Assembly instructions AMP32 and AMP32-PS

Thank you for choosing an audio product from 41hz.com!

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.

NOTE: components packaged in a shielded, aluminized bag should be considered ESD sensitive.

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 are double sided, weight copper so a high power, at least 50W, solder iron is recommended. Solder irons without temperature control should not be used. 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.

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.

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.**
2. **The chip and AMP32 board is not protected from over voltage or reversed 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.**

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.

If you are not familiar with soldering surface mount components, I recommend you to do some test soldering on a separate scrap piece of material.

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. Some information on how to solder SMT components is available in the forum on http://www.41hz.com

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 in the speakers. 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 = 20 Kohm and Cin = 3.3 uF, the cutoff frequency is F=1/(2*3.14*20000*0.0000033) =2.5 Hz. The cutoff frequency is best kept at least two octaves below the lowest frequency expected. Note that a big input capacitor may contribute to startup thumps.

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 20 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 7 of J2 is connected to ground (typically pin 8). You can make this connection by soldering a jumper between pins 7 & 8, or better, connect the pins through a switch on the front panel of your amp, which you can use to turn the amp on and off. In sleep mode the amplifier draws less than 0.3 mA. Note that many stabilized power supplies may not work properly at this low current. 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. So repeatedly soldering these may cause the copper tracks to come off, lift, because the FRP below them is beginning to deteriorate. It is then 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. Information is also included as APPENDIX 3 of this file.

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.

Board with most surface mount components in place

On the AMP32 kit PCB:s the +5V required is generated in the chip.

2. Optionally solder board connectors. If you do not use these, save cabling until last. See the APPENDIX 2 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. After these, C18 and C19

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

7. 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 20W of heat at full power. Then, a 3ºC/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

- 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.

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

APPENDIX 1

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	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.

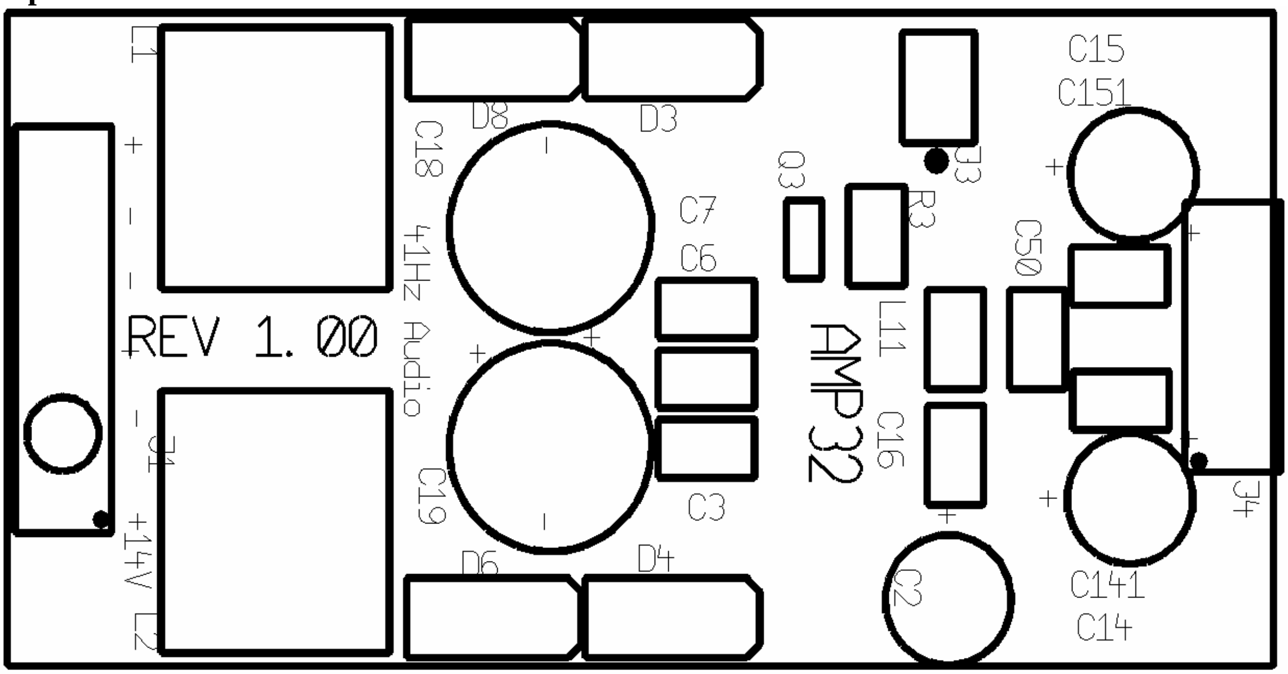
Note 2. Two different values of input / feedback resistors have been provided, so that the input sensitivity can be adjusted. See *table 1* in the assembly instructions.

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.

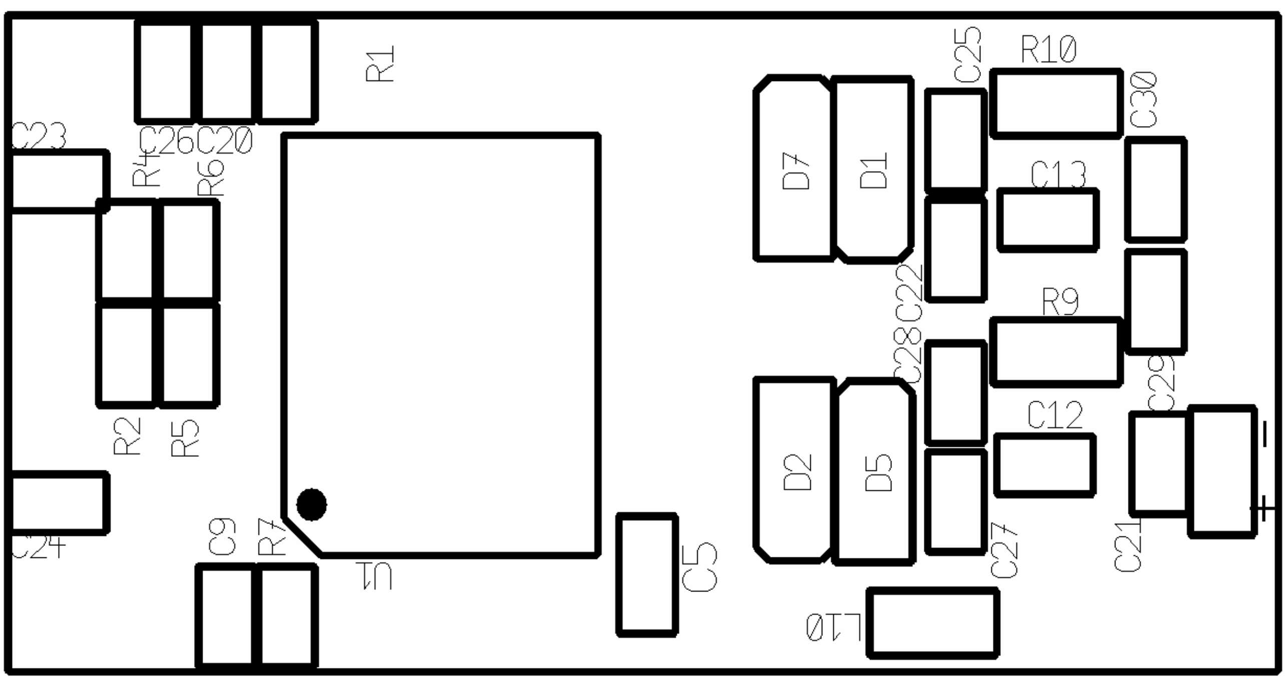
APPENDIX 2

AMP32 component placement

Top View

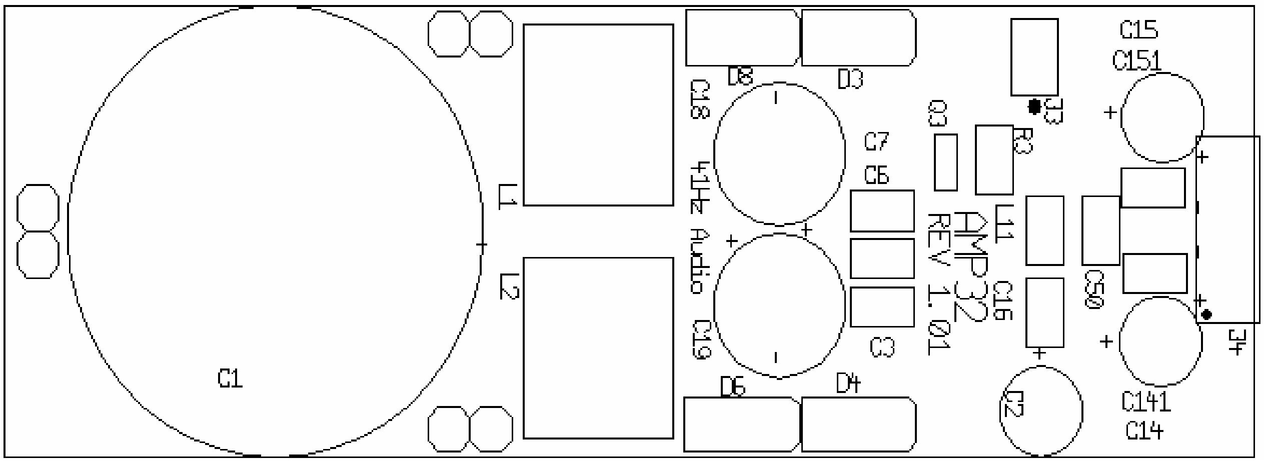


Bottom view

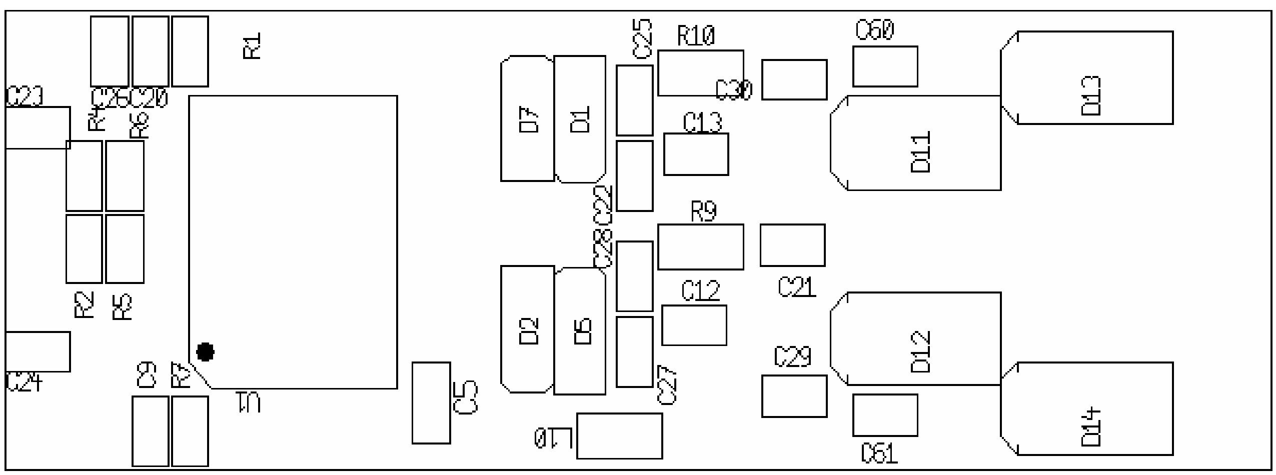


AMP32-PS component placement

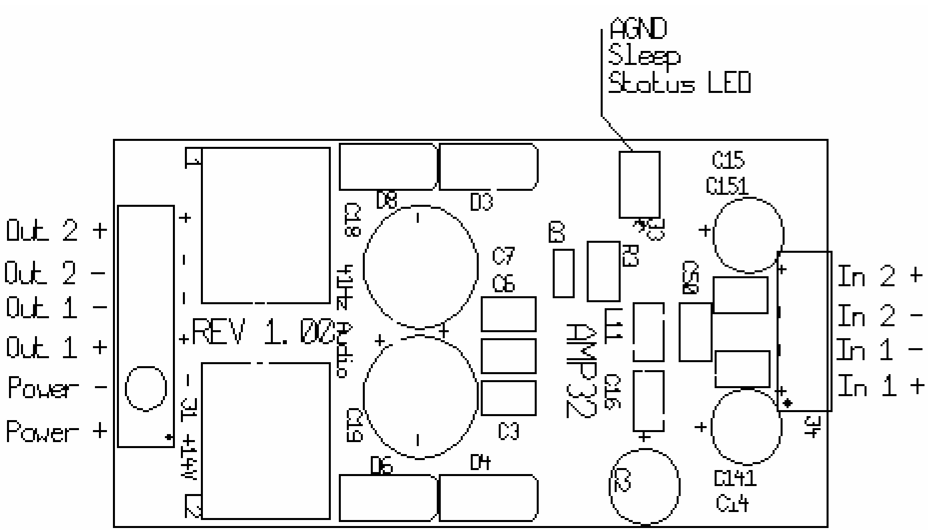
Top view



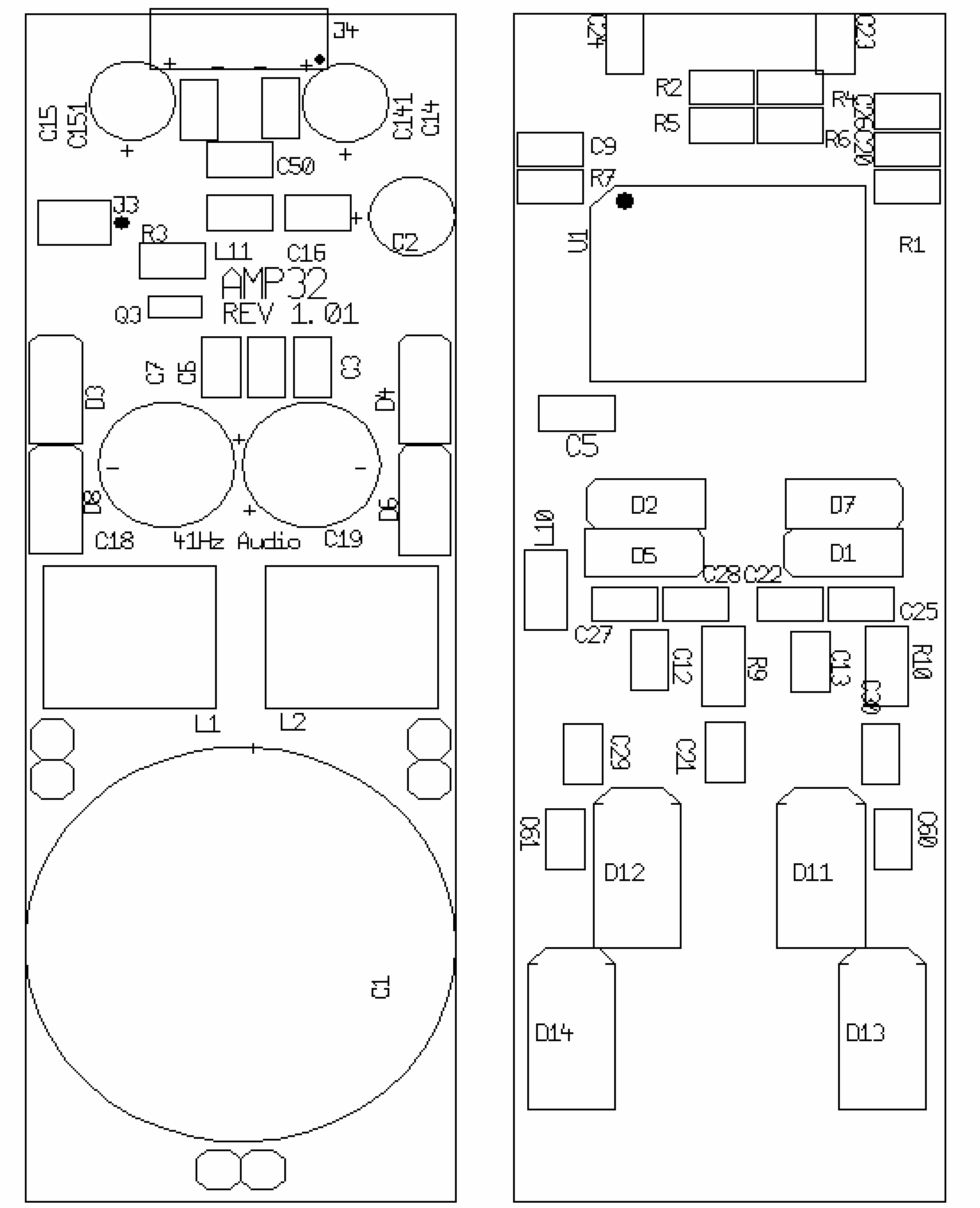
Bottom View



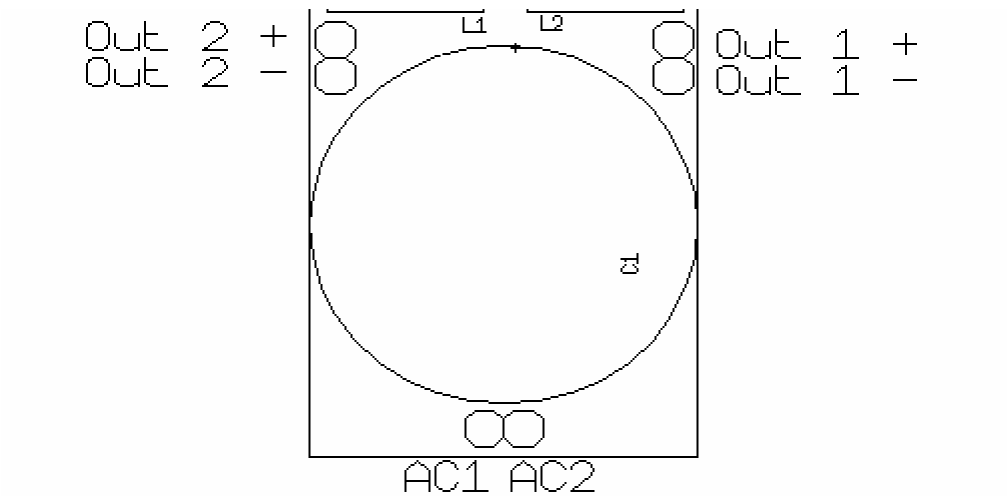
Pinout



AMP32-PS Top and bottom view



Pinout AMP32-PS power and output section



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

APPENDIX 3

Power supplies for AMP32

Amp32-PS

AMP32-PS has an onboard rectifier and bulk capacitor. You can connect it to a 9V transformer or 12V DC supply. The onboard rectifiers have a very low voltage drop so if you use this amp with a DC source it is hardly worth the effort to bypass them. The diodes also provide polarity protection (not on the AMP32, only AMP32-PS!)

AMP32, simplest

I expect many people will run this amplifier of batteries. I would recommend the sealed lead battery type, with a capacity of about 10 AH. This is the simplest power supply you can get! With a battery like this, the AMP32 will run continuously for days at moderate volume and for many hours even at high volume. A battery supply will benefit a lot from having large capacitors added. The capacitors will then handle fast power transients. Typically you could try using 12.000 uF of 16V capacitors.

Simplest nr 2

An option is to use a SMPS which are available in various power ranges and voltages. There are some new models which are very quiet and very reasonable priced.

How much power?

The AMP32 is rated at 2x25W into 4 ohms with a 14.6 V supply. It should be noted that this value is at 10% distortion. Power amps will deliver low THD+N up to about 60% the rated powers. At higher levels the amplifier will clip as there is simply not enough voltage to reproduce the signal peaks. For Tripath chip based amplifiers, clipping is quite clean/soft and not very disturbing. But you should probably turn down the volume a bit when the amplifiers start to clip. The average music power output will be a lot lower than the peaks. So in reality your amplifier will probably run at much lower average power than the maximum rated. If not you should get a larger amp... The efficiency of AMP32 at full power is about 80% into 4 ohm loads and close to 90% into 8 ohm loads.

VA Rating

So the full power with these amps is 50W and the efficiency is 80%. Therefore a maximum power of about 60W is needed. A budget solution would be about 30VA for 8 ohm loads and perhaps 60VA for 4 ohms, relying on that the amp will not be operated near the full rated power for extended periods. A higher VA rating is better as it gives margin in terms of overheating and stability. There is probably not much advantage in going above 50VA for 8 ohm loads and 100VA for 4 ohm loads. If the voltage is below the maximum rated, then a high VA rating can not damage your amplifier, only your wallet. You could use a 1000 VA transformer if you like but there would be no real advantage over 100VA transformer. The amplifier is rated at an absolute maximum of

16V while nominal operating voltage is set to 8.5-14.6V. With a 14.6V supply, you have some margin for transients that could otherwise damage the amplifier. The higher voltage, the higher the maximum power. But higher voltages than 14.6V are NOT recommended. Note that amplifier power output is proportional to voltage squared.

Transformer voltage ratings are for maximum output. The idle voltage is higher. The smaller the transformer, the higher the idle voltage will be. This is so to compensate for losses which are generally higher in small transformers. A typical 100VA transformer would have about 10% higher voltage at idle load than at full load. A typical 50VA transformer may have 15% higher voltage at idle than at full load. Allow for this when selecting a transformer. The voltage should not go above 14.6V even when idling.

Stabilized or un-stabilized power supply

These amps use a feedback topology and power supply rejection is quite good so for audio quality, a stabilized supply should not be required. But in reality, a stabilized supply has advantages. To illustrate how, let's look at an example of an example of an un-stabilized power supply:

Let's say you use a 50VA / 9V AC transformer. After rectification you would have a full power voltage of 1.41 x9 V = 12.7V. You will have a voltage drop of around 1V in the diodes so remaining you will have about 11.7V at full load. At idle you may have 20% higher voltage = 14V which is OK. A 10VAC transformer could have an idle voltage unpleasantly close to the 16V limit. With 11.7V at full power you would have about 2x16W output compared to 25W at 14.6V. So if your mains grid voltage is stable and you are not aiming for maximizing the power output you may be happy using a supply built with an un-stabilized 9VAC transformer.

But if your mains grid supply is less voltage stable and/or if you want to maximize the output, you may want to use a stabilized power supply. It could be designed so that you will have 14.6 V at full load and would not be any higher at idle, even if the grid voltage varies a bit. So a stabilized power supply makes sense for these amplifiers. And as the power is fairly small it is feasible to build one.

A stabilized power supply would consist of a rectifier diode, upstream bulk capacitor, voltage stabilizer and a downstream bulk capacitor. Add some protection diodes and a fuse and you are in business. As there is a certain loss over the voltage stabilizer, your transformer may need to be slightly larger than for an un-stabilized supply.

Capacitors

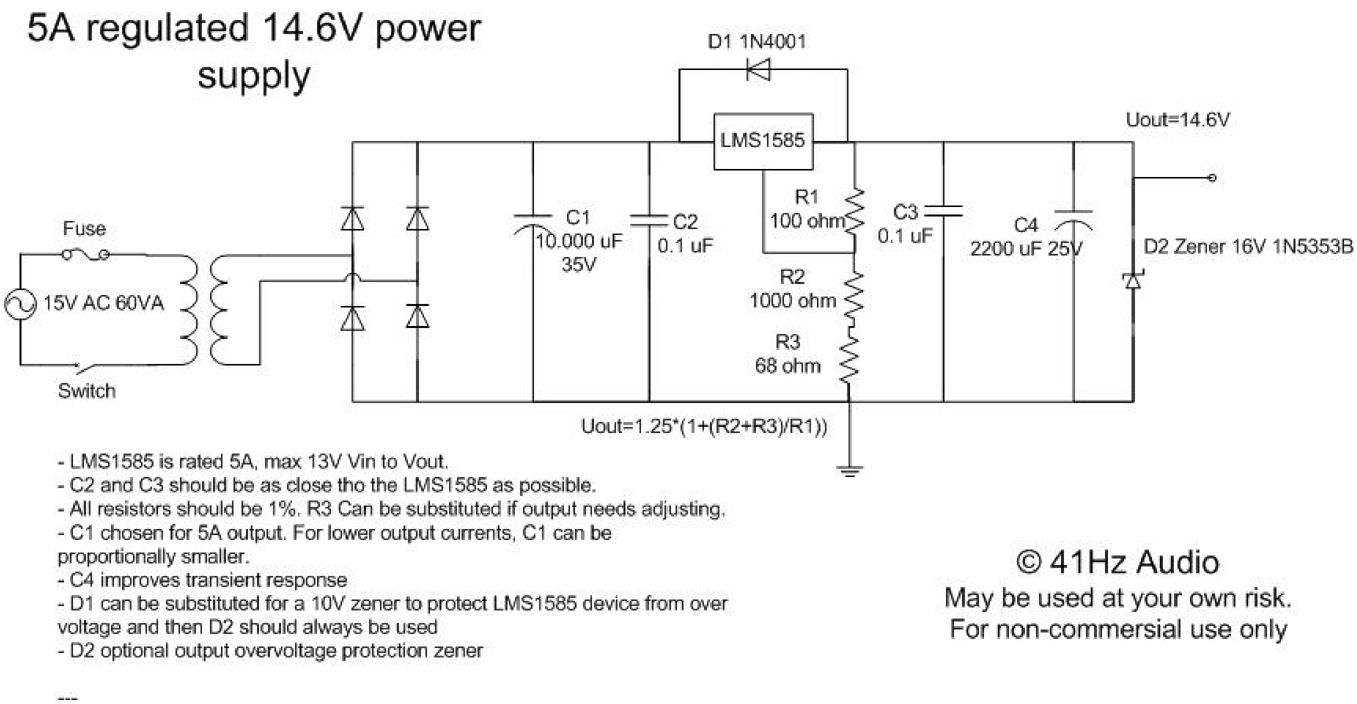
For an un-stabilized power supply, large bulk capacitors would take up load variations and filter out ripple from the rectifier diodes. Again; the bigger the better. But there is probably not that much benefit in using more than 20.000 uF. But 16V capacitors are relatively small and cheap so you can add more if you like.

As an example, the power supply rectifier will typically recharge your bulk capacitors every 1/100 s in a 50Hz system. A 10.000 uF capacitor from which you draw 5 amps will decrease 5V in 1/100 s. In reality, it is not quite as bad, because the recharging cycle time is less than 1/100s and you very rarely draw as much as 5A form the supply, but it gives you an indication that you need quite big capacitors in the power supply, to keep the voltage stable. Note that switch mode power supplies provide a DC at 50 kHz or more and therefore, do not need as large capacitors.

For a stabilized supply, you can probably use smaller bulk capacitors. If you have a voltage of about 22-25V in the upstream capacitor, you can accept more than a 5V drop before the voltage after the regulator starts to go down. The upstream capacitor has significant energy reservoir to top up the lower side capacitor. One 10.000 uF on the upstream side of the regulator and one of 2.200 uF downstream to act as a first buffer for the amplifier, should be sufficient. But as always, the bigger the better.

Design

Here is a design for stabilized power supply suitable for the AMP32. It is based on a LMS1585, 5 amp regulator. The LMS1585 is a fast response, low dropout regulator in a cost efficient in a TO-220 package. One weakness of the LMS1585 is that it only withstands a input-output voltage difference of 13V. While this is enough for normal use, the device could be damaged in case the speaker outputs are short circuited. A perhaps even better alternative is the LM1084ADJ which withstands 28V input to output. The LM1084ADJ has a slightly higher dropout (minimum input to output voltage) of 1.3V compared to 1.15V for the LMS11585 but this should not be of importance.



Comments:

A 12VAC transformer may be sufficient, instead of the 15VAC shown in the schematic, and this would decrease the loss in the regulator. This should be verified by measurements. In case a 12V transformer is used; C1 could be a 25V capacitor. Two resistors, R2 and R3 are placed in series, so that the wanted output voltage can be set, using E12 resistor values. These should be metal film 1% resistors. You could also replace R3 with a 100 ohm trimmer if you like, so the voltage can be fine tuned. D1 could be replaced by a 12V zener, to protect the regulator in case the input to output voltage is higher than the rated 13V. In that case, also D2 should be used, to protect the amp from high voltages. D1 and D2 should be rated high enough to withstand currents that make the fuse blow. D1 also protects the regulator from reverse voltage in case the power supply is shorted / C2 is discharged, while C4 is still charged. C1 could be replaced by an equal sum of smaller capacitors in parallel, to get higher total current ripple rating.

Unregulated supply

You could use the schematic above for an unregulated supply, with the following changes:

- The transformer should be 9V AC (check that the low load voltage is not higher than 14.7V!
- These components should be removed: The LMS1585/LM1084 regulator, D1, C2, C3, R1, R2, R3
- The C1 and C4 capacitors should be 16V 10.000 uF or more.