41Hz Audio

www.41hz.com

Assembly instructions AMP4

Assembly Instruction **Revision 1.0 December 2021**

Document ID: 26

Written by Jan Fredriksson

AMP4 FEATURES

- Signle rail supply. Power supply for main power and +5V on the board. Just add a transformer.
- Very low THD+N
- Stereo bridged outputs
- Suitable for 8 ohm loads with up tp 32V supply or 4 ohms at maximum 24V supply
- Line level analoge audio inputs. Sensitivity adjustable with external resistors
- High efficiency 85-90% mean small heat sink and small transformer requirements
- Over / under voltage turn off
- Over current protection, temperature overload protection
- Speaker outputs with relay
- The module is suitable for amplifiers and active speakers

TK2050 chipset features

- 0.007% THD+N @ 30W 8Ω
- 35W @ 6Ω, < 1% THD+N
- 50W @ 8Ω, < 3% THD+N
- Audiophile Quality Sound
- High Efficiency
- 92% @ 60W 8Ω
- Dynamic Range >100 dB

Table of Contents

Assembly instructions AMP4

The assembly instructions are always under development. You where sent the latest version when you placed the order. You may check the 41Hz Support web site download page for updates and related documents like chip datasheet, images etc.

Feedback is appreciated and helps improve assembly instructions.

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) and schematics are available as separate documents.

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 Components packaged in a shielded, aluminized bag should be considered ESD sensitive and should be handled with ESD (Electro Static Discharge) care.
The most sensitive parts are the small FETs. Also the Tripath chips use MOSFEt outputs which by nature are sensitive to ESD, even if the chip has internal ESD protection.

Preferably work on a conductive, grounded “ESD mat”, and avoid touching the chip leads with your fingers. Discharge yourself to something grounded 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. 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.
- 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 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 AMP4 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 in the forum on http://www.41hz.com

Considerations

1. On the board there are two signal input capacitors, C100 and C200. These are required, as the amplifier is internally biased to about +2.5V. Two 3.3 uF Panasonic FC electrolytic capacitors and two 1 uFpolyester capacitors are provided with the kit. 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 12 mm long, 7 mm wide. The input capacitors form a high pass filter together with the input resistor Rin. The cutoff frequency is $F=1/(2*\pi* Rin*Cin)$ For example, with Rin = 20 Kohm and Cin = 3.3 uF, the cutoff frequency is $F=1/(2*3.14*22000*0.0000033) \approx 3$ 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. 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 1 uF.

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, R21 and R29 are the Rin and R23 and R31 are the Rfeedback. With the kit, there are four 20 KΩ resistors and four 49.9 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.

Table 1. Gain setting recommendations

R_{in}	$R_{feedback}$	Input Gain	Suitable signal source
20 KΩ	49.9 KΩ	-2.5 V/V	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 V/V	General use
49.9 KΩ	20 KΩ	-0.4 V/V	Preamplifier with fairly high output signal

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. The chip has a *mute* function, which disables outputs. If you use a power on/off switch, you may permanently close the mute jumper.

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

6. Power supply. For testing at low power any 20-25V supply capable of delivering about 200 mA 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 startup current.

About the schematic of AMP4

The schematics used for AMP4 has similarities to the schematics in the Tripath data sheet for the TA2050 chip but is far from identical. The schematics are available in separate files / documents.

Mounting the components

1. Start by mounting all the surface mount capacitors and resistors and the two Tripath chip. Do not mount the small SOT23 transistors yet.

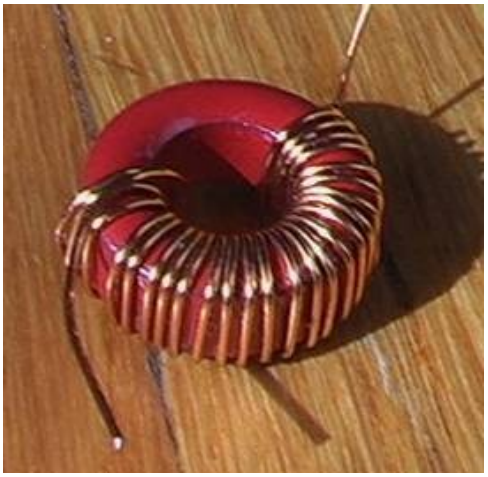
2. Complete the +9V section which is a pre-regulator for the +5V. The components needed are: R6, C50, C51, Q10, R56, R57 and L5. On early boards, the position of R57 and R56 are reversed on the PCB but correct in the components placement pictures later in this document. Currently shipping boards, the printout on the PCB is also correct.

3. Solder D99 (rectifier) in place.

4. Now you can do the first test, by connecting power to J2. Make sure the J5 and main power jumper J10 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.

5. When 9V is OK, continue with the +5V section. It is C91, Q2, C110, C112, R110, R111 and R59.

6. You can now test the +5V. J5 and J10 should still be open. 5V can be measured on J4, pin 7 to *agnd*, 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.



7. Continue with all the hole-mount components, except the big bulk capacitors C1000 and C1001.

8. Solder D1 in place.

9. 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.

10. Wind the toroid inductors. You should use 53 turns of 0.6 mm wire. Wind tightly, holding the core in one hand, the wire in the other hand. Do not use vices or other tools as you then risk cracking the toroid core. Pull the wire snug every quarter turn. Tight winding minimizes stray signal leakage from the inductors. You should be able to do the 53 turns on wire before coming back to the starting point, without overlapping. Leave the wire ends un-cut so you can pull the toroids snugly to the PCB when soldering them. If you have an inductance meter, measure the inductance. It should be 15 uH. If you do not have an inductance meter, count the turns. The picture 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.

Picture: *The finished toroid. Tight winding minimizes signal leakage*

11. Torch the enamel off the wires or carefully scrape them clean where they will be soldered. Tin the leads before soldering them to the PCB, by heating the wire ends with solder iron temperature set to maximum. Adding flux helps. Now mount the toroids to the PCB. Pull them snugly to the board by the wire ends with a tool when soldering so they are vertically aligned and can not move. Double check that the inductor soldering is good. Normally you should never cut the leads of a component after soldering as the soldering should be the last step. However, with the inductors,

soldering after cutting the leads short may make them move and you have no wire ends to hold on to so I recommend cutting the leads after soldering, cutting them a few millimeters form the solder so the solder joint is not damaged. 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.

12. Solder all the output filter components in place; C11, C15, C17, C23, C25, C28, C32, C36, R42 and R50. Solder the relay.

13. Mount and solder C102 and C103, making sure they are really resting towards the PCB before soldering, so they can not move and work loose.

14. Now close the J5 jumper. 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 a milliamp when muted.

15. Close the J10 jumper by soldering a piece of flattened wire, as long as the pads are wide. The cut off ends from Q10 will work fine.

16. Connect the power and check that it is not drawing more than about 100 mA.
17. Solder the trimmers and input capacitors C100 and C200.

18. 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. 100 ohm and 270 ohm 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.

19. Turn on power and verify that the relay is closing by temporarily connecting R100 to ground.

20. Solder all the small SOT23 transistors in place. U10, U11 and U12 maneuver the relay while Q4, Q5 and Q6 drive status LEDs connected to J3.

21. Turn on the power, un-mute the amp by closing J4 pin 5 and pin 6. These pins are normally connected to a switch on the amp casing. Verify that the HMUTE signal on J4 pin 8 goes from high to low when the amp is unmated. The HMUTE signal can drive a red LED (positive side to J4/pin 8, negative to AGND) directly, but not a blue LED, which requires much more power.

22. If all is OK, adjust the POT1 and POT2 trimmers until the voltage on the speaker outputs is within 10 mV of zero.

23. Connect speakers and signal source and enjoy the music!

Pin-outs

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

1	Out1+
2	Out1-
3	Out2+
4	Out2-

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

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

1	Ground / Chassis
2	AC1
3	AC2

Note: Transformer should not be connected to Ground, but to AC1 and AC2 only. If a DC source is used, connect negative to pin 1 and positive to pin 2 or 3

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

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

Note: The LED outputs are driver 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. LED voltage can be set by changing R81, R82, R83.

J4 Inputs connector. (Pin 1 = rectangular pad and dot mark)

1	Input2	
2	AGND	
3	AGND	
4	Input1	
5	MUTE REF GND	
6	MUTE	Amp mutes if this pin is +5V, unmutes if pin is grounded
7	+5V	(for MUTE pin or external devices)
8	HMUTE	Output high indicates amp is muted bu mute pin or TC2000 chip fault detected. Output low = amp unmuted. This pin can drive a LED directly. The HMUTE signal also controls the relay via a drive FET.

Troubleshooting

This section will gradually be increased, as feedback comes in. Please let us know!!! There is now also a generic troubleshooting document on the support site / downloads section.

+5V 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 500 mA or more. The AMP4 +5V section components need a total of about 80 mA.

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.

If R52 and R54 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 others. Details of how to calculate the limits can be found in the TA2050 chipset data sheets from Tripath.

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

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

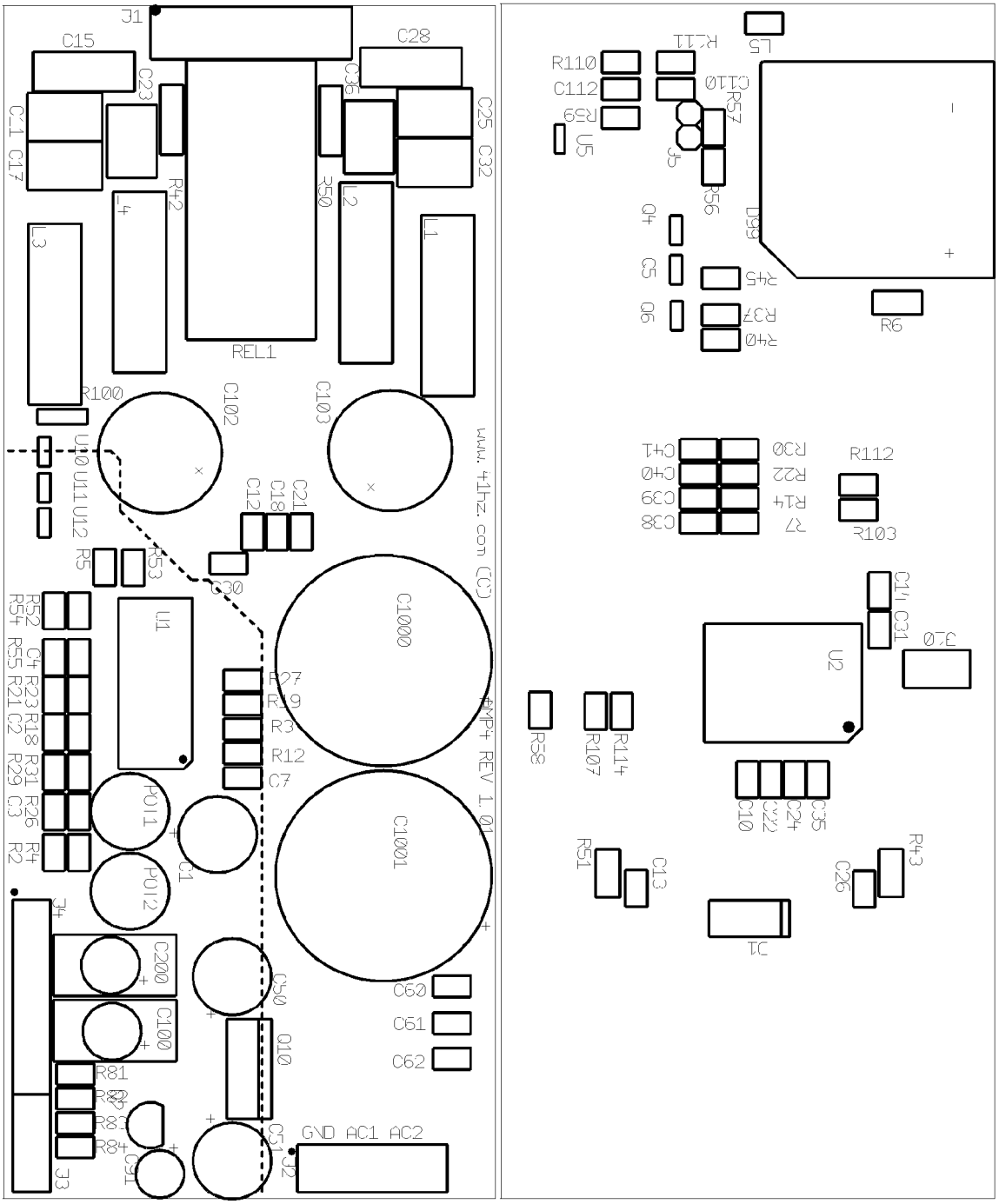
If you have any questions, comments or feedback, please write

- On the support web site http://www.support.41hz.com
- In the forum on the web site http://www.41hz.com.
- By email to jan@41hz.com

APPENDIXES

- BOM (Bill Of Materials): Separate file
- Schematics: Separate file

Components Placements



Top side *Bottom side*