps, so it is best not to use excessively large capacitors. The input capacitors form a high pass filter together with the input resistor Rin. The cutoff frequency is F=1/(2*π* Rin*Cin) For example, with Rin = 22 Kohm and Cin = 3.3 uF, the cutoff frequency is F=1/(2*3.14*22000*0.0000033) =2.7 Hz. The cutoff frequency is best kept at least two octaves below the lowest frequency expected.

2. The amplifier input stage, in the Tripath chip, is of the operational amplifier type. The maximum possible voltage the input stage can handle is 4V peak to peak (1.41 VRMS). You can set the gain of the input stage so that it matches your signal source. The gain is calculated as for a normal inverting operational amplifier: Input Gain=-1*Rfeedback/Rin [V/V]. The minus sign is due to the fact that the input stage is inverting. On the board, R2 and R4 are the Rin and R5 and R6 are the Rfeedback. With the kit, there are four 22 KΩ resistors and two 36 KΩ. With these resistor values, you can choose one of three different input sensitivities as shown in table 1. If you use other input resistors they should be of a low noise type.

   The gain of the whole amplifier ($volt_{out}/volt_{in}$) is 12 times the input gain.

3. Will you use a volume control / pot? If you have a preamplifier or sound source with its own volume control, it may be best to leave out the volume pot. If not, a volume pot of 50 kohm pot would be suitable. With a volume pot, there will be some signal damping so you may need to increase the gain a little. Some examples of gain settings are given in table 1. Note that some portable players will clip badly at full volume; that is the signal source output clips, even if the power amp does not clip.

4. Sleep and mute. The Tripath TA2021B chip has a *sleep* and a *mute* function. Both can shut down the amplifier.

   a. The amp is in sleep mode until pin 2 of J3 is connected to ground, J3, Pin 3, or better, connect the pins through a switch on the front panel of your amp. In sleep mode the amplifier draws less than 1mA, so you can use the sleep switch to turn the amp on and off. Note that some stabilized power supplies may not work properly at very low currents. You should check this.

b. The chip *mute* input is hard wired to the chip error/over-temperature sensing output on the PCB. In case a too high temperature is detected, this mutes the amp. It automatically and un-mutes again when the chip has cooled down a bit. In case of over-current the amp is muted in a latched way and must then be power toggled off/on to be restarted.

5. You can use screw terminals or solder hookup wire to the PCB. Soldering is generally the best connection from an electrical / signal point of view but may be unpractical. Note that you should avoid soldering on/off the cables, especially the power and speaker cables. As these cables are usually quite thick, they will require substantial heating. Repeatedly soldering these may cause the copper tracks to come off, lift, because the FRP below them is beginning to deteriorate. It is better to unsolder/cut the “other” end of the cable or use a board connector.

6. Power supply. For testing, any 12V supply should work. For more permanent supply, you can look at the web site for some hints on power supplies for the amp.

R_{in}	$R_{feedback}$	Input Gain	Suitable signal source
22 KΩ	47 KΩ	-2.13 V/V	Direct connection of portable MP3/CD player with built in volume control or a volume pot in the power amp.
22 KΩ	22 KΩ	-1 V/V	General use
47 KΩ	22 KΩ	-0.47V/V	Preamplifier with fairly high output signal

Table 1. Gain setting recommendations. R2 and R4 are the R_{in} and R5 and R6 are the $R_{feedback}$

Mounting the components

1. First, solder all surface mount components except the TA2021B chip. It is suggested you work in the order of the BOM. The BOM is in APPENDIX 1. Components are placed on both sides of the board. A picture of how the components are placed is included in APPENDIX 2. Note the polarity of the diodes.

2. Optionally solder board connectors. If you do not use these, save cabling until last. See the APPENDIXes for pin-out.

3. Solder though-hole components, but *not* C18 and C19 yet. When through hole components are soldered properly on the back of the board, there should be solder right through the board holes and solder should be coming out on the other side, covering all around the component leg. Respect the polarity of electrolyte capacitors as marked on the PCBs. For the C2 capacitor, the rectangular pad is positive, the round one is negative.

4. Solder the TA021B chip in place. Take care getting the chip straight. Use a fine tipped solder iron for soldering. Use as little solder as you can. It is advised you use some extra flux as this makes soldering a lot simpler. There are two pads that are doubled, with two of the chip leads soldered to one PCB pad/trace. It is very useful to have a fine grade solder wick to remove excess solder. Solder wick is fine braided wire that can soak up excess molten solder when heated on top of a location with excess solder. Heat the braid and remove it before the solder solidifies again.

5. Solder the inductors in place. It is important to get good connection here. Poor soldering of the inductors may cause strange problems.

6. Now solder C18 and C19

7. For AMP32-PS, solder the power supply components in place (these are not on the AMP32-board, only the AMP-32-PS boards) These are the four SMC size diodes, two 0.1 uF capacitors and the bulk capacitor.

8. Last connect signal and power connectors and you are ready for testing. For testing at low power no heat sink is required. For low to medium power applications the amplifier housing may be sufficient as a heat sink. For high power use, into 4 ohm speakers, the amplifier can dissipate about 10W of heat at full power. Then, a 5ºC/W heat sink is reasonable. Medium or low power applications will not dissipate a lot of heat. The heat slug on the chip is connected to ground and does not require electrically insulated mounting. Silica heat transfer compound or similar should be used to improve cooling.

Four of the six mounting holes on the PCB are connected to ground. The two mounting holes in the signal input end of the board are not, and should not be, grounded on the board. The other four screw holes are connected to the power supply ground. It is normally preferable to allow these to be grounded.

Trimming and testing

Do not connect power yet!

- Open the jumper J3 for *sleep* mode.
- For testing, use a 300 mA slow fuse on the power supply.
- Make sure the power supply is off.
- Connect the power supply, +12 volts (+ 14.6V absolute maximum). ***Check the polarity of the power supply. Wrong polarity will permanently destroy the TA2021B chip****.*
- Turn on the power supply. When in sleep mode, the amp should draw less than 0.5 mA. If it draws more, disconnect the amp and check everything.
- If all is OK, connect the J2 jumper to make the amp Awake
- The supply should now draw about 30-40 mA.
- Check the fuse. If it has blown, shut of the power, disconnect the board and check all components and solder connections.
- If all seems OK, shut of the power
- Connect the speaker wires to J1. Important: the output is bridged, so each speaker should connect to its own respective plus and minus. The minus is NOT ground and negative is NOT common for the two channels and NOT common to the power supply minus/ground.
- Connect a signal source with its ground leads to J4
- Connect a signal source and set the volume very low
- Turn on the power and check if you get any sound.
- If everything seems OK, you can slowly increase the power. Note the 200 mA fuse will act as a resistor when near its rating, lowering the input voltage. Switch off power, replace the power supply fuse for a larger one and try again with higher volume. For testing at higher power, the chip should be mounted on a heat sink.
- Enjoy the music!

Note: if the amplifier seems to clip a lot at medium/high power output, then check the voltage supply, if possible with an oscilloscope. At high power your amp draws a lot of current. If the voltage from the supply then drops a lot, you will get clipping at a lower power than the amplifier is capable of delivering. In worst case, the voltage drops below 8V where the amp will switch off, then the voltage picks up again and the amp switches on etc. The amp can switch of /on at a high rate which sounds terrible and can potentially be damaging. If this happens, immediately shut of the signal or power and check if your power supply is capable of delivering a steady voltage at the required current. There are two 330 uF capacitors on the board. These handle the high frequency (≈600kHz) currents drawn by the amplifier but are too small to sustain high currents at low frequencies so the board needs a stable supply to fill up these onboard capacitors. In APPENDIX 3 there is a discussion on power supplies and some suggestions.

There is a troubleshooting guide available on the support web site among the downloads.

If you have any questions, comments or feedback, please write in the forum on the web site http://www.41hz.com. You can of course also contact us at jan@41hz.com

APPENDIXES

BOM (Bill Of Material)

Count	Name	Value	Package
2	C29, C30	0.010 uF	SMT 0805
8	C3, C6, C7, C9,C16, C20, C26, C50	0.1 uF	SMT 0805
2	C12, C13	0.22 uF	SMT 0805
4	C22, C25, C27, C28	0.47 uF	SMT 0805
1	C5	1 uF	SMT 1206
2	C23, C24	100 pF	SMT 0805
1	C21	1000 pF	SMT 0805
2	C14, C15 *(note 1)*	2.2 uF or 3.3 uF	SMT 1206
1	R1	1M	SMT 0805
1	R3	5.6 K	SMT 0805
2	R9, R10	10 ohm	SMT 1206
4	R2, R4, R5, R6 *(note 2)*	22 K	SMT 0805
1	R7	8.2 K	SMT 0805
1	L11	HF filter choke	SMT 0805
1	L10	HF filter choke 6A	SMT 1206
8	D1, D2, D3, D4, D5, D6, D7, D8	Ultrafast SMA diode	SMA
1	Q3	N-chan FET SOT23	SOT23
1	U1	TA2021B	PSOP36
1	C2	10 uF 10V	RADIAL RM2.5
2	L1, L2	Dual inductor	
2	C18, C19	330uF 16V	RADIAL RM2.5
1	PCB	-	
	The components below are for the AMP32-PS only		
2	C60, C61	0.1 uF	SMT 0805
1	C1	12000 uF 16V	Cap 25 mm RM10
4	D11, D12, D13, D14	Schottky 4A diodes	SMC

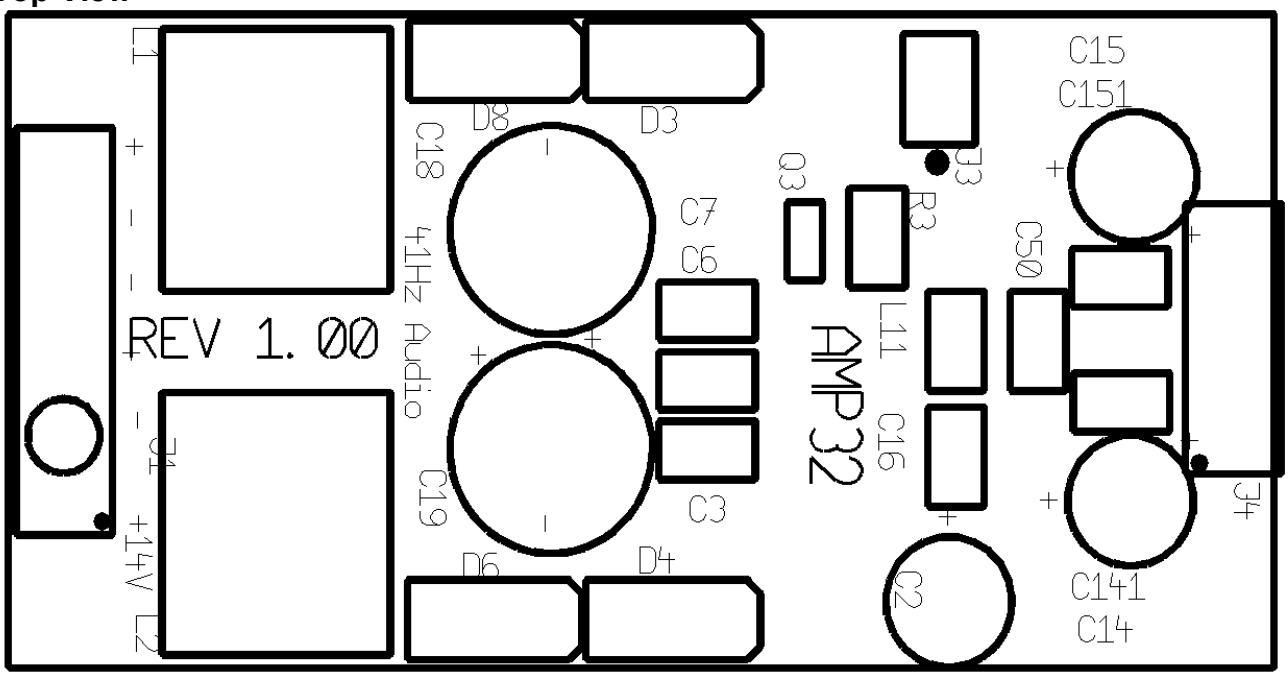
Note 1. As the result of a bad case of indecisiveness, two types of capacitors have been provided for the signal inputs, C14 and C15: ceramic chips of 2.2 uF and electrolyte of 3.3 uF. You can use either type.

Note 2. Two different values of input / feedback resistors have been provided, so that the input sensitivity can be adjusted. See *table 1* in the assembly instructions.

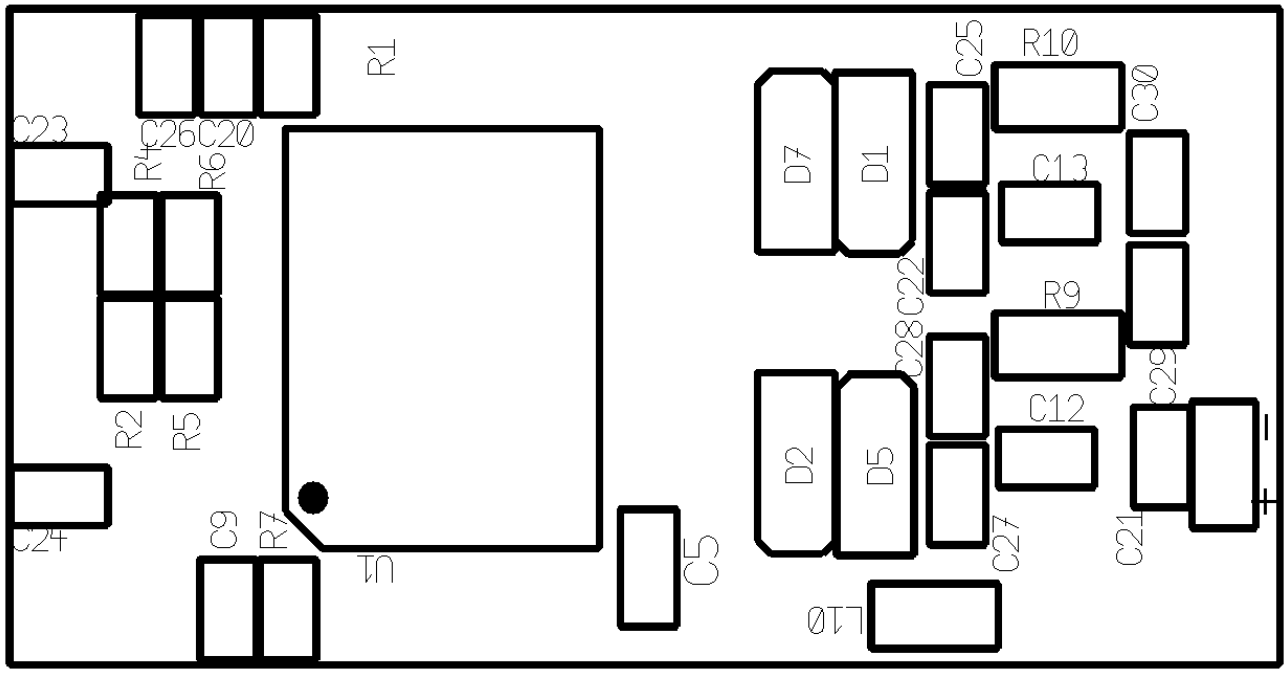
Note 3. The resistor values are printed on the components. The resistor value is given with three or four digits. The last digit always tells you the number of zeroes to add to the value. Example: **100** means 10 with no zero = 10 ohms. **105** means 10 with 5 zeros **=** 10 00000 = 1M ohm. **363** means 36 with three zeroes = 36 000 = 36 Kohms. Surface mount capacitors do not have any value printed on them, so you must keep the different values apart.

AMP32 Components Placement

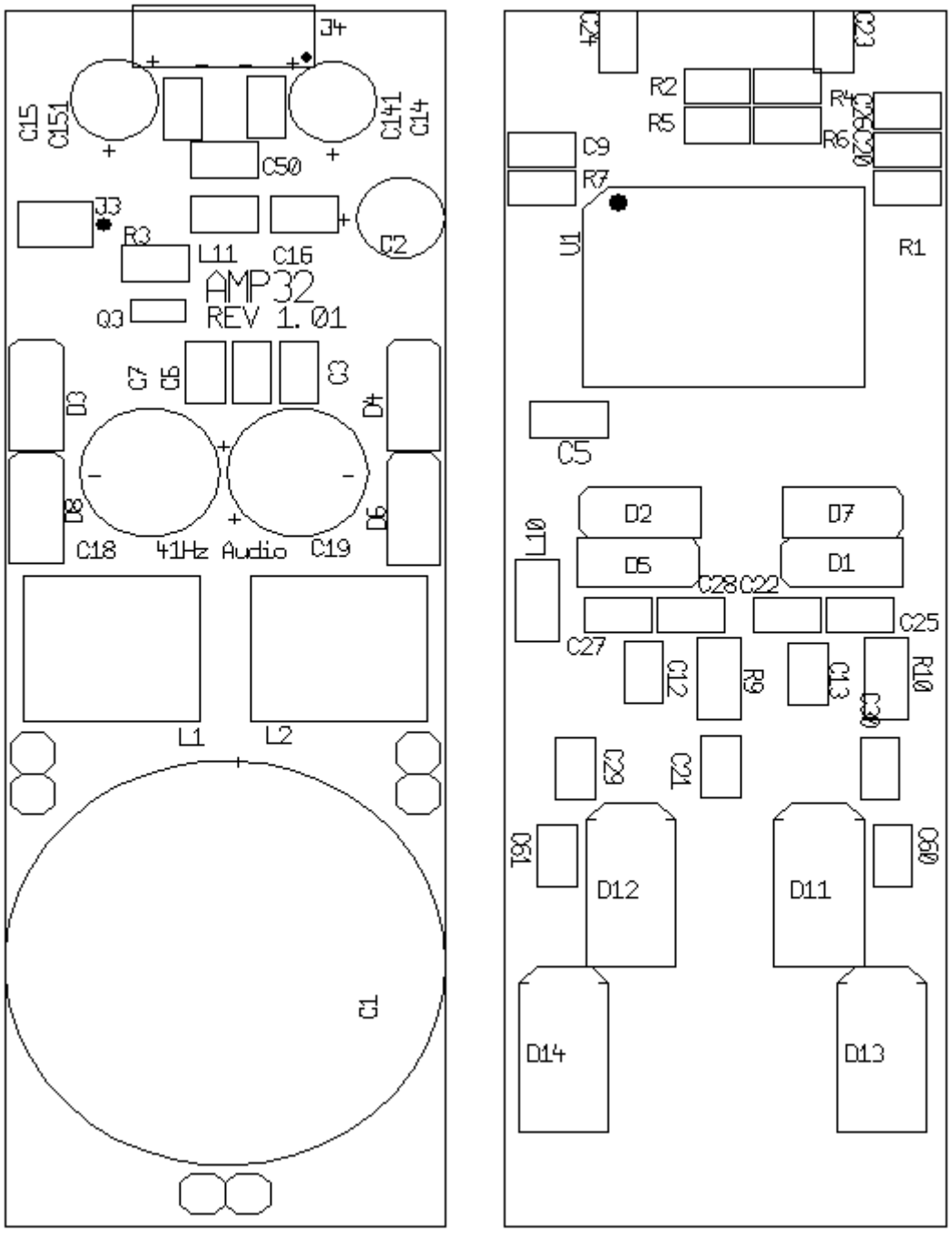
Top View



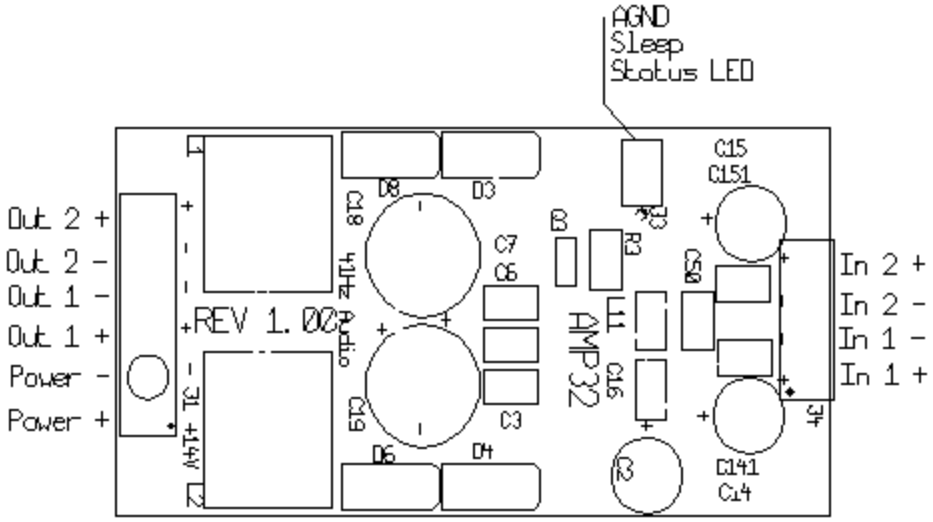
Bottom view



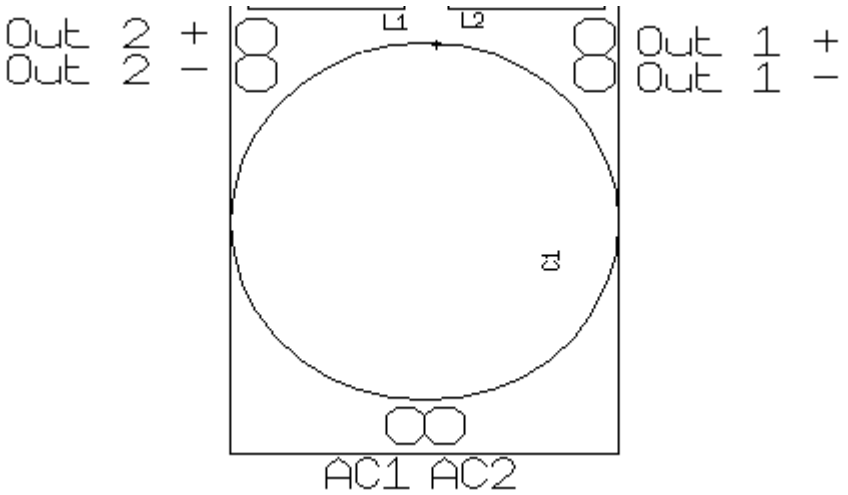
AMP32-PS Components Placement



Pin-out for AMP32



Pin-out out for AMP32-PS power and output section



Pins for J3 and J4 for AMP32-PS are identical to AMP32