

AMP2-T Assembly Instructions

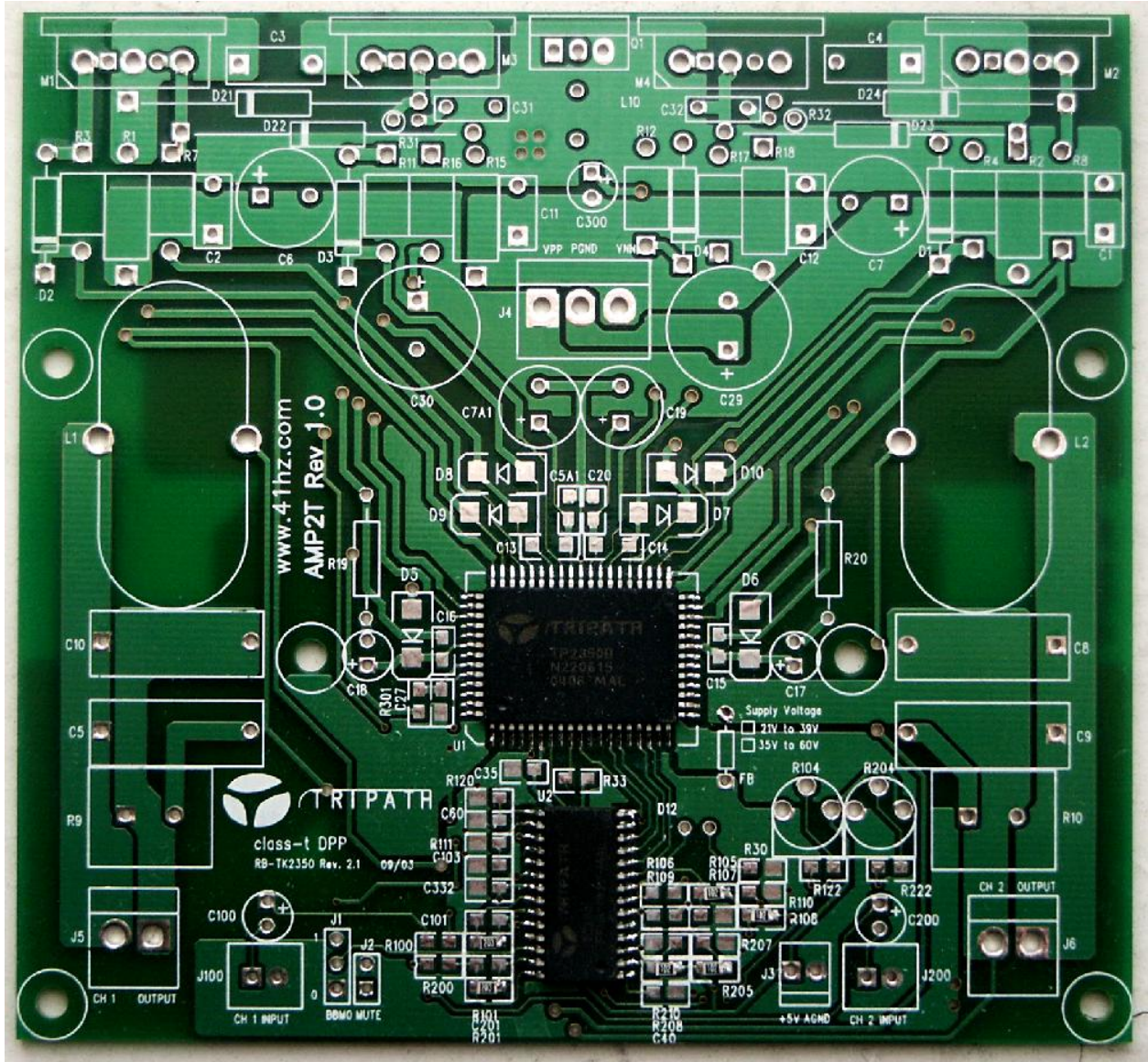


TABLE OF CONTENT

Assembly instructions for AMP2.....	3
Considerations before you start	3
Tools needed	4
The PCB, a summary	4
Components not included in the kit	6
Mounting the components.....	7
Trimming and testing.....	10
APPENDIX 1: Bill Of Materials (BOM)	16
BOM by component symbol / component name	17
APPENDIX 2: Fine tuning the AMP2.....	20
Input impedance, input gain.....	20
Input capacitors.....	21
Modulator gain.....	22
Current limiting.....	22
Voltage limiting	23
APPENDIX 3: Mounting the TP2350B chip.....	24
Soldering the heat slug.....	24
APPENDIX 4: Connector Pinouts	25
APPENDIX 5: Power supply board for the AMP2 amplifier.....	25

The AMP2-T is an advanced high power amplifier. Building this amplifier is not for “beginners”. It is assumed that you have thorough knowledge about electronics and building electrical and electronic devices.

WARNING: The voltages and currents involved can be lethal if not handled properly. If you do not have sufficient knowledge, do not proceed in building and using this kit. 41hz Audio can in no way be held responsible for the consequences of the use of the kit.

Assembly instructions for AMP2

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

On delivery, check that all components have been included. If something is missing, let us know immediately. A bill of material (BOM) should have been included with your delivery and is also included here, as APPENDIX 1

Download the data sheet for the Tripath TK2350 reference board, as the schematic and layout of AMP2-T is identical. However, component specifications and values may differ. Generally, better components are delivered with the AMP2T kit than specified for the reference board. The data sheet for the TK2350 is another useful source of information.

Considerations before you start

The AMP2 kit is designed for certain operating conditions. Depending on the way you will use it, there will some considerations are left to you. The AMP2 kit is intended for supply voltages of about +/- 40V to +/- 50V and four ohm or eight ohm speaker loads. With eight ohm loads, +/- 60V supplies can be used, with only minor changes. Power supply voltages or impedances outside this range may require changing of some components. If you want to use the AMP2 board with 4 ohm loads and voltages higher than +/-50 and thereby very high power outputs, you may also need to update some components. Also for lower voltages, you should change a few component values. With the supplied components you will have the following:

- Current limiting is set to 10A. You may want to increase or decrease this depending on your application.
- Voltage limiting is set to about +/-33 to +/-70 V. You may want to change this. Note that supply voltages between +/-40 to +/- 60V are still recommended with the components in the AMP2 kit.
- Input sensitivity. With the supplied components the amplifier is optimized for a nominal average signal of 1.23V RMS at full power. This is the standard signal level for most pro audio equipment and some HiFi preamplifiers. Domestic audio devices and portable players without a preamplifier typically have a lower signal level and for those, a higher input sensitivity setting in AMP2 may be required.
- Modulator gain (power stage gain) is selected so that the amplifier will start clipping at 2V RMS signal input with a +/-45V supply and the suggested input sensitivity. You may want to fine tune this.

Please read more about how to fine tune the AMP2 kit in APPENDIX 2.

The driver chip TP2350B will dissipate some heat and must be kept cool. The board has mounting holes for mounting a small heat sink over the chip. The bottom of the chip has a heat slug. Heat transfer from the slug can be enhanced by heat transfer paste or by soldering the slug to the board.

If you are not familiar with soldering surface mount components, it is recommended that you do some test soldering on a separate scrap piece of material. Preferably buy some simpler kit or components and practice on these.

Tools needed

Assembly of the kits requires the usual set of electronics working tools; soldering iron, wire cutter etc.

The boards for AMP2 are *triple* weight, double sided copper. This requires quite a powerful soldering iron to heat. Especially the components and pads that are connected to the ground plane require substantial heating. A 50W soldering iron is the very minimum recommended. Preheating the board to about 100°C will help a lot.

If the heat slug of the TP2350B chip is to be soldered to the board (typically for AMPs used at very high power) you will also need access to a solder oven or similar (more about this in APPENDIX 3). Some information on how to solder SMT components is available on the web site.

The PCB, a summary

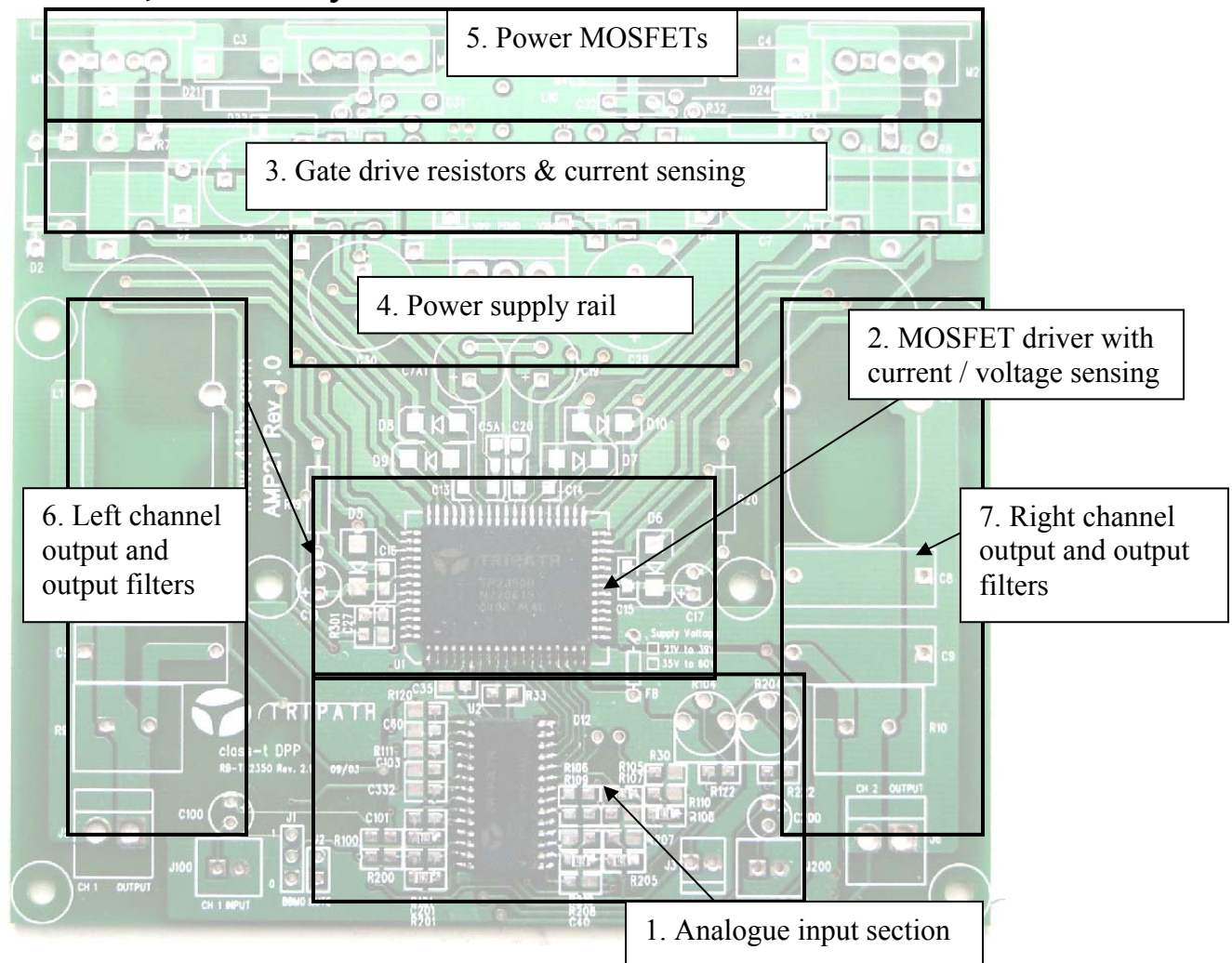


Figure 1: Top view of the PCB

The numbers refer to the sections shown in

Figure 1: Top view of the PCB.

1. The main component of the input section is one of the two chips in the TK2350 chipset; the TC2001 signal modulator. This chip sees only low voltages 0 and +5V. It translates analogue inputs to a digital signal train. Input and feedback resistors set the gain of the input operational amplifiers in the TC2001 chip. Input capacitors block DC signal to and from the chip. There are also offset trimmers and timing jumpers here. There are connectors for low level signals to and from the board. The TC2001 chip modulator generates binary logic level outputs that control the TP2350 MOSFET driver chip. The driver frequency is 600 kHz at idle and is a spread spectrum type signal varying from about 200 kHz to 1.2 MHz.
2. The TP2350B driver translates the logic signals from the modulator chip into level shifted 10 volt high current pulses required to switch the power MOSFETs gates on and off. The TP2350B chip also has voltage and current sensing inputs that can shut off the gate drivers if the correct conditions are not met. The chip controls external power MOSFETs. These are basically voltage controlled devices. But in reality, the gates of the MOSFETs have a capacitance and a significant current pulse is required to switch the MOSFETs on and off at high frequency. The TP2350 chip may get hot and therefore it

has a cooling slug that helps transporting heat to the PCB or a heat sink. There are mounting holes through the PCB for optionally using an external heat sink on top of the chip.

3. Gate drive resistors and protection diodes. These are between the driver chip and the FETs and control the switching speed and indirectly also the heat dissipated by the driver chip. Also, the current sense resistors are here.
4. This is the main power supply rail with positive, ground and negative supplies. The rail has high ripple, low ESR, bulk capacitors that the MOSFETs draw current from. There are also decoupling capacitors and protection diodes.
5. This is where the power switching MOSFETs are. There are normally two MOSFETs for each side; one for the negative side and one for the positive side. Most users will probably use only the standard MOSFETs. Each of the four FETs has a gate drive resistor and protection diode. Between the power rail and the MOSFETs there are current sense resistors for over current detection / protection. These shut down the output in case the current is too high, such as if the speaker cables are shorted (don't try it...). The 10V SMPS generator components are also located here. For the low side FETs, the gate drive voltage must be 10V above the negative power rail. The TP2350B chip has a built in 10V SMPS output that drives an external MOSFET and generates this voltage. The VN10 generator has its own inductor and capacitors to smooth out the ripple. Up to 200 mA current is drawn from the negative rail for this.
6. Left channel output section. The output part for each side has filters that remove high frequency patterns from the power switches that could otherwise recombine into audible noise. A second order, two pole filter is used. It consists of a toroid inductor, some high ripple filter capacitors and a snubber resistor. The corner frequency is in the order of 100 kHz. This section also holds the connection points for the loudspeaker leads.
7. The same as section 6 but for the right hand channel.

Components not included in the kit

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

- Heat sink with screws and heat conductive paste to mount the heat sink. One of the great advantages with PWM mode amplifiers is that the heat losses are small. However, this is a high power amplifier and even if only a small percentage of the power is dissipated as heat, the heat can be considerable for a high power / low impedance application.
- Copper wire for the inductors. About 2.5 meters (8 ft) of 1.2 or 1.3 mm (AWG 16) enamel insulation copper wire is needed. "Litz" type multi-strand wire of the same equivalent area can also be used and is recommended as it is much easier to wind and has good high frequency characteristics. However, Litz wire is expensive and a bit hard to source.

- Connection terminals and connection wire. I recommend soldering wires to the board unless you plan to connect and disconnect them a lot. I also recommend you use the wire you plan to use in your final application right from the start. The PCB leads and wire required for connecting the power supply and speakers are heavy duty. Soldering them requires substantial heat and with repeated soldering you risk damaging the PCB. There is much debate in the audio world about connectors and IMO the best connector is none at all; a well made solder connection is better than the best connector. Unless you need to disconnect them frequently.
- Wire jumpers. Depending on how you configure the board you will need some wire jumpers.

Mounting the components

1. First, decide how to solder the TP2350B chip. If you are soldering the heat slug to the board this **must** be done first, or at the same time as soldering the other components in an oven. You should read more in APPENDIX 3 about this.
2. Solder all surface mount components on the bottom side. A picture of how the components are placed is included below in *Figure 2: Components, bottom view*.
3. Mount the axial lead diodes D12, D22, D23, and D24 on the bottom side. Cut the leads snugly on the top side of the board (and always do the final soldering after cutting the wires).
4. Turn the board over and solder all surface mount components on the top side. A picture of the component placement can be found in *Figure 3: Components top view*. The component numbers are better seen in this figure than on the board. Most surface mount components are placed around the U2 chip. Four diodes and four 0.1 uF decoupling capacitors are placed above the U1 chip.
 - a. J1 is a jumper wires that set the BBM (Break Before Make) timing. Normally set the BBM to 80 nS by placing the jumpers as shown in *Figure 4*. Shorter BBM timing may cause current shoot-through and is not recommended. 120 nS BBM timing can be used for heavy duty (woofer amps) for increased shoot through margin but may increase distortion in the > 2 kHz range.
 - b. Leave R33 open. It's connected to an undocumented test pin. Tripath just state it should be left open.
5. Solder the L3 inductor bead. This component connects the analogue signal ground plane around the TC2001 chip to the rail power ground plane and dampens HF power ground plane fluctuations from disturbing the analogue section.
6. Solder the axial resistors and diodes.
 - a. R31 and R32 are placed vertically, with one lead doubling back.

- b. The leads of the current sense resistors R1, R2, R15, and R18. These should preferably be mounted with some standoff that helps keeping them cool. The whole supply rail current will pass through these resistors.
 - c. Mount the gate diodes and resistors D1, D2, D3, D4, R3, R4, R11 and R12.
 - d. Mount the other axial resistors.
7. Solder the offset trim potentiometers R104 and R204
8. Before connecting the final components, now is the time do a first test of the board. Do not proceed further until the test described below, has completed successfully. Please look at APPENDIX 4 for the connector pin assignments. Testing:
- a. Check the resistance on the power supply connector J1. It should be about 1 Mohm between positive and ground, about 1.3 Mohm between negative and ground and about 2 Mohm between positive and negative rail.
 - b. The +5V input should measure something like 0.5 kohm to ground.
 - c. The +5V supply should be well stabilized and fused, (before the voltage regulator), at 200mA. Use a shielded cable for connecting the +5V and connect + to pin 8 and negative to pin 7 of J2. **The +5 supply should never go above +6V, or be connected with wrong polarity, as this will damage the Tripath chipset.**
Do not turn on the power yet.
 - i. Note, the +5V is used for both the analogue preamp section of the Tripath chip and also for the logic level signals in the chipset. It is essential that the +5V supply is quiet / well stabilized. Using shielded cable for hooking up the +5V prevents feedback of high frequency EMI from the power amp into the analogue section.
 - d. Connect the rail positive, ground and negative leads to J1. **Double check the polarity! Connecting the wrong polarity will damage components.** The amp may draw around 100 mA on the positive rail and around 200-250 mA on the negative rail at “transistor radio volume”. *Do not turn on the power yet.*
 - i. As a protection you should use current limiting resistors in series with the power supply lines. If the amp draws unexpectedly high current, the voltage will drop below the level where under-voltage limit shuts off the amp. Suitable resistor values can be 68 to 100 ohms, 3W for the positive rail and 33 ohms to 47 ohm 3W for the negative rail. With the default components, the amp will mute if the supply voltage drops below about +/- 34V. *Do not turn on the power yet.*
 - ii. Alternatively, the amp could be protected with a 100mA fuse, placed on the AC side of the supply. This way, both rails de-power in the case of failure. The down side is that the energy in the supply capacitors may discharge through the amp, even if the fuse has blown. Other fuse values may be required, depending on your transformer and capacitors. *Do not turn on the power yet.*
 - e. The onboard LED is connected via R90 (2.2K). Optionally, instead connect an external LED from pin HMUTE (J2 pin 9), for example on the front panel of your amp. An external LED needs its own 2.2 kΩ resistor to ground. A LED helps a lot when testing.
 - i. Do not connect two LEDs at the same time, as this may draw too much current from the HMUTE output

- ii. The HMUTE output can be used to control speaker output relay, via a driver circuit. The HMUTE will go high (+5V) in case the amp is muted or in case of over current, over/under voltage and other errors. Shorted outputs or shorted output transistors would also trig the over current sensing. If no errors are detected and the amp is unmuted HMUTE will be low (0V).
 - f. Connect the MUTE input (J2 pin 11) to +5V (J2 pin 8) in a manner that the MUTE signal can be shifted from +5V to ground (J2 pin 5, 6 or 7).
 - g. Switch on the +5V supply, *but not the main rail voltage yet*. The +5V should draw about 25 mA. If a LED is connected to J2 pin 9, it should light up. If a LED is not connected, check if there is +5V on the J2 pin 9.
 - h. Now, switch on the rail (main) power supply. It should draw a current of about 1mA.
 - i. Shifting the MUTE input from +5V to ground will toggle the amplifier from mute to active. Now the amps should go active, and the +5V should draw about 50mA and the LED (or J2 pin 9) should go low/off. If it does not, this indicates the amplifier is still muted, for example because the voltage is too low or high or current is over limits (shorted outputs...) If this happens, shut off the power, discharge the rail capacitors with a suitable resistor. Check all soldering and everything else in the previous paragraphs. *Do not proceed further until this testing stage has passed successfully.*
 - i. Note that under-voltage protection has a 200 mS delay and is non-latching and the amp may “instantly” turn on again when the voltage recovers, and then go on-off-on-off several times per second. In that case, turn the power off!
 - ii. Note with the supplied voltage sense resistors, R112, R113, R114 and R115, the lowest voltage the amplifier can be operated at is about +/-34V. At lower voltages, the amp will *not* leave the mute state.
 - iii. Note also that the current sense (0.01 ohm) resistors must be in place for this test to work.
 - j. When the amplifier is un-muted (LED is turned off), the positive rail should draw a maximum of 40-80mA. The negative rail current may be 150-200 mA.
 - k. If everything looks OK, then power down, remove the current limiting resistors, and proceed as described below.
9. Mount and solder film and ceramic radial lead capacitors.
 - a. There are two types of 0.1uF 250V film capacitors. C3 and C4 are rail to rail mounted to reduce ripple on the DC supplies. These are Panasonic polyester caps in a brownish dipped casing. C8 and C10 are part of the output filter and remove high frequency ripple from the speaker outputs. A 100V rating would be sufficient here but the 250V give a higher high frequency ripple voltage margin.
 - b. There are also four 100V 0.1 uF decoupling capacitors on the power rail; C1, C2, C11 and C12.
10. Mount and electrolytic capacitors. Respect the polarity as marked on the PCBs.
 - a. There are two high ripple, low ESR bulk capacitors; C29 and C30. One on each rail to ground and rated 100V. The C6 and C7 are rail to rail and therefore rated 160V.
 - b. C100 and C200 are the input capacitors. There are 1uF electrolytic Panasonic FC capacitors are supplied in the kit. These are good quality standard capacitors.

- c. There are three low ESR 100uF (150 uF) 35V for the onboard V10 PWM supply. These are the C300, C19 and C71.
 - d. There are two 47uF for the bootstrap circuitry, C17 and C18.
11. Solder the inductor L4 and the Q1 MOSFET for the V10 supply.
 12. Wind the toroid inductors. You should use 29 turns of 1.2 or 1.3 mm enamel insulated wire for the T108-2 toroids included in the kit. The inductance should be 11 uH. Wind as tight as you can. Tight winding minimizes any stray signals. This thick wire is a challenge to wind. Leave a bit of wire so you can pull the toroids snugly to the board.
 - i. For high frequency, high current windings like the main inductors, a good alternative is to use several thinner wires, with the same cross section area, in parallel. A bundle of thin wires is easy to wind and the insulation between strands reduces the “skin effect” and thereby reduces losses. You can buy “Litz” wire which is simply a pre-made multi-strand wire, where each strand is separately insulated with enamel insulation.
 13. Scrape the enamel off the wires where they will be soldered and solder the wires to the PCB. It may be a good idea to glue the toroids to the PCB to prevent the copper wires to break from fatigue in case the PCB is subjected to vibrations. Use a temperature resistant glue as the toroid will be warm at high loads.
 14. Optionally solder board connectors ***J1 for power and J5 and J6 for speakers. Connectors should have a current rating exceeding the currents you expect to have and also exceed the current limit setting you use. The default for the AMP2 kits is 10A. If you do not use connectors, solder the power cabling and speaker output cabling in place.
 15. Connect signal connectors. Again consult the APPENDIX 4 and decide what connections you would like to make.
 - a. Bridging or no bridging? Permanent bridging or an external switch?
 - b. Mute switch? Strongly recommended! If you do not use a switch, you should permanently ground pin 11 of J2
 - c. Hmute indicator? The Hmute (pin 9 of J2) can drive an external LED via a 2 kΩ resistor. The LED will be *on* if the amplifier is muted by pin 11 or due to over voltage or over current limit has been exceeded.
 16. Mount the four MOSFETs in positions M1, M2, M3 and M4. IMPORTANT. Take care that the leads of the MOSFETS do not touch adjoining tracks before soldering them. The legs of the provided MOSFETS have a fairly wide shoulder that may short circuit to adjoining tracks. Therefore the MOSFETS should be lifted by about a millimeter before they are soldered. Make sure they are in line, at the same height and flatly on the same surface. They need to adhere well to the heat sink.
 17. You are now ready to test the amplifier as described below. A heat sink is not required for low power testing but will surely be required for higher power levels.

Trimming and testing

- Put the amplifier to Mute (J2 pin 11 to pin 8)

- Ground the signal inputs to signal ground (connect IN1 and IN2 to AGND on connector J2). Do NOT connect the speakers yet.
- Connect the +5V supply and its ground. A 100 mA fast fuse is recommended. Important; If the +5V supply is higher than 6V the chipset will be permanently damaged. The 5V supply should be well stabilized and filtered. A good option is to use a type 8705 with a 1000 uF before and another 1000 uF after the regulator. Voltage regulators should always be bypassed with a 0.1 uF or similar at input and output, close to the chip. The regulator data sheets usually give suggested schematics for this.
- Connect the power supply. **Double check the polarity of the power supply.** Wrong polarity will permanently destroy some components, including the Tripath chipset. For the first testing, use current limiting resistors as described above.
- Turn on the power.
- When muted and with no speakers connected, the +5V should draw about 20 mA and each main rail should draw about 1mA.
- Switch the MUTE setting from Mute to Awake
- Check the fuses. If they have blown, disconnect the board and check all components and solders.
- With the amp unmated (awake) the +5V should draw about 50mA and the positive rail should draw about 80mA, the negative about 200 mA.
- If the fuses are OK then, adjust the trimmer pots until the speaker output signal is as close to 0V as you can get it. This minimizes turn on/off pops and minimizes DC through the speakers. You should be able to trim the offset to below 10 mV. Do not run the amp for extended periods without speakers or a dummy resistor connected.
- Shut of the power.
- Connect the speaker wires
- Connect a signal source with music.
- Set the volume on the signal source very low
- Turn on the power and check if you get any sound.
- If everything seems OK, switch off power, replace the power supply fuses for larger fuses like 1A and try again at slightly higher volume. (The 5V fuse should remain about 100-200mA).
- 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 outputs, you should have an efficiency of 90-95% with 8 ohm loads and 85-90% with 4 ohm loads. The rest of the power, 5-15%, can be assumed that is dissipated from the MOSFETs. I.e. if you drive the amplifier to 2x150W into 8 ohms you will have 15-30W of heat dissipated. If you drive the amp to 2x300W into 4 ohms you will have 60-90W of heat. Select a heat sink / cooling method that can handle this. Keeping the FET temperature down is important for reliability and lifetime. The MOSFETs **must** be electrically insulated form the heat sink. Using thick pads between the FETs and heat sink is a good idea s this decreases capacitive coupling between FETs and heat sink, and thereby minimizes the EMI that may otherwise be emitted by the heat sink.
- Connect the amplifier to your power supply and change the power supply fuses to the proper value.
- Enjoy!
- Please note:
 - Speaker leads must be grounded to the PCB connectors, not to the power supply. The modulator feedback has separate returns from these connections. The speaker returns are grounded in a star point on the PCB.

- Audio signal sources and the 5V supply must be grounded to J2. This connector is in turn connected via an inductor bead to the star grounding point where the power supply is connected.

If you have any questions, write any questions on the forum on our web site

<http://www.support.41hz.com> , as the questions may be of general interest.

For any questions please do not hesitate to contact us at jan@41hz.com

Feedback is very much appreciated!

Figure 2: Components, bottom view

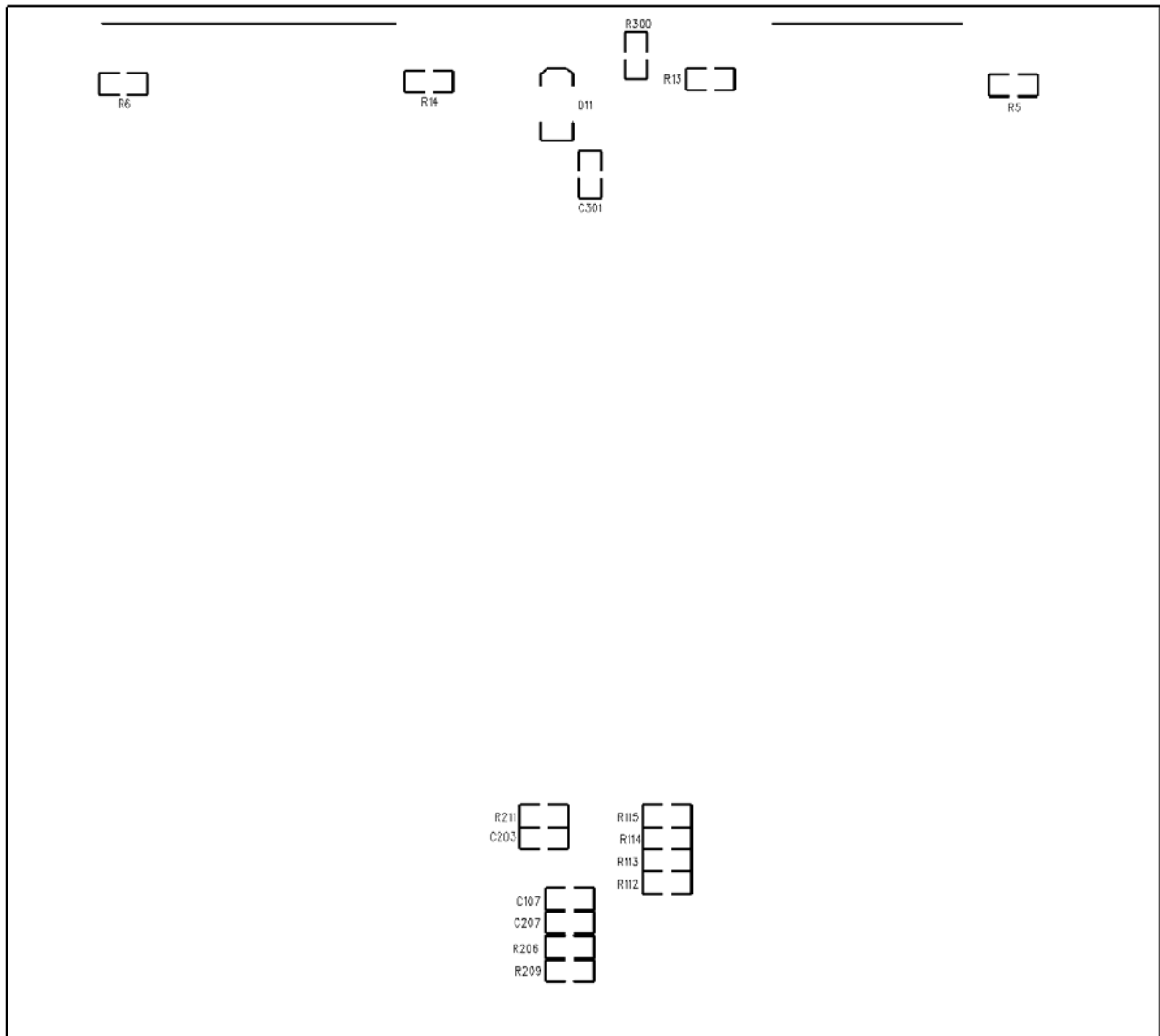


Figure 3: Components top view

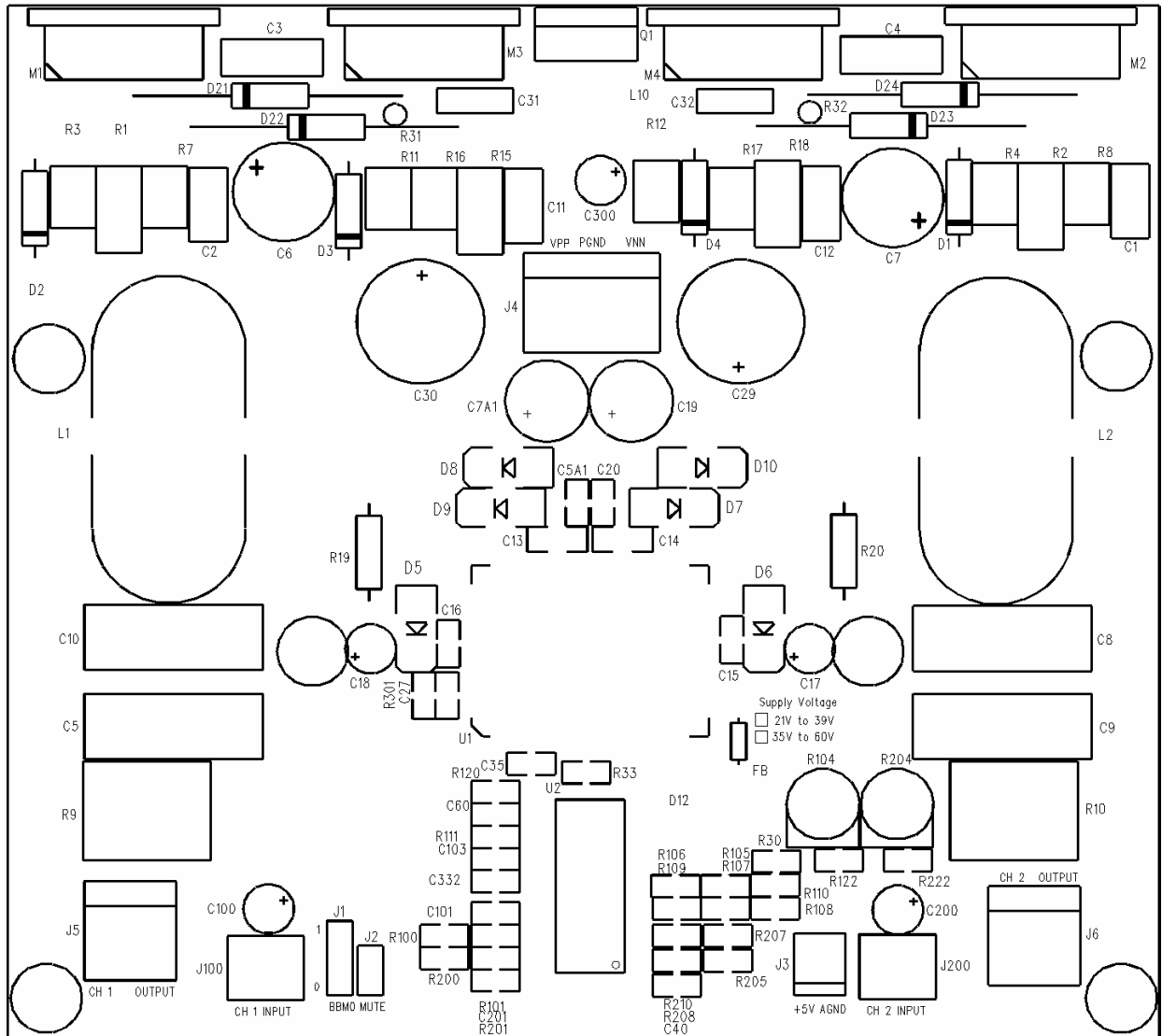


Figure 4: Jumper J1 BBM settings

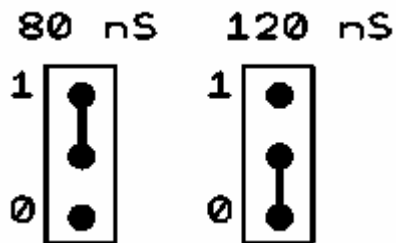
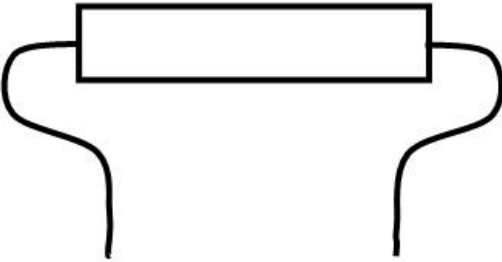


Figure 5: Shape of current sense resistors



APPENDIX 1: Bill Of Materials (BOM) ***

Count	Value	Package
(R) 4	0.01 ohm	Axial 1% 2W
(C) 11	0.1 uF 50V	SMT 0805
(C) 4	0.1 uF 100V ceramic	Radial
(C) 2	0.1 uF 100V (250V)	Radial
(C) 2	0.1 uF 250V	Radial
(C) 2	0.22 uF 100V (250V)	Radial
(R) 4	1.1 K	SMT 0805
(R) 1	1.18 M	SMT 0805
(R) 9	1 K	SMT 0805
(C) 2	1 uF 10V (63V)	Radial
(R) 4	2.2 ohm 1W	Axial
(R) 1	8.2 K	SMT 0805
(R) 2	10 K	SMT 0805
(R) 1	10 ohm	SMT 0805
(R) 6	15 ohm 1W	Axial
(R) 4	15 K	SMT 0805
(R) 5	20 K	SMT 0805
(R) 2	22 ohm 3W	Axial
(C) 2	33 pF 50V	SMT 0805
(C) 2	47 uF 25V	Radial
(R) 2	49.9 k	SMT 0805
(C) 1	100 pF 50V	SMT 0805
(C) 1	100 uF 10V	Radial
(C) 3	100 uF 35V low ESR	Radial
(L) 1	100 uH 2A inductor	Radial
(C) 1	150 pF 50V	SMT 0805
(C) 2	33 uF 160V Low ESR, high ripple	Radial
(R) 2	220 ohm 0.25W	Axial
(C) 2	220 pF 50V (200V)	SMT 0805
(C) 2	220 pF 200V	SMT 0805
(C) 1	270 pF 50V	SMT 0805
(C) 4	220 uF 100V Low ESR, high ripple	Radial
(R) 1	392 K	SMT 0805
(R) 2	422 K	SMT 0805
(R) 4	510 K	SMT 0805
(L) 1	Ferrite bead	Axial
(Q) 1	IRF9510 or MTP12P10	TO220
(M) 4	MOSFET STW34NB20	TO247
(D) 8	MUR120 diode, ultra fast	Axial
(D) 6	MURS120 diode, ultra fast	SMB
(R) 2	Pot 50K	Trimmer
(D) 1	Schottky 1A 35V	SMB
(U) 1	TC2001	SOIC28MIC
(L) 2	Toroid (11uH)	Toriod for winding, T106-2
(U) 1	TP2350B	64PIN-LQFP
(Board) 1	AMP2 PCB	

BOM by component symbol / component name

Symbol	Value	Footprint	Comment
C1	0.1 uF 100V	CAP_RM5_7.2X2.5	Rail decoupling
C2	0.1 uF 100V	CAP_RM5_7.2X2.5	Rail decoupling
C3	0.1 uF 250V	CAP_RM7.5_10X6	Rail-Rail decoupling
C4	0.1 uF 250V	CAP_RM7.5_10X6	Rail-Rail decoupling
C5	0.22 uF 100V	CAP_RM15_20X7	Output filter
C6	33 uF 160V	CAP RADIAL	Rail-rail power supply decoupling
C7	33 uF 160V	CAP RADIAL	Rail-rail power supply decoupling
C8	0.22 uF 100V	CAP_RM15_20X7	Output filter
C9	0.22 uF 100V	CAP_RM15_20X7	Output filter
C10	0.22 uF 100V	CAP_RM15_20X7	Output filter
C11	0.1 uF 100V	CAP_RM5_7.2X2.5	Rail decoupling
C12	0.1 uF 100V	CAP_RM5_7.2X2.5	Rail decoupling
C13	0.1 uF 50V	SMT 1206	Decoupling of feedback
C14	0.1 uF 50V	SMT 1206	Decoupling of feedback
C15	0.1 uF 50V	SMT 0805	Decoupling of boot circuit
C16	0.1 uF 50V	SMT 0805	Decoupling of boot circuit
C17	47 uF 25V	CAP6MM_RAD_RM2.5	Boot circuit cap
C18	47 uF 25V	CAP6MM_RAD_RM2.5	Boot circuit cap
C19	100 uF 35V	CAP8MM_RAD_RM3.5	VN10 decoupling
C20	0.1 uF 50V	SMT 0805	Decoupling of VN10
C27	0.1 uF 50V	SMT 0805	Decoupling of VN10 SMPS feedback
C29	330 uF 100V	CAP RADIAL	Power supply close-up buffering
C30	330 uF 100V	CAP RADIAL	Power supply close-up buffering
C31	220 pF 200V	SMT 0805	Output snubber
C32	220 pF 200V	SMT 0805	Output snubber
C35	0.1 uF 50V	SMT 0805	Decoupling of V5
C40	0.1 uF 50V	SMT 0805	Biascap
C5A1	0.1 uF 50V	SMT 0805	Decoupling of VN10
C60	100 pF 50V	SMT 0805	Decoupling of VREF
C71	100 uF 35V	CAP8MM_RAD_RM3.5	VN10 decoupling
C100	1 uF 10V	CAP_RM15_20X7	Input capacitor
C103	220 pF 50V	SMT 0805	Current set point delay
C107	270 pF 50V	SMT 0805	Feedback delay / switch frequency
C200	1 uF 10V	CAP_RM15_20X7	Input capacitor
C101	33 pF 50V	SMT 0805	Input RF decoupling
C201	33 pF 50V	SMT 0805	Input RF decoupling
C203	220 pF 50V	SMT 0805	Current set point delay
C207	150 pF 50V	SMT 0805	Feedback delay / switch frequency
C300	100 uF 35V	CAP8MM_RAD_RM3.5	VN10 decoupling
C301	0.1 uF 50V	SMT 0805	Decoupling of VN10
C332	0.1 uF 50V	SMT 0805	Decoupling of V5
D1	MUR120	MUR_RM10	Gate discharge diode
D2	MUR120	MUR_RM10	Gate discharge diode
D3	MUR120	MUR_RM10	Gate discharge diode
D4	MUR120	MUR_RM10	Gate discharge diode
D5	MURS120	SMB_DIODE	Boot diode
D6	MURS120	SMB_DIODE	Boot diode
D7	MURS120	SMB_DIODE	Protection of HF feedback

D8	MURS120	SMB_DIODE	Protection of HF feedback
D9	MURS120	SMB_DIODE	Protection of HF feedback
D10	MURS120	SMB_DIODE	Protection of HF feedback
D11	Schottky 1A 100V	SMB_DIODE	VN10 SMPS diode
D21	MUR120	MUR_RM30	Overshoot protection of outputs
D22	MUR120	MUR_RM10	Overshoot protection of outputs
D23	MUR120	MUR_RM10	Overshoot protection of outputs
D24	MUR120	MUR_RM10	Overshoot protection of outputs
L1	Toroid 11uH	T108-2	Output filter inductor
L2	Toroid 11uH	T108-2	Output filter inductor
L3 (FB)	Ferrite bead	R05W	EMI damping analog ground ⇔ ground
L10	100 uH	Inductor	VN10 SMPS inductor
M1	MOSFET	TO220 or TO248	Main FET
M2	MOSFET	TO220 or TO248	Main FET
M3	MOSFET	TO220 or TO248	Main FET
M4	MOSFET	TO220 or TO248	Main FET
Q1	IRF9510	TO220	VN10 FET
R1	0.01 ohm	R2W Ohmite axial 1%	Current sensing
R11	15 ohm	R1W	Gate drive current limiting
R2	0.01 ohm	R2W Ohmite axial 1%	Current sensing
R12	15 ohm	R1W	Gate drive current limiting
R3	15 ohm	R1W	Gate drive current limiting
R4	15 ohm	R1W	Gate drive current limiting
R5	510 K	SMT 0805	Gate ground reference
R6	510 K	SMT 0805	Gate ground reference
R7	2.2 ohm	R1W	Feedback damping
R8	2.2 ohm	R1W	Feedback damping
R9	22 ohm	R3W	Output zobel filter
R10	22 ohm	R3W	Output zobel filter
R13	510 K	SMT 0805	Gate ground reference
R14	510 K	SMT 0805	Gate ground reference
R15	0.01 ohm	R2W Ohmite axial 1%	Current sensing
R16	2.2 ohm	R1W	Feedback damping
R17	2.2 ohm	R1W	Feedback damping
R18	0.01 ohm	R2W Ohmite axial 1%	Current sensing
R19	220 ohm	R05W	Boot circuit current limiting
R20	220 ohm	R05W	Boot circuit current limiting
R22	20 K	SMT 0805	Bridging resistor
R30	2.2 K	SMT 0805	LED current limiting
R31	15 ohm	R1W (vertical)	FET snubber
R32	15 ohm	R1W (vertical)	FET snubber
R100	49.9 k	SMT 0805	Input feedback
R101	20 K	SMT 0805	Input feedback
R104	Pot 50K	TRIM_3_RM5_D6	Trimmer for nulling output offset
R105	1 K	SMT 0805	Feedback & gain setting
R106	15 K	SMT 0805	Feedback & gain setting
R107	1.1 K	SMT 0805	Feedback & gain setting
R108	1 K	SMT 0805	Feedback & gain setting
R109	15 K	SMT 0805	Feedback & gain setting
R110	1.1 K	SMT 0805	Feedback & gain setting
R111	20 K	SMT 0805	Over-current trip set point
R112	422 K	SMT 0805	Voltage limit sensing
R113	422 K	SMT 0805	Voltage limit sensing

R114	1.18 M	SMT 0805	Voltage limit sensing
R115	392 K	SMT 0805	Voltage limit sensing
R120	8.2 K	SMT 0805	REF internal voltage reference
R122	10 K	SMT 0805	Trim pot limiter
R200	49.9 k	SMT 0805	Input feedback
R201	20 K	SMT 0805	Input feedback
R204	Pot 50K	TRIM_3_RM5_D6	Trimmer for nulling output offset
R205	1 K	SMT 0805	Feedback & gain setting
R206	15 K	SMT 0805	Feedback & gain setting
R207	1.1 K	SMT 0805	Feedback & gain setting
R208	1 K	SMT 0805	Feedback & gain setting
R209	15 K	SMT 0805	Feedback & gain setting
R210	1.1 K	SMT 0805	Feedback & gain setting
R211	20 K	SMT 0805	Over-current trip set point
R222	10 K	SMT 0805	Trim pot limiter
R300	10 ohm	SMT 0805	VN10 gate drive current limiting
R301	1 K	SMT 0805	VN10 feedback
R33	-----	SMT0805	Leave open. Internal Tripath test only.
U1	TP2350B	64PIN-LQFP	Tripath FET driver
U2	TC2001	SOIC28MIC	Tripath processor chip

APPENDIX 2: Fine tuning the AMP2

The components included in the AMP2 kit work well for many users. For optimal performance the AMP2 board may be fine tuned to each application. Here is a summary of what to think of.

Input impedance, input gain

The TK2350 can be described as a two stage amplifier. The first is an operational amplifier while the second is the power stage / modulator.

So first we have to select input and gain resistors for the input operational amplifier. The input impedance has an effect on any equipment upstream of the amplifier, such as pre amplifiers. Modern pre-amplifiers are usually capable of driving fairly high currents / low impedance loads. But usually, the THD+N is better if the impedance is not too low. On the other hand very high impedance in the power amplifier will introduce current noise so a compromise has to be found. Values of between 10k Ω and 100K Ω are typical while 20 K Ω to 50 K Ω are most common. Some people prefer carbon thick film resistors for the input and feedback resistors while others prefer thin film or metal film resistors which usually have lower noise. With thick film carbon resistors we could use lower values, say 20 K Ω for low noise while higher values can be used with thin film or metal film, often 50 K Ω .

Most consumer audio equipment like CD players etc have a signal level output of -10dBV which corresponds to 0.316 V RMS. Portable devices like MP3 players, MD and CD players often have lower output to extend battery life. A preamplifier would usually have a gain of about +10 dB giving an output of 0dBV or 0.775 V RMS maximum. Similar levels could be expected from computer soundcards. Professional equipment usually has a nominal +4 dBU maximum output signal which corresponds to 1.23 V RMS.

The TK2350 amplifier chip modulator stage has a maximum input level of about 4V peak to peak, equivalent to 1.41 V RMS. For best performance we should select the gain for the input stage so that at maximum volume (from the preamp) the voltage in the input stage is close to the maximum 4V_{ptp}. Some headroom margin for transient peaks is however required.

The following table gives some examples. For a consumer audio level signal of 0.316 V and select a gain of one (1) for the power amplifier input stage, we would have headroom of 15dB which is a bit much and means the amplifier can not reach its maximum power. If we add a +10dB gain preamplifier we would have headroom of 5 dB which is reasonable. With a pro audio input of 1.23 V and an amplifier gain setting of -0.4 we would have 10dB headroom, which again is reasonable.

	Consumer audio	With +10 dB pre-amp	Pro audio +4dBu	Value in AMP2 kit	Component in AMP2
Nominal V_{inRMS}	0.316	0.775	1.23	1.23	
AMP2 Gain	1	1	0.4	0.4	
R_{in}	50 K Ω	50 K Ω	50 K Ω	50 K Ω	R100, R200
$R_{feedback}$	50 K Ω	50 K Ω	20 K Ω	20 K Ω	R101, R201, (& R99 for bridging)
Headroom/margin	15 dB	5 dB	10 dB		

The gain, G is calculated: $G = -R_{feedback}/R_{in}$

The AMP2 kit is shipped with 50 K Ω input resistors and 20 K Ω feedback resistors for a 0.4 gain. These are 0805 size low noise thin film type. You may want to adjust these values for your own application. On request we can supply other resistor values than the ones included.

SUMMARY

- The input sensitivity with the AMP2 kit components are suitable for a pro audio signal level of 1.23VRMS (+4 dBu)
- For home audio equipment you should probably increase the input stage gain by replacing the 20 K feedback resistors with other, higher values.

Input capacitors

Some people have an almost religious in the belief of the influence on the sound quality from the input capacitors. The AMP2 PCBs have space for RM5, RM10 and RM15 lead spacing for the input capacitors so you can practice your belief. You can use many different types of capacitors. Input capacitors are recommended as they block any DC from flowing between equipment components. The inputs of the Tripath chips are biased by about 2.5V, so a capacitor is usually required. However, if you hardwire the power amp to a preamplifier that has a capacitor on the output, then the input capacitor to the power amplifier can be omitted and replaced by a jumper. The input impedance resistor and the input capacitor form a high pass filter. The corner frequency is

$$f_c = 1/(2*\pi*R_{in}*C_{in}).$$

The value should be chosen so that the corner frequency is well below the used lowest frequency. There will be a phase shift close to the corner frequency so a margin of at least one octave is desirable. A f_c of lower than 10 Hz should usually be used for full range and woofer amplifiers. Higher values can be used for mid range / tweeter amplifiers. Sometimes it is claimed that input capacitor values should be about 5 uF or more. This may be true with older carbon film input resistors where impedance had to be low to minimize noise. But with low noise thin film resistors, fairly high impedance can be used and a corresponding low value for the capacitor. The AMP2 kit is shipped with 50 K Ω input resistors and 1 uF capacitors, giving a corner frequency of 3.2 Hz.

It should be noted that two capacitors in series (in the pre amp output and power amp input) will

combine into half the equivalent value, increasing the corner frequency. It can also be noted that the Tripath boards and measurement results given in the Tripath data sheets are usually done with standard electrolyte capacitors as input caps.

Modulator gain

The AMP2 and TK2350 is a feedback topology. The voltage gain of the power section of the amplifier is set by resistors that feed back the loudspeaker output voltage to the chipset. The ground for each side has separate lead backs, for accurate sensing. This means there are four feedback paths and four feedback resistors. For optimal amplifier use, the gain resistor values could be changed, depending on the used rail voltage and the application. Some "over gain" could be used so that the power stage will start clipping, i.e. output voltage is as high as the rail voltage, when the input signal is a bit over the nominal maximum. If the gain is set to low then the power stage will run below full power even when the input stage signal is high enough to saturate the input stage. If the gain is too high, the power stage will start clipping at a low input signal and this implies the preamp level has to be turned down, which may be unfavorable from a THD+N point of view, especially at low volume. A reasonable balance has to be found.

Calculating the modulator gain / feedback resistor values

	Description	Typical value		AMP2 Component
Vpp	Rail voltage (+/-)	40-50V	50-60V	
Rfba	Typical value	1 K Ω	1 K Ω	R105, R108, R205, R208
Rfbb	= Rfba * Vpp/(Vpp-4)	1.1 K Ω	1.07 K Ω	R107, R110, R207, R210
Rfbc	= Rfba * Vpp/4	12 K Ω	15 K	R106, R109, R206, R209

Current limiting

The TK2350 chipset has built in over current setting. Actually, it senses the voltage over a current sense resistor via set point resistors. The set point resistors are of the 0805 size. At an over current situation the amplifier will go into mute and has to be power cycled off/on to reset. The resistors supplied in the AMP2 component kit are 20 K Ω and will trip the amplifier at 10A. You may want to select other values for your application. The on-board power supply capacitors are placed after the current sense resistors. Therefore the MOSFETs may draw current from the caps and the current sense resistors will see a slightly smoothed current signal. The over current sensing is still fast and will mute the amplifier even on short over current peaks. *Please note that the over current shut-off sets the chipset into mute state but if the MOSFETs are damaged, the rail voltage can still be on the speaker outputs.* The over current protection should therefore be regarded as "soft" protection. *Proper fuses are still required.* But fuses may not blow before the loudspeakers are damaged. For "hard" shutdown you can use a relay on the speaker outputs, controlled by the HMUTE signal output. This will shut of the speaker outputs even if the output FETs are damaged.

Calculating the over current sense resistor values

	Description	Current limit trip point					Component
		6A	10A	12A	16A	22A	
I_{sc}	Over-current trip point	6A	10A	12A	16A	22A	
R_{sense}	Current sense resistor	0.01 Ω	0.01 Ω	0.01 Ω	0.01 Ω	0.01 Ω	R51, R52, R53, R54
R_{orc}	$=3473/(R_{sense} * I_{sc}+0.0716)$	27K	20K	18K	15K	12K	R111, R211

Voltage limiting

The TK2350 chipset also has over / under voltage sensing. The voltage values are set with four resistors in the 0805 size. If the voltage is outside the specified values, the amplifier will mute. Unlike the over current sensing the voltage sensing is non latching i.e if the amplifier is unmuted it will mute when outside the specified range and go back to the unmuted state as soon as the voltage is within specified values again. Please again note that the over voltage sensing sets the chipset in mute and can prevent damage by stopping instable conditions like power supply transients. However *it can not protect the chipset from severe over voltage damage; the maximum chipset voltage is +/- 70V also when muted.*

The voltage limiting trip point resistor values

	Description	Value in the AMP2 kit	Component in AMP2
V_{pp}	Rail voltage (positive)	(+40 to +60V)	
V_{nn}	Rail voltage (negative)	(-40 to -60V)	
$I_{vppsense}$	Positive sense current from data sheet	(0.162mA)	
$I_{vnnsense}$	Negative sense current from data sheet	(0.174mA)	
R_{vpp1}	$= V_{pp} / I_{vppsense}$	422 K Ω	R112
R_{vpp2}	$= V_{pp} / I_{vppsense}$	422 K Ω	R113
R_{vnn1}	$= V_{nn} / I_{vnnsense}$	392 K Ω	R115
R_{vnn2}	$= 3xR_{vnn1}$	1.18 M Ω	R114

The tolerances of the voltage sensing and limiting are not very tight so there may be variations in the actual trip points. Please see the TK2350 chipset datasheet for details.

The resistors supplied with the AMP2 component kits will set the lower voltage limit to about +/- 33 volts and the upper limit to about +/- 65 volts.

Note you should have a margin of about 5V from the nominal power supply voltage, or the voltage protection may mute the amp due to voltage transients and component tolerances.

APPENDIX 3: Mounting the TP2350B chip

The TP2350B driver chip can produce some heat, especially if driving MOSFETs with high gate charges and / or at high rail voltages. The chip has a slug on the bottom side that enhances the heat transfer from the chip. The AMP2 PCB has a corresponding area with a number of holes / vias that lead heat from the chip to the back of the PCB. There are also four larger mounting holes around the chip so a heat sink can be mounted with screws on top of the chip. So to enhance the heat transfer, it is possible to

- Use heat conductive paste on the bottom of the chip
- Solder the chip slug to the PCB.
- Use a heat sink on top of the chip and also under the chip.

It is recommended that at least one of these methods is used, especially at high power outputs and high rail voltages.

The use of heat conductive paste on the bottom of the chip is probably the most straight forward method and this method can be used for light and medium loaded amplifier boards. It also has the advantage that it is possible to replace the chip without too much trouble. For higher power applications, such as woofer amplifiers and highly loaded pro amplifiers it is recommended to solder the slug to the PCB or / and use a heat sink.

Soldering the heat slug

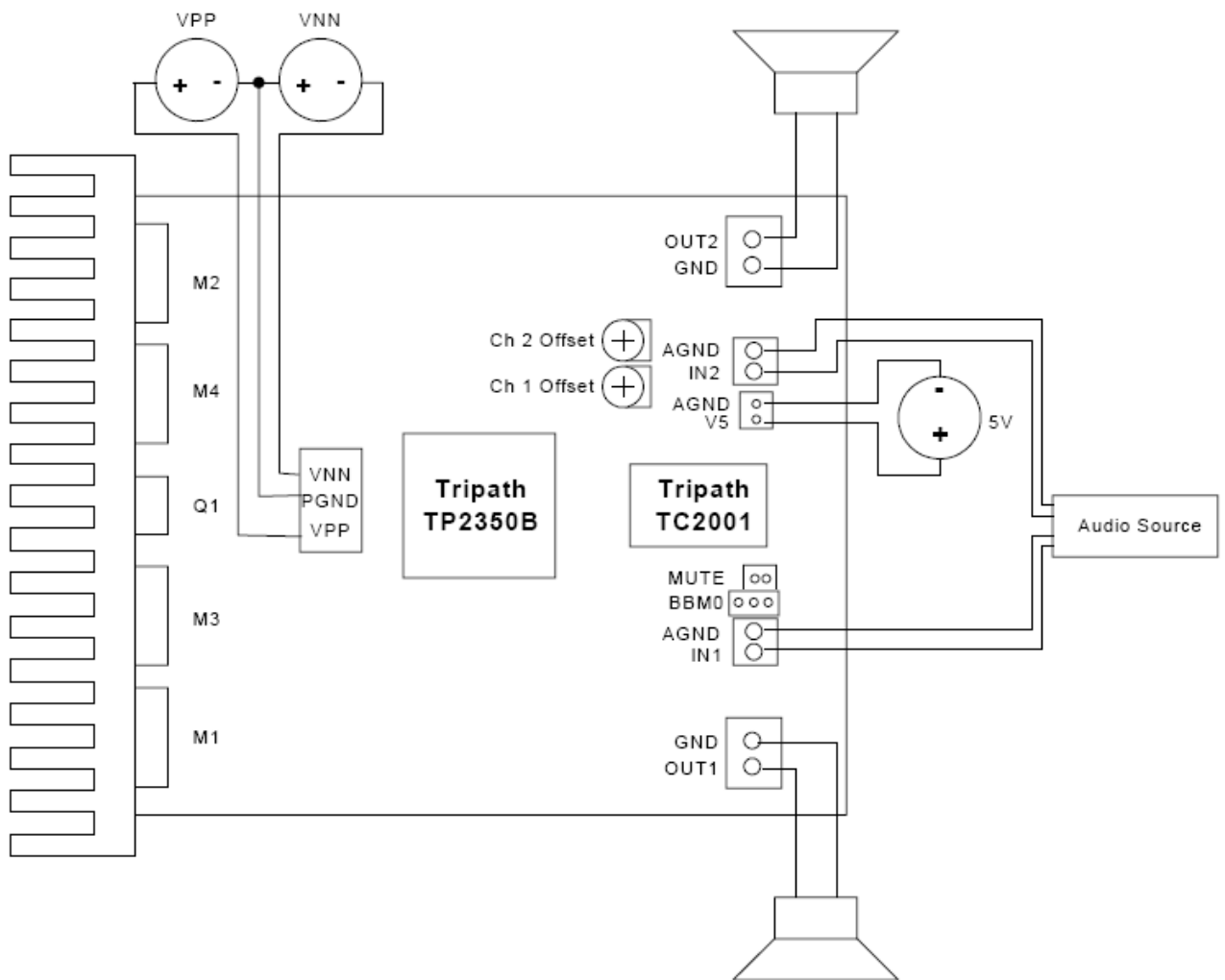
The chip heat slug can not be soldered with a solder iron as it is at the bottom of the chip. A solder oven or similar must be used. Even a domestic electric oven can be used if no solder oven is available. Do as follows:

1. Make sure your solder material has a melting point that is not higher than 190 °C. Most 60%Zn 40%Pb will melt at around 180°C
2. Heat an oven to about 230-240 °C
3. Scrape of the solder mask in the rectangle in the middle of the U1 chip position on the PCB with a sharp knife. Make sure you remove all solder mask varnish in this area.
4. Place about 20 mm (3/4") long bit of solder with a diameter of 0.6 mm or the equivalent amount, bent into a circle, on the cleaned PCB area. Place the PCB in the oven and measure the time it takes before the solder has melted and flowed out on the solder area. It may take about 2-3 minutes.
5. Remove the PCB from the oven and let the PCB cool down.
6. Spread some "no-clean" non aggressive solder flux on the solder area. A solder flux pen is handy for this.
7. Place the TP2350B chip perfectly in place on the PCB. Make sure all chip leads are aligned to the solder pads of the PCB.
8. Place the PCB in the oven and allow as much time as before, but add 20 seconds for the chip to heat up. After this time shut of the oven power, open the oven door and gently fan cool air into the oven.
9. Let the PCB cool down and inspect it.
10. If everything is OK, solder the chip legs to the PCB pads.

Most Zn-PB solder will melt at around 180°C. The TP2350B chip is tested for soldering for 10 seconds in 260°C so you have some margin. Subjecting the chip to high temperatures for a prolonged time may damage the chip.

The TC2001 chip does not have a heat slug and does not require a heat sink.

APPENDIX 4: Connector pinouts



- Mute jumper; **close** to awake the amp
- The V5-agnd must be well stabilized / filtered. Voltages above 5.5V will damage the Tripath chipset
- In1 and In2 should be grounded as indicated, not elsewhere. Input signals above +/- 2.5V can damage the TC2001 chip.

APPENDIX 5: Power supply board for the AMP2 amplifier

Coming...

APPENDIX 6 AMP2 Specifications

Coming...