AMP-4-PS Assembly Instructions

DISCLAIMER! It is assumed that you have thorough knowledge about electronics and building electrical and electronic devices. 41Hz.com assumes no responsibility or liability for any injury, loss or damage incurred as a result of any use or reliance upon the information and material contained within or downloaded from its website.

Introduction

Thanks for selecting the AMP-4-PS kit from 41Hz Audio, good choice.
This document will help you build the kit.
It has NOT been updated for revision 1.1 boards yet, but in progress.

Needed documents, available under the Downloads tab in your Shop account.

- The schematics
- BOM (Bill Of Materals)
- The Tripath chip datasheet for your kit

Considerations before building

There are a few things you should consider before building, because they may influence your selection of component values.

- On the board there are two signal input capacitors C-in, C100 + C200.
  These are required, as the amplifier is internally biased to about +2.5V. Two 3.3uF Panasonic FC electrolytic capacitors and two 1uF polyester capacitors are provided with the kit, you choose which to use.
  The board provides space for RM 2.54 (100 mil) and RM5 (200 mil) and RM7.5 (300 mil) lead spacing capacitors, in case you want to fit other input capacitors.
  The footprint of the caps can be 12mm long, 7mm wide. The input capacitors form a high pass filter together with the input resistor R-in.
  The cutoff frequency is F=1/(2*pi* R-in*C-in) For example, with R-in = 20kΩ and C-in = 3.3uF, the cutoff frequency is F=1/(2*pi*20000*0.0000033) ≈ 3Hz.

  The cutoff frequency is best kept at least two octaves below the lowest frequency

expected.
NOTE
That a big input capacitor may contribute to start-up thumps/plops. Smaller input capacitors can be used at the expense of low frequency damping. Large caps are believed by some to have less positive high frequency characteristics.
If there is a separate woofer in your system, you could use input capacitors with values below 1uF.

- The amplifier input stage, in the Tripath-TC200x chip, is of the operational amplifier type.
  The maximum possible voltage the input stage should handle is about 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*R-feedback/R-in V/V.
  The minus sign is due to the fact that the input stage is inverting. On the board, R21+R29 are the R-in and R23+R31 are the R-feedback.
  With the kit, there are four 20/22kΩ (bag 3) and four 49.9kΩ (bag 2) resistors. With these resistor values, you can choose one of three different input sensitivities as shown in table 1 below.

  If you use other input resistors they should be of a low noise (metal film) type.

R-in R21, 29	*R-feedback R23, 31*	*Input Gain*	*Suitable signal source*
20 kΩ	49.9 kΩ	-2.5	Direct connection of portable MP3/CD player with built in volume control or a volume pot in the power amp.
20 kΩ	20 kΩ	-1	General use
49.9 kΩ	20 kΩ	-0.4	(Pro) preamplifier with fairly high output signal

Table 1. *Gain setting recommendations*

- 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 kΩ 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.
  In that case increase the power amp input gain.
- The chip has a mute function, which disables outputs. If you use a power on/off switch, you may permanently close the mute jumper.
- 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 then better to unsolder/cut the “other” end of the cable or use a board connector.
  Also, try to label the cables, will be very helpful while building and future.

- Power supply.
  For testing at low power any 20-25V supply capable of delivering about 200mA should work.
  The board has a rectifier, two bulk capacitors and two chip close-up capacitors. For testing, a fuse of 0.5A after the transformer / power supply is recommended.
  For final use, the transformer/power supply would normally determine the fuse rating. The fuse should not be higher rated than recommended for the transformer. Fuses on the primary side may need to be quite large and slow blow type, to allow the transformer start-up current.

Building/Assembly instructions

IMPORTANT Components packaged in a shielded, aluminized bag should be considered ESD sensitive and should be handled with ESD care. 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.

On rev.10, R55 and R58 were not listed on the BOM: fixed on 1.1.
Also, C59 was on the BOM of 1.0, but is not on 1.1 (not needed).
(C59 was a 100nF cap across C50, to be put flat on bottom)

In the BOM, there is in Bag-2 a 49.9k resistor input gain resistor named 'R13', this should be 'R31', just like the normal 20/22k values in Bag-3.

Any other quirks in rev. 1.1 BOM+Schematic we'll try to note in AI.
Read AI carefully and completely. TWICE

- Start by mounting all the surface mount capacitors + resistors and the two Tripath chips.
  Make sure the TP2050 power-stage chip has it's casing flat against the PCB, this will avoid mechanical stress when mounting the heatsink/case against the bottom of the PCB/chip.
  Do not mount the small SOT23 transistors yet.
- Complete the +9Vdc section which is a pre-regulator for the +5Vdc.
  The components needed are: R6, C50, C51, Q10, R56, R57 and L5.
  On early boards, the positions of R57 and R56 were reversed on the PCB but corrected in the components placement pictures later in this document.
  On currently shipped boards, the silkscreen on the PCB is corrected.
- Solder D99 (rectifier) in place.
  (or possibly not, if you use DC power supply and don't want the voltage drop of about 1V)
- Now you can do the first test, by connecting power to J2.
  Make sure the J5 (5Vdc) and jumper J10 (main power) are open.
  This ensures the chips are not subject to any voltage yet. You should have around 9V out from Q10. Note the output of Q10 is also on the heat sink of Q10. Save the cut off ends from Q10 for future use.
- When +9Vdc is OK, continue with the +5Vdc section.
  It is C91, Q2, C110, C112, R110, R111, R59.
- You can now test the +5V.
  J5 and J10 should still be open. 5V can be measured on J4, pin 7 to pin 2/3 Agnd or J3, pin 4.
  It is not critical that +5V is exactly 5.00V but it should be stable and within 4.9V and

5.1V.

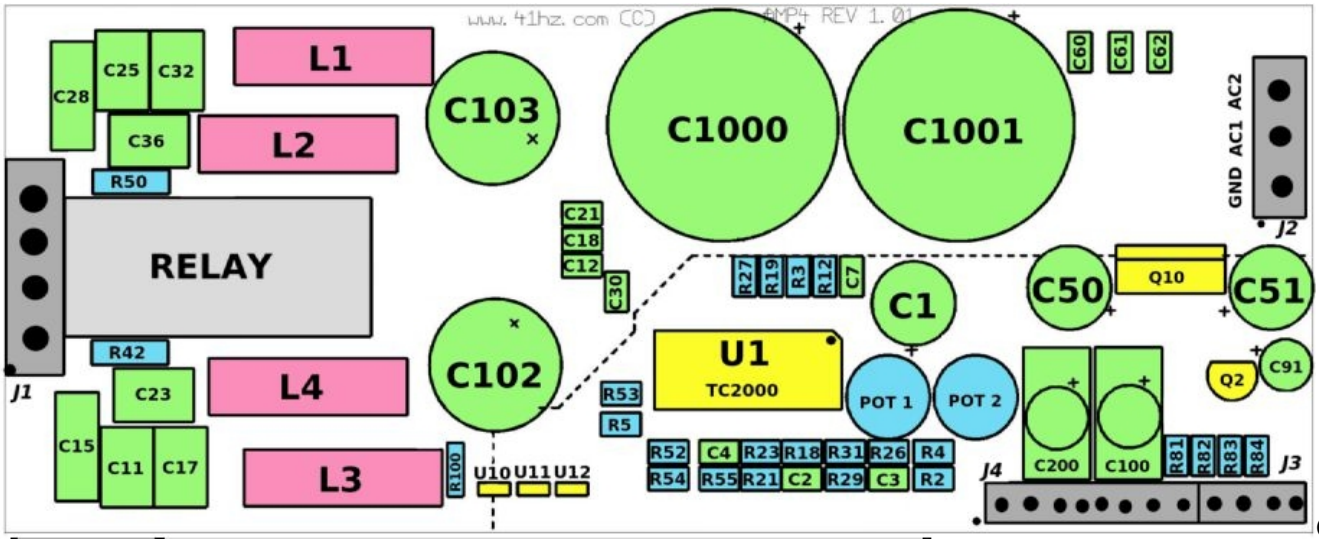
- Continue with all the hole-mount components, except the big bulk capacitors C1000 + C1001; do those in step 23.
- Solder D1 in place.
- Solder the small voltage supervisor U5 in place.
  If you have an adjustable voltage source, you can check the voltage supervisor by monitoring J4 pin 5.
  When increasing the supply voltage from zero, this pin should follow +5V up to about 4.75V and then drop to near 0. The MUTE of the amp (J4 pin 6) is connected to J4 pin 5, optionally via an external switch. The amp is un-muted when J4 pin 6 is low.
  The voltage supervisor thereby prevents muting before +5V has reached at least 4.75V.
- Wind the toroid inductors.
  Related article under Tech info.. You should use 53 turns of 0.6 mm wire. You should be able to do the 53 turns of wire before coming back to the starting point, without overlapping.
  If you have an inductance meter, measure the inductance. It should be 15uH. If you do not have an inductance meter, count the turns.
  When done, fix then in place with a tyrap or a drop of quick drying paint (nail-polish maybe).
- Now mount the toroids to the PCB.
  Double check that the inductor soldering is absolutely good!
- Solder all the output filter components in place; C11, C15, C17, C23, C25, C28, C32, C36, R42, R50.
  Solder the relay too.
- Mount and solder C102 + C103
  Making sure they are really resting towards the PCB before soldering, so they can not move and work loose.
- Now close the J5 jumper (wire, drop of solder).
  Alternatively, you may solder two wires there, so you can measure the current before closing it.
  Turn on the power and make sure nothing is running hot. The chip will draw less than 1mA when muted.
- Close the J10 jumper by soldering a piece of flattened wire, as long as the pads are wide (or a drop of solder). The cut off ends from Q10 will work fine.
- Connect the power
- Check that it is not drawing more than about 100mA. Turn off and continue work.
- Solder the trimmers POT-1/2=R13 + R16. Then the input capacitors C100 + C200 (bag1 or alternative in bag5).
- Solder R100 in place.
  Note on R100; The resistor value should be such that the voltage over the relay coil is between 20 and 24V.
  For voltages up to 25V use a wire/zero ohm jumper. 100Ω (bag5) and 270Ω (bag6) resistors are provided. You can use any standard resistor, rated at least 0.3W. The coil voltage is nominally 24V.
  The relay will close at around 17V and open again below 10V. Above 25V over the coil generates excessive heat in the coil, but the relay will not be damaged by voltages up to at least 35V for short periods.
  With voltages below 18V the relay may not close reliably.
- Turn on power and verify that the relay is closing by temporarily connecting a side of

R100 to ground.

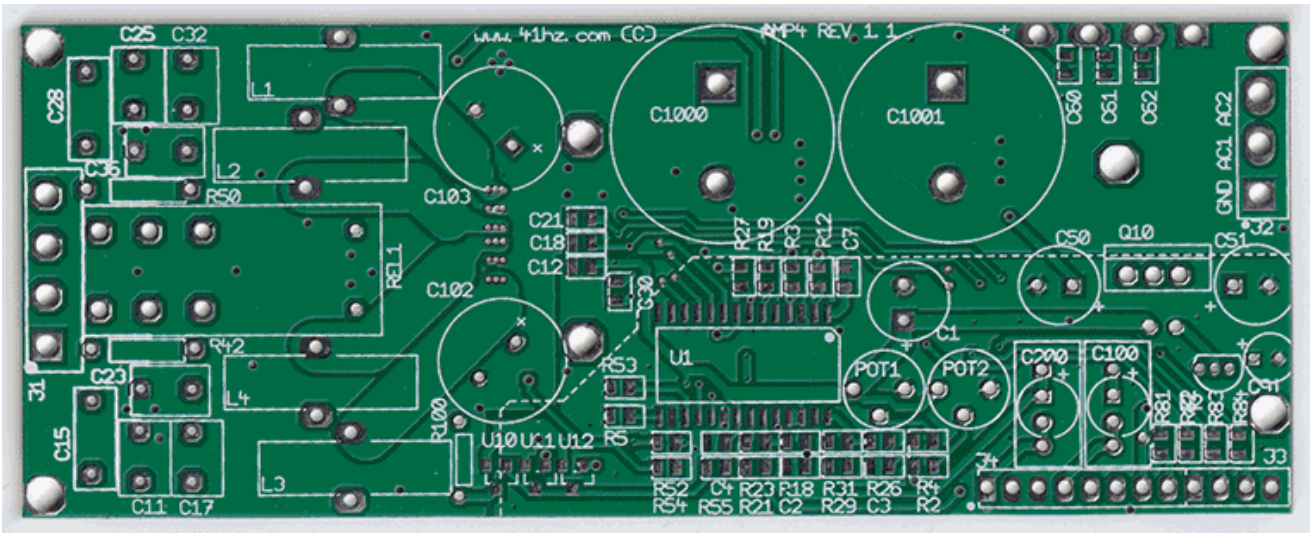
- Solder all the small SOT23 transistors in place.
  U10, U11, U12 work the relay while Q4, Q5, Q6 drive status LEDs connected to J3.
- Turn on the power, un-mute the amp by closing J4 pin 5+6.
  These pins are normally connected to a switch. Verify that the HMUTE signal on J4 pin 8 goes from high to low when the amp is un-muted.
  The HMUTE signal can drive a red LED (positive(An) side to J4/pin 8, negative(Ka) to AGND) directly, but not a blue LED, which requires much more power.
- Now mount the big bulk capacitors C1000+C1001 (you could mount them flat, one on either side with thick wire, if you use a separate appropiately sized heatsink, not if you mount the amp-bottom on a flat surface/casing), if all is OK adjust the POT1 and POT2 (R13, R16) DC-offset trimmers until the voltage on the speaker outputs is 0-10mVdc, max 25mV (100-400mV is NOT acceptable).
- Connect speakers and signal source and...
- Enjoy the music!

Component placement

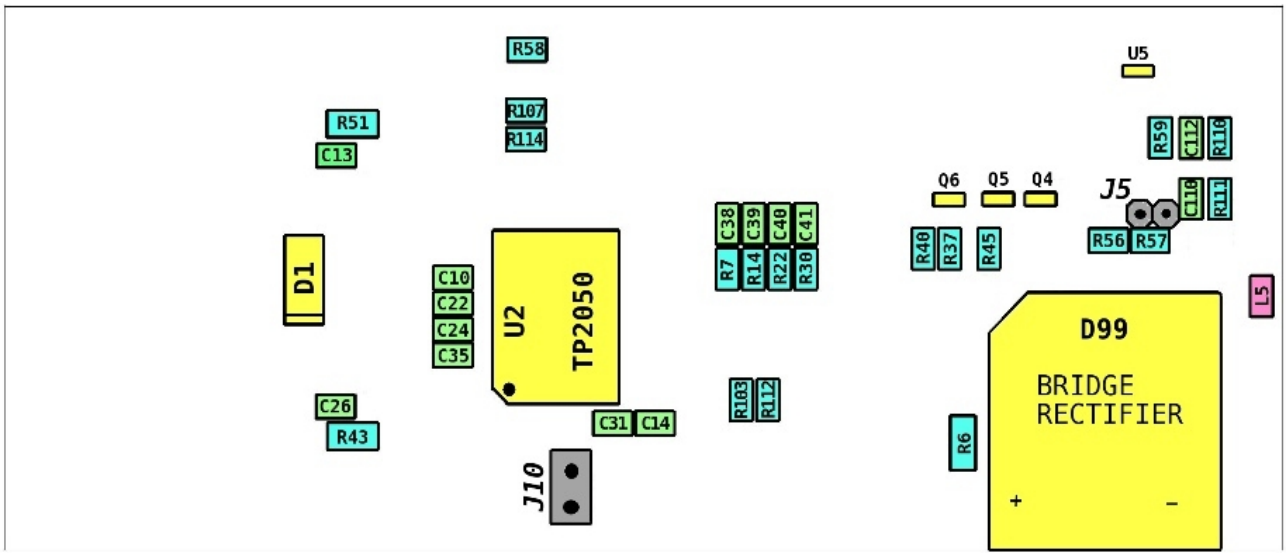
Top view



REAL PCB:

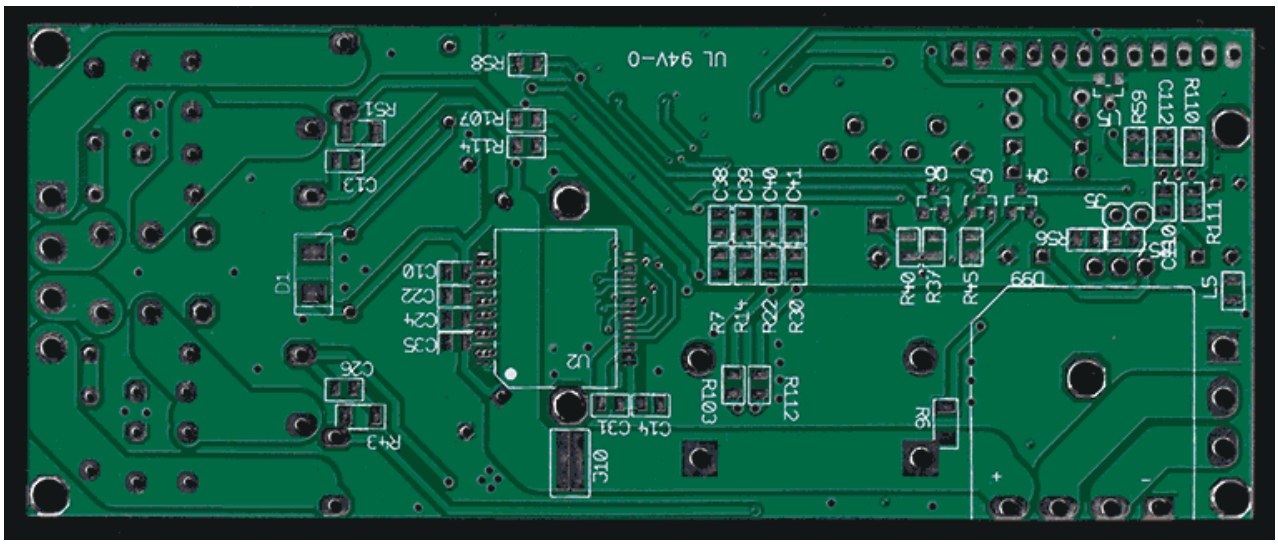


Bottom view



Clic

REAL PCB:



Connections

J1 = Speaker connector (Pin 1 = rectangular pad and dot mark)

1 Output 1 +
2 Output 1 -
3 Output 2 +
4 Output 2 -

Note: Outputs are bridged (BTL) and must NOT be connected to ground

J2 = Power input connector (Pin 1 = rectangular pad and dot mark)

1 P-Ground (might be connected to chassis through TP2050-heatslug)
2 AC1
3 AC2

NOTE
Transformer should not be connected to Ground, but to AC1 and AC2 only (AMP-4 is single supply, no 'center-tap' needed). If a DC source is used, connect negative to pin 1 and positive to pin 2 or 3 (if you have D99, the rectifier bridge, mounted, there will be a voltage drop of about 1V).

J3 = LED/status connector. (Pin 1 = rectangular pad, towards J4)

1	Overload LED output	ON/high if input signal is too high (from Ovrldb signal)
2	Fault LED output	ON/high if power stage failure (over-temperature / overcurrent etc)
3	Overtemperature LED output	If ON/high: warns if the power stage is near over-temperature shutdown
4	AGND	Analogue ground/return for LEDs

NOTE
The LED outputs are driven by Q4,Q5,Q6 from the +5V. The FETs can drive up to 100 mA each but the +5V regulator delivers 100mA max and the Tripath chip use about 80 mA, which leaves you 20mA. Three low-power LED's are provided.
LED voltage/current can be set by changing R81, R82, R83.
(four small LEDs are in the kit)

J4 = Input connector (Pin 1 = rectangular pad and dot mark), MUTE status

1	Input 2	Signal/Source input 2/Right
2	AGND	Analog Ground
3	AGND	Analog Ground
4	Input 1	Signal/Source input 1/Left
5	MUTE REF	From supervisor chip, normally connected to pin 6.
6	MUTE	Normally connected to pin 5, AMP mutes if forced high, or un-muted if forced low, with a switch, then no jumper on pin 5-6
7	+5V	(for MUTE pin or external devices)
8	HMUTE	Output high indicates amp is muted by mute pin or TC2000 chip fault detected. Output low = amp un-muted. This pin can drive a LED (low-power LED provided) directly. The HMUTE signal also controls the relay via a drive FET.

J5 = 5Vdc connect (on underside PCB) Jumper J5 for normal operation

1	+5Vdc
2	To TC2001

J10 = Main power connect (max 32Vdc) (on underside PCB) Jumper J10 for normal operation

1	Upto 32Vdc
2	To TP2050

Minimum requirements to build a working amplifier
There are some essential parts needed to turn this kit into a working amplifier:

- A suitable power supply, like.a toroid-22Vac or smps-30Vdc, 100-150VA (PCB has

PS so it provides rectifier, bulk caps and an 5V regulator on-board)
- Heat sink, connectors

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. The main source of heat on the board is the Tripath TP2050 chip. In most cases, if you mount the board and chip to a metal amplifier casing, it is sufficient to cool the chip. The Tripath chip does not need to be insulated, as the back of the chip is internally connected to ground, but if casing is earthed so will be your amp. Check earthing article and decide.
- Hookup wire. I recommend soldering connection wires to the board. Optionally you can fit a 4 terminal screw/solder terminals with 2.54 mm spacing for the inputs, a four terminal 5.08 mm connector for outputs and a tree terminal 5.08 mm connector for power.
- Mute/un-mute switch or jumper. Preferably wire this to a switch on your panel. Optionally use a 2.54 mm jumper (50 mil) on the board. AMP4 has a voltage supervisor that disables the amp until the +5V system is stable so turn-on thumps are to a large extent eliminated.
- Transformer, power switch and fuse. The maximum voltage out of the board is 32V, limited by bulk and chip close-up capacitors which are rated 35V. A min. 100VA power rating of the transformer is recommended. Use a fuse size as recommended by the transformer manufacturer. Use a good quality mains switch.

The boards for AMP-4-PS are double weight, double sided copper.
Even if the PCB and components are small, quite a powerful soldering iron is helpful. Especially components and pads connected to the ground plane require significant heating. A temperature controlled 50W soldering iron is recommended. At the same time, applying excessive heat may damage the board, causing the copper leads to lift.
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 both SMT and hole-mounted components is available under Technical-Info

Testing and Troubleshooting

+5Vdc section
The 5V regulator Q2 is current limited to nominally 100 mA, and if you try to draw more, the voltage from Q2 it will simply drop until the current limit can be kept. If the regulator gives +5V with J5 in place but the voltage drops when J5 is closed, this suggests there is a short or something fried in the +5V paths.

The over-current protection limiting level of 100 mA is the main reason why a TO-92 size regulator is used on this board and not the more common TO220 size, which usually have a limiting at 500mA or more. The AMP4 +5V section components need a total of about 80mA.

Rail voltage sensing
There is a voltage sensing in the TC2000 chip. The thresholds are set with R52 and R54. If R52 is 22K and R54 is empty, the sensing is disabled, or actually just sensing the +5V. Then the amp may work down to 10V. But the relay will need about 17V to close, so you will not get any sound on the outputs unless you fix that somehow.

If R52 and R54 are in place the voltage sensing is active, sensing the positive rail. The voltage limits are then from 17 to 32.5V with 200K and 19V to 35V with 220K. You can not set the upper and lower limits independently of each other. Details of how to calculate the limits can be found in the TK2050 chipset datasheet from Tripath.

R53 is there to disable the negative rail sensing, as this amp has no negative rail. (single supply)

Note that the over-voltage limit sensing gives no hard protection! It just gives an indication that the voltage is out of bounds.

Troubleshooting
Read article under Tech Info, Building Kits.

AMP4 tweaks + tips

- You MIGHT want to replace the TP2050 with a STA517B which can take higher voltages with higher currents, making the AMP-4-PS cope much better with 4 ohm and at the max.voltage of 32Vdc. Make SURE you have a good heatsink (preferably with a copper plate) to drain off extra heat! Use good quality heat-paste and check if the heatsink is flat on the chip! (only about 1 square cm of metal on chip here!!)
- Some thicker wire on toroids might be useful too in that case (4 ohm+max voltage)
- PCB and other parts are NOT designed for sta517b power, no guarentees!!
- With an sta517b, power source should be increased from 100-150VA to 200-400VA

AMP4 revision history

- 1.0
- 1.01 Not covered in this AI yet.