Assembly instructions AMP6

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. 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 and should be handled with ESD care.

Tools needed

Assembly of the kits requires the usual set of electronics working tools; soldering iron, wire cutter etc. The boards are double sided, double weight copper so a high-power solder iron is recommended, especially for components connected to the ground plane. Solder irons without temperature control should not be used. A magnifying glass/loupe 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.

IMPORTANT

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.

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. There are two main sources of heat on the board; the Tripath chip and the voltage regulator. In most cases, if you mount the board and chip to an aluminum amplifier casing, it is sufficient to cool the chip. However, the voltage of the power supply strongly influences the heat dissipated by the voltage regulator, and you must ensure this has adequate cooling. The voltage regulator must be electrically insulated from the heat sink. The Tripath chip does not need to be insulated, as the back of the chip is internally connected to ground.
- Hookup wire. I recommend soldering connection wires to the board. Optionally you can fit screw/solder terminals with 2.54 mm spacing for the inputs and 5 mm spacing for power and outputs.
- 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. Muting the amp before power on minimizes the turn on thump and is recommended.
- Transformer, power switch and fuse. The default voltage out of the voltage regulator on the board is 14.4V. The voltage regulator needs a couple of volts to work well. With a 15VAC transformer, the voltage regulator works well, but will run quite hot at high power. With a

12VAC transformer, the regulator does not work so well at high power. So if you use a 12VAC transformer, you may need to decrease the voltage by changing the value of R100 /R101 /R102. Replacing R102 with a wire jumper lowers the voltage to about 13.5V, which is suitable for a 12VAC transformer. See the LM1084 data sheet for details. A 80VA (power) rating of the transformer is recommended. If you use the amp for 8 ohm loads only, you can safely go down to 40VA. That will save some space and little on the cost. Use a fuse size as recommended by the transformer manufacturer. Use a good quality mains switch.

The boards for AMP6 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 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 both SMT and hole-mounted 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. Two 3.3 uF capacitors are provided with the kit. The board provides space for RM 2.54 (100 mil) and RM5 (200 mil) lead spacing 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) ≈3 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. Smaller input capacitors can be used at the expense of low frequency damping. If there is a separate woofer in your system, you could use input capacitors of 1 uF or even smaller.

2. The amplifier input stage, in the Tripath 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*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 39 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.

R_{in}	$R_{feedback}$	Input Gain	Suitable signal source
22 KΩ	39 KΩ	-1.8 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
39 KΩ	22 KΩ	-0.56 V/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 $R_{feedback}$

The total 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. In that case increase the power amp input gain.

4. Sleep/mute. The chip has a *sleep* and a *mute* function. Both can shut down the amplifier. You can reach the *sleep* via jumper J2. In sleep mode the chip draws less than 0.3 mA, plus the onboard LEDs. If you use a power breaker or a power supply with an on/off switch, you may permanently close the J2 jumper, but there may be turn on thumps. Therefore, using a switch for the sleep function is recommended rater than using a power switch.

   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 reached 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. The board has a rectifier, bulk capacitor, voltage regulator and smoothing / second bulk capacitor. The voltage can be set by changing the voltage regulator resistor values. For testing a slow fuse of 1A, secondary side 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 startup current.

Schematic of AMP6

The schematics used for AMP6 is almost identical to the schematics in the Tripath data sheet for the TA2020 chip which is in turn more or less identical to the TA2021B chip used in our AMP3. For AMP6, the symbols / component names are the same as for AMP3/TA2021B, rather than the ones used in the TA2020 data sheets.

Differences between AMP6 schematics and the schematics for the Tripath TA2020 evaluation boards.

There are some differences between the schematics for the AMP6 boards and the schematics found in the Tripath data sheets.

- L11 does not exist in the Tripath data sheets. It is a filter choke that should prevent HF between the power ground and analogue ground.

- The C2 capacitor has been changed from 1 uF to a 100 uF electrolytic. These are part of the +5V onboard supply and should improve the +5V supply stability. Also the C50 0.1 uF has been added between the +5V supply and ground.
- C18 and C19 have been upgraded from two 180 uF to one 1800 uF for better transient response.
- L1 and L2 have been removed as they have no advantage. These filter chokes are zero ohms at low frequency, increasing to 60 ohm at 50MHz but they where in series with the 20 kohm inputs and will therefore have little or no effect and have therefore been removed.
- The output filters have been changed after discussions with Tripath and these changes comply with the latest recommendations from Tripath as found in the mars 2005 update of the datasheet for the TA2021B chip. The output capacitors and all components subjected to rail voltage are rated 25V or higher in the kit.
- The output filter inductors have been changed from axial to toroids, to decrease the HF signal leakage.
- The power supply has been added to the board.
- Two status LEDs with driver MOSFETs have been added.

Mounting the components

NOTE 1. FET Q1 and Q2, which drive the LEDs, are drawn the wrong way around on the PCB. Turn the FETs half a turn compared to how they are drawn on the board, i.e. the flat surface of the components should face the center of the PCB. Otherwise, the LEDs will always be on.

NOTE 2. There are three 0.1 uF capacitors, C50, C6, and C7, which should be mounted and soldered into the holes of the TA2020 chip. These are critical to the function of the amplifier. The locations of these are printed on the bottom side of the PCB. *Do not solder these capacitors until when you fit the TA2020 chip*. So put aside one 0.1 uF, 2.5 mm lead spacing capacitor and the two 0.1 uF, 5 mm lead spacing capacitors for this.

1. First, solder the small capacitors on the bottom side of the board, but NOT the ones that are mounted with the TA2020 chip. C23, C24, C29 and C30 must be mounted on the bottom side, while C16, C17, C26 and C102 can be mounted either on the bottom or top side of the board. A picture of how the components are placed is included in APPENDIX 2. Solder C5 (charge pump) in place.

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

3. Mount and solder the rectifier and the voltage regulator. Try to get these in a right angle to the PCB and at the same plane so they can be mounted flat on a heat sink. Mount and solder the voltage adjusting resistors R100, R101 and R102. The regulator voltage output can be calculated as U = 1.25*(1+(R101+R102)/R100)). With R101 = 1 K and R100=100 ohm. The R101 and R102 are mounted in series. With the kit are supplied 68 ohm, 47 ohm resistors for R102.

R102	Nominal voltage
68 ohm	14.6
47 ohm	14.3

0 (wire jumper)	13.75

The Tripath TA2020 chip is rated 16V absolute maximum. About 14.6V is a reasonable maximum in real life, to allow for fluctuations.

4. Solder the rest of the resistors in place. See table 1 for selecting the input resistor values.

5. Solder C14, C15, C100 and C20 in place. As with all electrolytic capacitor the polarity must be respected. The positive pad is usually rectangular and marked with a "+" on the PCB, while the negative is usually round. On the capacitor, the negative side is usually printed and the positive lead is longer than the negative. Solder C2 (for the +5V supply).

6. Solder the diodes in place. The diodes are for overshoot protection of the speaker outputs. These diodes are surface mount of a fairly large "SMB" size. Note the diodes have polarity that MUST be correct or the amp will be damaged. The diode cathodes are marked by a thin band across the component and this is also on the symbol in the PCB. The band on the component may be hard to see but it is there. D20 is a discharge diode for C1819.

7. Solder output capacitors in place. These are C12, C13, C22, C25, C27, C28

8. Solder Q1 and Q2. These should be rotated half a turned compared to how they are drawn on the pcb, i.e. the flat surface of the FETs should face L7 / C13 / R10. These are the current driver transistors for LEDs. R200 and R201 limit the current to the LEDs. Solder the LEDs D15 and D16 if these are to be on the board. Otherwise, they can be placed on the amp front panel. If placing the LEDs away from the board, you can take the positive supplies to the LEDs from pin 1 and 2 of J2. The returns should in that case be to AGND, normally J2, pin 8. See also in the APPENDIX 3 for a jumper pin-out. D16, the green diode, indicates normal operation. It actually uses the OVRLDB output of the chip. This output is high when the amp is un-muted but starts going low if the input signal is to high. This would be synonymous to the amp output clipping, and indicates you should probably decrease the volume… The red led is connected to the Tripath chip FAULT output. It turns on the LED ion case the amp is overheated or has experienced over current.

9. Solder the bulk capacitor C99 in place.

10. Solder L11 in place. It is a choke that connects the ground planes of the analogue and power sections. Mount C1819 in place but do not cut the leads very short. Place a test load in form of a 1 kohm or similar resistor over the C1819 leads.

11. Now, you can connect a transformer and check the voltage from the regulator. If it is OK, you can remove test load resistor from C1819 and continue**. Do not proceeding any further until this test has passed.**.

12. Wind the toroid inductors. You should use 39 turns of 0.5 mm wire. Wind as tight as you can, drawing the wire snug every quarter turn. Tight winding minimizes stray signal leakage from the inductors. You should be able to do the 39 turns on wire before coming back to the starting point, without

overlapping. When you have completed the winding, you should have a total of about 6 cm of wire ends left, if you started with a 75 cm long piece of wire. Leave the wire ends so you can pull the toroids snugly when soldering them to the board. In one of the Tripath datasheets, the winding is described as follows: *It should be noted that when multiple layers are used there may be an increase in winding capacitance, which can cause ringing and increased radiated emissions. Winding techniques, such as bank winding, can minimize this effect. It is important that the initial windings not be crossed over by the last few windings. If a few windings more than the single layer are required it is best to wind the core with a full single layer, back off a number of turns, and rewind over the last few windings.* If you have an inductance meter, measure the inductance. It should be 10 uH. If you do not have an inductance meter, count the turns. The picture above shows a toroid with half of the turns done. The first turns are difficult do well, and will have to be redone when the rest of the windings have been finished.

The finished toroid. Tight winding minimizes signal leakage

13. Scrape the enamel off the wires where they will be soldered and solder the toroids to the board. Pull the toroids snugly to the board when soldering. You can also glue the toroids to the board to prevent them from working loose. In case the board is subject to vibrations, the copper wire could break due to fatigue if not fixed properly. This is especially important for in-speaker mounted amplifiers and mobile (car) amplifiers. Temperature resistant glue should be used, as the toroids may get hot at high power.

14. Mount the TA2020 chip in place. Take care getting the chip straight and in the same plane as the voltage regulator and the rectifier. Solder a few of the leads lightly and mount C50, C6, and C7 in the holes indicated on the bottom side of the board. *It is essential to keep the leads of these three capacitors as short as possible.* You need to cut the leads of these capacitors before mounting them. You will need to push them gently into place. When they are in place, solder all of the TA2020 leads.

15. 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 and the voltage regulator may dissipate something similar. Then, a 3ºC/W heat sink is reasonable. Medium or low power applications will not dissipate a lot of heat. The heat slug of the TA2020 chip is connected to ground and does not

require electrically insulated mounting. Silica heat transfer compound or similar should be used to improve cooling.

IMPORTANT: the voltage regulator heat slug is at output voltage and MUST be electrically insulated from the heat sink. A heat insulating washer and an insulating bushing for the mounting screw is included in the kit.

The mounting holes on the PCB are NOT connected to the ground plane. Pin 1 of J3 is grounded.

Trimming and testing

- (Remove the jumper from J2 pin 7 of for *sleep* mode, if it is there).
- For testing, use a 200 mA fast fuse on the power supply.
- Turn on the power supply. When in sleep mode, the amp should draw less than a mA.
- If all is OK, connect the J2 jumper from pin 7 to pin 8 to make the amp Awake. The green LED should go on (even better is to mount a switch on your amp front panel instead of this jumper).
- 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 J2
- 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. If all is OK, switch off power, replace the power supply fuse for a larger one, rated as your transformer and try again with higher volume. For testing at higher power, the chip should be mounted on a heat sink.
- Enjoy the music!

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

Some notes on the schematic:

A schematic is under development and is available on request. The AMP6 schematic is very similar to the Tripath evaluation board schematics. However, the symbols used, are taken from the TA2021B schematics. The TA2020 (used for AMP6) and the TA2021B (used for AMP3) are virtually identical internally, but the packages are very different.

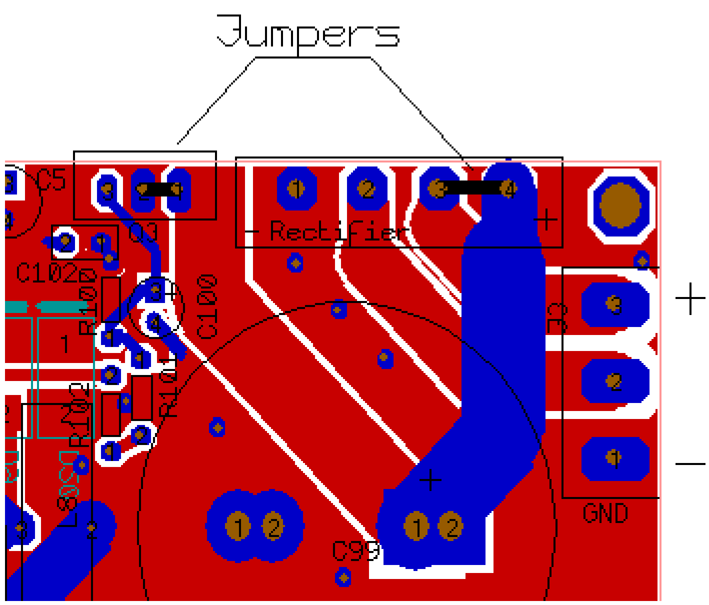
On the AMP6 kit PCB:s the required +5V is generated in the chip. In the Tripath demo board and schematics, the +5V is decoupled to ground by one 0.1 uF, C17 and one 1uF, C2. On AMP6 the there are three caps, C17, C2 and C50. The C50 is parallel to C17. On AMP6, the value of C2 has been increased from 1uF to 100 uF to give good transient stability for +5V. The +5 “internal output” is from pin 30 and connected to pin 2 and pin 8. The small 0.1 uF caps should decouple HF noise while the big C2 stabilizes the voltage. One pad of each of these caps is grounded. C50 is one of the capacitors mounted with the TA2020 chip. C23 and C24 are for EMI damping on the input and C29 and C30 for the outputs.

Q1 and Q2 are small MOSFETs that take low level signal outputs and drive the LEDs, as the output from the Tripath chip can not drive LEDs directly.

AMP6 with 12V DC

If you use AMP6 with a 12V DC source like a 12V battery, or an external 12V DC supply, you can remove the rectifier and voltage regulator, as they cause a voltage drop of about 2V and thereby limit the maximum input power. Note that keeping the rectifier, gives input polarity protection. Without the rectifier, you must ensure the amp is connected with proper polarity, or the Tripath chip will be damaged.

- Optionally remove the rectifier
- Remove the voltage regulator Q3
- The following components will not be active and can be removed
    - R100, R101, R102, C100, C102, D20
- C99 can still be used, if the power supply works with a capacitive load
- Jumper the rectifier 8if it has been removed and Q3 as shown below.
- Connect the external power as shown below. Polarity MUST be correct, or the Tripath chip will be damaged!



APPENDIX 1 BOM (Bill Of Materials)

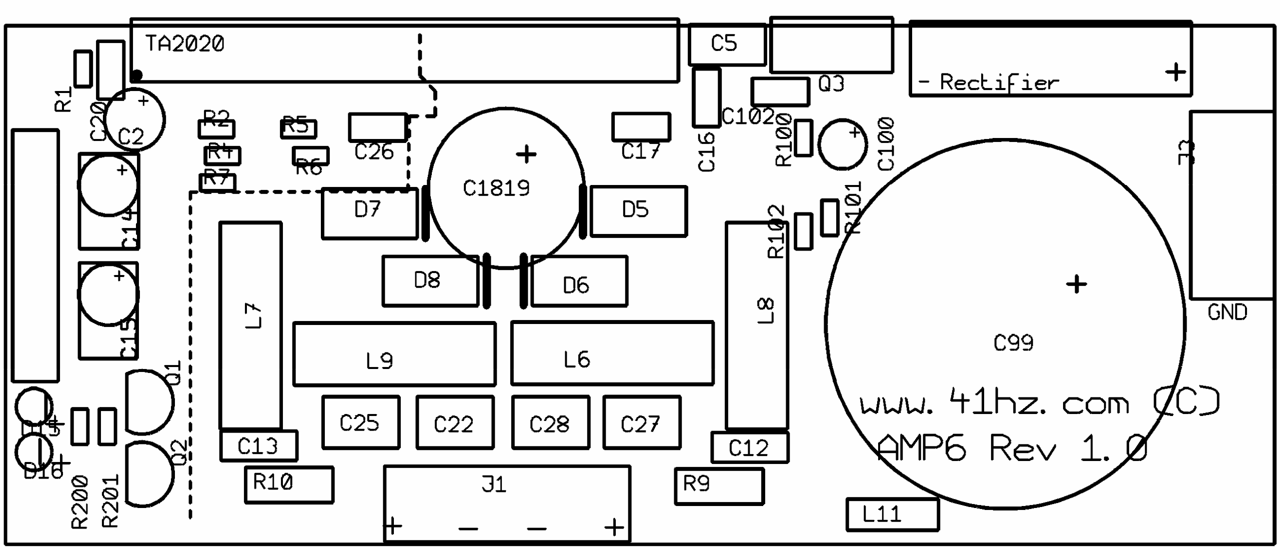
Name	Value	Function
C2	100 uF 10V	+5V bulk cap
C5	1 uF	Charge pump output decoupling
C6	0.1 uF RM 5	Power supp decoupling
C7	0.1 uF RM 5	Power supp decoupling
C12	0.22 uF	Speaker output filter
C13	0.22 uF	Speaker output filter
C14	3.3 uF	Signal input cap / DC blocking
C15	3.3 uF	Signal input cap / DC blocking
C16	0.1 uF RM 2.5	Charge pump switching pin cap
C17	0.1 uF RM 2.5	+5V decoupling
C20	0.1 uF RM 2.5	Sleep jumper de-bounce
C22	0.47 uF	Speaker output filter
C23	100 pF	Line input EMI filter
C24	100 pF	Line input EMI filter
C25	0.47 uF	Speaker output filter
C26	0.1 uF RM 2.5	BIAS decoupling
C27	0.47 uF	Speaker output filter
C28	0.47 uF	Speaker output filter
C29	0.01 uF	Output EMI suppression cap
C30	0.01 uF	Output EMI suppression cap
C50	0.1 uF RM2.5	+5V decoupling
C99	15.000 uF 25V	Bulk capacitor on un-stabilized side
C100	10 uF 25V	Voltage regulator ripple rejection damping
C102	0.1 uF RM2.5	Voltage regulator decoupling
C1819	1800uF 16V	Bulk capacitor on stabilized side
CMP1	TA2020	Tripath TA2022
CMP15	Rectifier	Power supply rectifier
D5	SMB Ultra-fast diode	Output overshoot protection diode
D6	SMB Ultra-fast diode	Output overshoot protection diode
D7	SMB Ultra-fast diode	Output overshoot protection diode
D8	SMB Ultra-fast diode	Output overshoot protection diode
D9	SMB Ultra-fast diode	Output overshoot protection diode
D10	SMB Ultra-fast diode	Output overshoot protection diode
D11	SMB Ultra-fast diode	Output overshoot protection diode
D12	SMB Ultra-fast diode	Output overshoot protection diode
D15	LED_3MM Red	Status LED
D16	LED_3MM Green	Status LED
D20	SMB Ultra-fast diode	Discharge of C1819 capacitor
L6	TOROID_T60	Output filter inductor
L7	TOROID_T60	Output filter inductor
L8	TOROID_T60	Output filter inductor
L9	TOROID_T60	Output filter inductor
L11	Choke, axial 60 ohm 3A	Ground separator
Q1	2N70000 TO-92	Signal MOSFET LED driver
Q2	2N70000 TO-92	Signal MOSFET LED driver
Q3	LM1084	Voltage regulator, 5A adjustable voltage
R1	1 M	Sleep signal pin pull-up
R2	22 K (alt 39 K)	R_input
R4	22 K (alt 39 K)	R_input
R5	39 K (alt 22 K)	R_feedback

R6	39 K (alt 22 K)	R_feedback
R7	8.2 Kohm	R ref (nominally 1.2V)
R9	10 ohm 0.5W	Speaker output filter Rzobel
R10	10 ohm 0.5W	Speaker output filter Rzobel
R100	100 ohm	Regulator set resistor
R101	1 K	Regulator set resistor
R102	68 ohm (alt 47 ohm or 0 ohm)	Regulator set resistor (in series with R101)
R200	5.6 K	LED current limiting
R201	5.6 K	LED current limiting

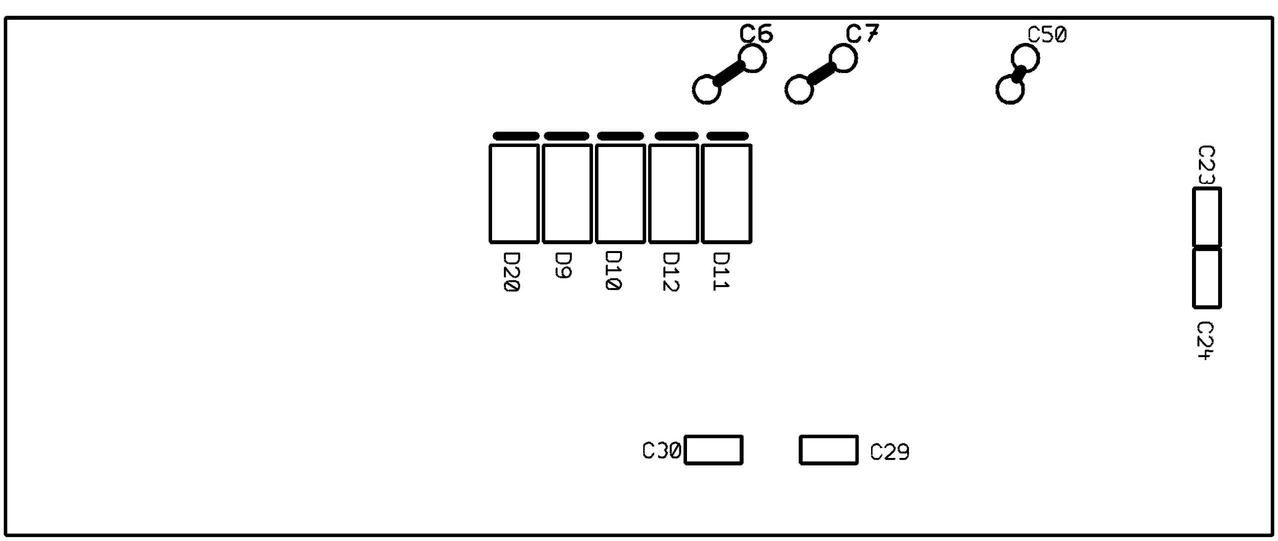
APPENDIX 2

Component placement AMP6

Top View



Bottom view



Pin connections

All numbering assuming board is placed with Tripath chip upwards, and placed as shown on below.

		Note
J1	**Speaker outputs (nr 1 at left to nr 4 at right)**	
1	Speaker output 2 Positive	
2	Speaker output 2 Negative	*NOT connected to ground*
3	Speaker output 1 Negative	*NOT connected to ground*
4	Speaker output 1 Positive	
J2	**Low level signals (nr 1 at bottom 8 at top)**	
1	Fault LED output (to red LED).	*Return to J2 pin 4, 5 or 8. Indicates faults (Red led turns ON)*
2	OVRLDB LED output (to green LED)	*Return to J2 pin 4, 5 or 8. Indicates too high input signal. (Green LED turns OFF)*
3	Signal input 2	
4	Analogue ground	*Input ground*
5	Analogue ground	*Input ground*
6	Signal input 1	
7	Sleep (to sleep switch)	*Return to J2 pin 4, 5 or 8*
8	Analogue ground	
J3	**Power (nr 1 at bottom to nr 3 top)**	
1	Ground	*Connect to amp casing and earth.*
2	AC 1 (from transformer)	*NOT to amp casing / earth.*
3	AC 2 (from transformer)	*NOT to amp casing / earth.*