Assembly instructions AMP-1

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

On delivery, check that all components have been included. If something is missing, let us know immediately. Components packaged in an aluminized bag should be considered ESD sensitive and should be handled using normal ESD precautions.

A bill of material (BOM) is found as APPENDIX1 here.

The layout and schematic of the AMP1 board is identical to the Tripath Reference Board for the TA2022 chip. However, most component types and values have been upgraded and are different compared to the Tripath reference board. Datasheets for the TA2022 chip and the demo board are available on the 41hz.com web site.

Optional 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.
- It is recommended that you solder hookup wires to the board. Optionally you can fit screw / solder terminals.
- A mute/un-mute switch is recommended for thump-less power-on. You can wire this to a switch on your panel. Optionally use a 2.5 mm jumper (50 mil) on the board. This is the same size of jumper that is commonly used in computers.

Soldering

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

The boards for AMP5 are double weight, double sided copper. Even if the PCB and components are small, quite a powerful soldering iron is very helpful. Especially components and pads connected to the ground plane require significant heating. A temperature controlled 50W soldering iron is the minimum recommended. At the same time, applying excessive heat may damage the FRP plastic of the board, causing the copper pads 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. Increasing the solder iron temperature is NOT recommended as it increases the risk of damaging the board. Some information on how to solder components is available in the forum on http://www.41hz.com

Selecting the gain

The amplification, or gain, of the amplifier is set in two stages: Input stage gain and power stage gain. Optimally you should match the input gain to your signal source signal level and the power stage gain to your supply rail voltage. Just remember that music signals are very dynamic by nature, signal levels are approximate and impossible to predict accurately.

Input stage gain

In *table 1* you can see some typical input signal sources and gains you can set with the supplied components. The maximum recommended voltage *out* of the input stage is +/-2V peak to peak (1.41 VRMS), including some margin. At higher signals, the input stage may clip and the amp may even be damaged.

The amplifier input stage in the Tripath chip is of the inverted operational amplifier type. The gain is calculated as:

Gain = -1 * Rfeedback/Rin

On the board, R51 and R32 are the R_{in} and R43 and R44 are the $R_{feedback}$. With the kit, there are four 22 KΩ resistors and two 47 KΩ. With these resistor values, you can choose one of three different input gains/sensitivities as shown in table 1. If you use other input resistor values, they should be of a low noise metal film type.

Table 1. Input stage gain setting recommendations

R_{in}	**$R_{feedback}$**	**Gain [V/V]**	**Suitable signal source**
22 KΩ	47 KΩ	-2.1	Direct connection of portable MP3/CD player with built in volume control or via a passive volume pot.
22 KΩ	22 KΩ	-1	General use
47 KΩ	22 KΩ	-0.47	Pro / preamplifier with fairly high output signal

Modulator gain

The "modulator gain" is the power stage voltage gain. You can select this to match your power supply voltage, by selecting the value of the modulator feedback resistors R31 and R52. *Table 2* gives the values for modulator feedback resistors and the rail voltages they are suitable for.

Table 2. Modulator / power stage gain

R_{mfb}	**Gain [V/V]**	**Suitable rail voltage**
8.2 KΩ	16	+/-24V to +/- 30V
10 KΩ	19.3	+/-28V to +/- 35V

The total gain is the input stage gain multiplied by the modulator gain. If we assume you have selected the input stage gain, as you normally should, for a maximum +/-2V output, then the maximum output voltage to the speakers will be 2 x the modulator gain. For example with 10 kOhm feedback resistors, and +/-2V from the input stage, you would have +/- 38.6V out maximum. You can see that the calculated maximum output can be higher than the rail voltage. In reality, the output voltage can only be as high as the rail voltage. Therefore there will be some clipping of the highest peaks at the maximum input signal. This is quite OK and assures you can reach maximum output power at near the maximum input volume setting. If you decrease the modulator gain, you will not reach maximum power/clipping even at maximum input signal. If contrary, the modulator gain is too high, you will reach maximum output before the input stage is at full signal. The disadvantage here is that at low volume, the signal voltage will be very low, and the background noise may be higher than it could be.

Table 3. Gain setting; summary

Input stage gain	**Result**
Input Gain to low	Maximum output is to low. Amp not fully used.
Input Gain correct	Output from input stage is +/-2V peak to peak at full input volume
Input gain to high	Input stage clips before maximum output power is reached
Power stage gain	
Modulator gain to low	Output never reaches maximum power, even with maximum input signal
Modulator gain correct	Output clips slightly with maximum input signal
Modulator gain to high	Output clips excessively at full volume. Input stage is operating at low voltage; possibly increased THD+N at low power levels

Summary:

- Select input gain to match the music source signal level
- Select the modulator gain to match the supply rail voltage

In the data sheet for the Tripath TA2022 chip there are more details on how to calculate the gain.

Power supply

AMP1 requires a dual rail power supply plus a stabilized +5V 100 mA supply. In the “toolbox” section of the web site, there are some recommendations for power supplies but below is a summary.

The rail voltages could be +/-20V to +/-35V. Higher voltage gives a higher possible maximum power output. With 4 ohm speakers or 8 ohm bridged mono, +/- 31V is the maximum recommended, to avoid over-current shutdown at high power. The VA rating depends on your application. It could range from 80VA for a +/-24V supply connected to an 8 ohm speaker to 300VA or more for a bridged 8 ohms subwoofer.

The rail supply does not need to be stabilized. Bulk capacitors of at least 10.0000 uF per side are recommended.

The +5V is used for the analogue input amplifier and the in-chip digital parts of the TA2022. It needs to be well stabilized and well bypassed. You can use a 7805 voltage regulator or similar by tapping current from the positive main supply, or you can use a separate transformer supply for the +5V. The +5V should be connected using shielded cable, to prevent it from picking up noise. The +5V should not draw more than about 60 mA.

Hookup and shielding.

Switched mode amplifiers are a bit noisy by nature, in the sense that they emit EMI that is generated by the high power, high frequency output transistors. This can be transmitted via cabling or as radiated in the atmosphere and picked up by other equipment like radios, preamplifiers etc or by the amplifier inputs. It is therefore recommended that some precausions are taken. The most important is that the amp is housed in a metal/shielded casing.

Proper grounding is also important. Note that input ground should be taken to the board J2 connector ground, not to the housing or power supply ground. The speaker returns should lead to J3 ground, not to the casing or power supply ground.

Assembly instructions AMP1 V1.0© 41Hz Audio / Jan Fredriksson

It is recommended that hookup cable for the signal input and +5V is shielded and as short as possible so that it does not pick up noise from the outputs. Input cables and +5V cables should lead away from the outputs as far as possible.

Speaker cable and power cables can be twisted. All cables should lead the shortest way out of the casing.

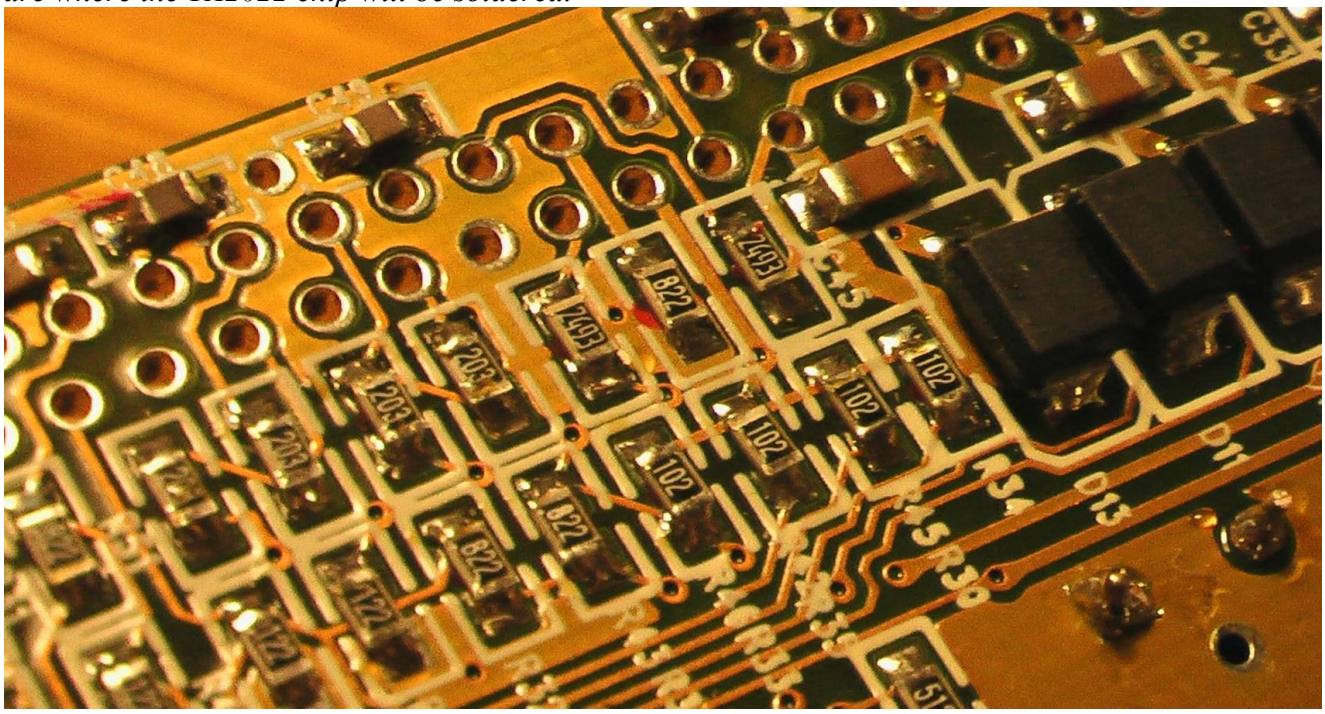
For most users, using shielded cables and a metal housing provides sufficient EMI damping. If this is not the case for you, for example if you get disturbances on radio receivers, damping can be used on the cables. Typically, the amplifier inputs can be decoupled with 100pF capacitors on the RCA connectors, between "ground and live", or using two 100 pF capacitors, from each lead to the housing. On the speaker outputs, capacitors of 4700pF can be placed between ground and live on the speaker connectors or from each speaker lead connector to the metal housing. For more information, there are some Tripath application notes available on the download page of the 41Hz Audio web site dealing with EMI and how to prevent it.

Mounting the components

1. First, solder all surface mount components. They are all placed on the "bottom" side of the board. A sketch of how the components are placed is included below. It may be best to work in the order resistors – capacitors – diodes. Note the polarity of the diodes. The side of the diode with a line across should be at the end that has mitered / cut of corners on the PCB print.

   Some information on how to solder surface mount components is available on the web site.

Bottom view showing most SMT components. The two rows of holes in the upper part of the picture are where the TA2022 chip will be soldered.



2. Optionally solder board connectors. If you do not use these, save cabling until last.

3. Solder the conventional resistors and trim of excess wire. When through hole components are soldered properly on the back of the board, there should be solder right through the board holes and solder should be coming out on the front side, covering all round the component legs on both sides of the PCB.

4. Solder the film capacitors and trimming potentiometers.

5. Solder electrolytic capacitors. Respect the polarity as marked on the PCBs.

6. Solder the smaller vertical inductor, L4

7. Wind the toroid inductors. You should use about 42 turns of wire to get 11 uH (10-12 uH is acceptable). Wind as tight as you can. Tight winding minimizes HF signal leakage from the inductors. If the wire is packed tight on the inner perimeter, you can do a single layer of about 38 or 39 turns of wire before coming back to the starting point, so about 2-3 turns will have to overlap if you use 0.6 mm (AWG 22) wire. The first turns where probably not so neat so unwind these and redo them. From one of the Tripath datasheets*; 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 your application is not going to use high currents, for example if you connect the amp only to 8 ohm speakers, you can also use slightly thinner wire. Using 0.5 or 0.55 mm wire will let you wind all turns without overlapping. If you have an inductance meter, measure the value and adjust if required.

8. Scrape the enamel off the wires where they will be soldered and solder the wires, pulling the toroid snugly to the board when soldering them. You can glue the toroids to the PCB to prevent the wires from breaking of due to vibration fatigue or shocks. If so, use temperature resistant glue as the inductors get warm at high power.

9. Solder the TA2022 chip in place. The board holes are fairly large compared to the chip leg diameters so take care getting the chip straight. Some leads connect to the board ground plane and internally in the chip to the heat slug and again, these are a bit difficult to solder, due to the amounts of heat required. Preheating the board or chip will help.

Last connect signal and power connectors. A toggling switch can be connected J1 for muting and turning on the amplifier. If you do not use an external mute switch, you can permanently put a jumper from the middle hole of J1 to the Awake side of J1

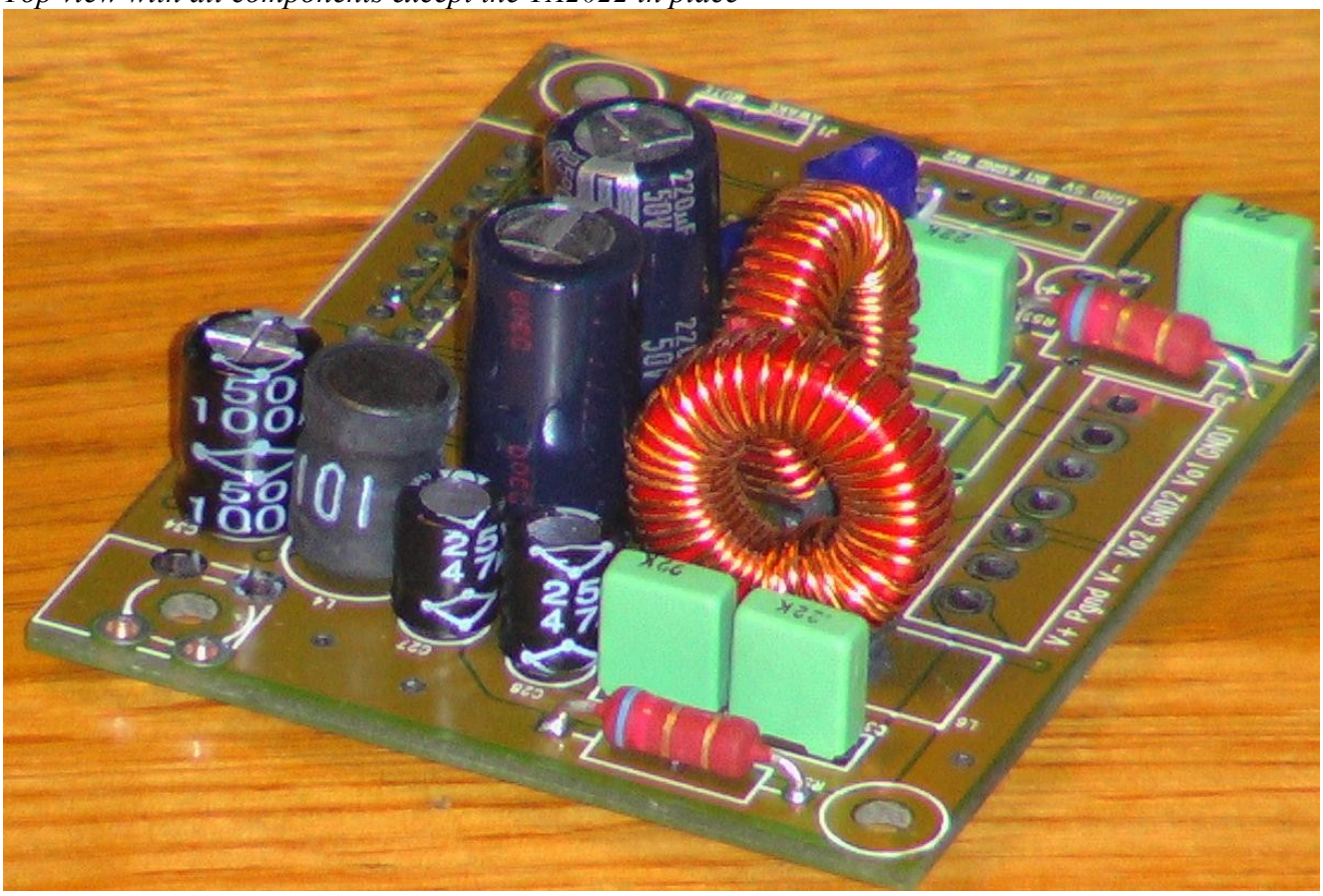
Trimming and testing

- Put the jumper J1 to Mute
- Ground the signal inputs (connect IN1 and IN2 to AGND on connector J2). Do NOT connect speakers yet.
- Connect a power supply of +/-20 to +/-30 volts (+/- 35V absolutely maximum). *Check the polarity of the power supply. Wrong polarity will most likely permanently destroy the TA2022 chip.*
- For testing, use 300 mA slow blow fuses on the power supply. Note that you should only use these for low power testing. It is best to have the fuse on the AC side in such a way that both rails de-power in case the fuse blows. In this way, you will not subject the chip to power on one rail only. You may need slow blow fuses in case you have big bulk capacitors.
- Connect a well stabilized +5V power supply and its ground to J2. Use a shielded cable.
- WARNING: *Wrong polarity or voltages above +6V may permanently damage the chip.* A 100 mA fast fuse, placed *before* the voltage regulator, is recommended. If you have big bulk capacitors for your +5V supply, you may need slow blow fuse. Fuses have a substantial resistance, especially near their current rating. Therefore, the fuse should be before the voltage regulator, not after.
- Turn on the power supplies. When muted, the +5V should draw about 20mA while the main rails should draw 0.5 to 2 mA per side
- Move the J1 jumper from Mute to Awake
- The +5V supply should draw about 50 mA when the amplifier is awake / un-muted. The main rail positive should draw about 20 mA and the negative about 50 mA without signal (current is a bit higher on the negative side, because the VN10 gate voltage is generated from the negative rail).
- Check the fuses. If they have blown, disconnect the board and check all components and solders.
- 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. It should be possible to trim it to below 10 mV
- Shut of the power
- Connect the speaker wire leads to J3. Note that also the speaker ground leads should be connected here, not to the power supply or other ground connection.
- Connect a signal source with its ground leads to J2. It is important to use a shielded cable. Note that also the signal source ground should be connected here, not to the power supply or other ground connection.
- Set the volume very low and check if you get any sound.
- If everything seems OK, switch off power, replace the power supply fuses for larger fuses and try again (The +5V fuse should remain 100mA)
- 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. The amplifier may dissipate a maximum of about 20W of heat at full power with 4 ohm speakers. Then, a 2ºC/W heat sink is reasonable. The heat slug on the back of the chip is connected to the chip ground and does not require electrically insulated mounting.
- Enjoy!

If you have any questions, write any questions on the forum on our web site http://www.41hz.com , as the questions may be of general interest.

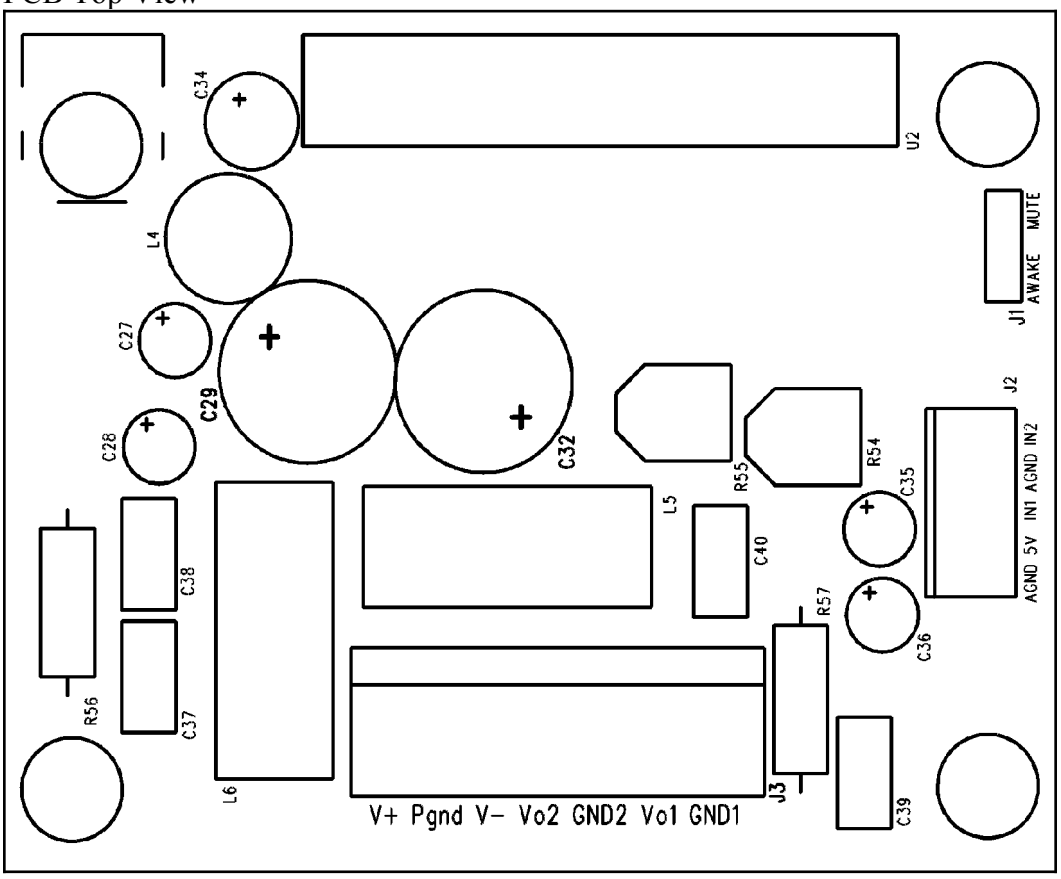
For any questions please do not hesitate to contact us at support@41hz.com .

Top view with all components except the TA2022 in place

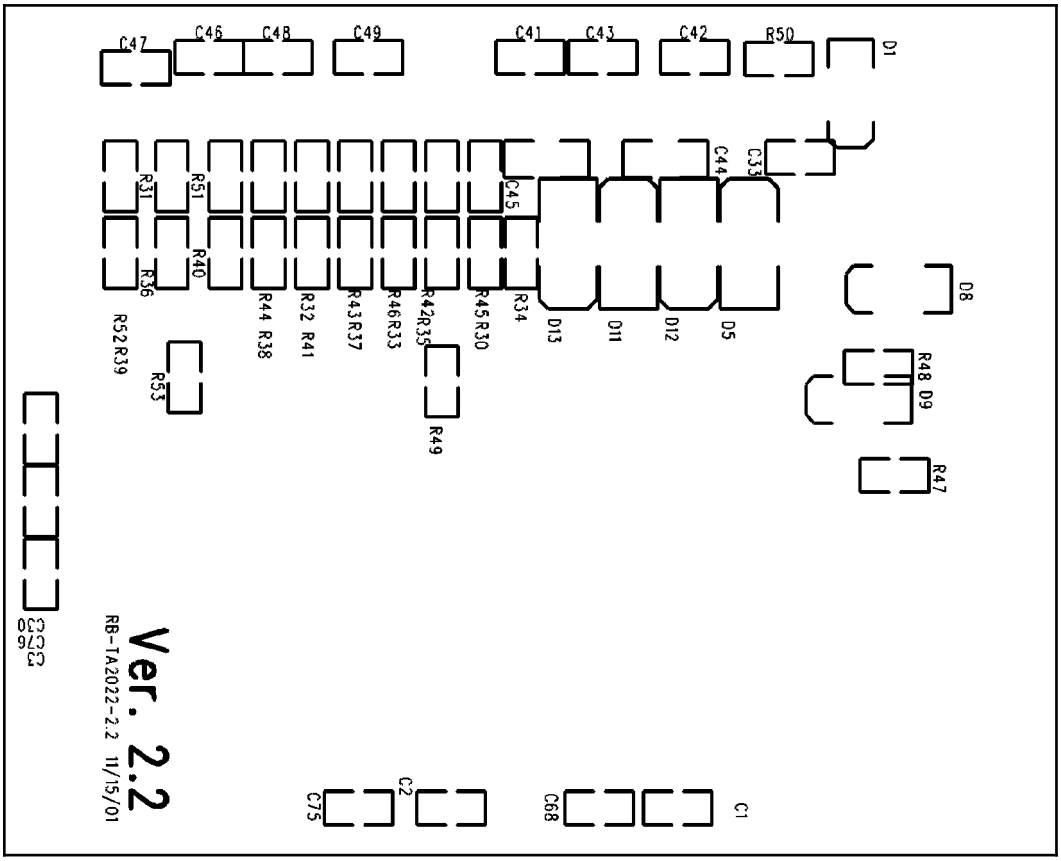


Assembly instructions AMP1 V1.0© 41Hz Audio / Jan Fredriksson

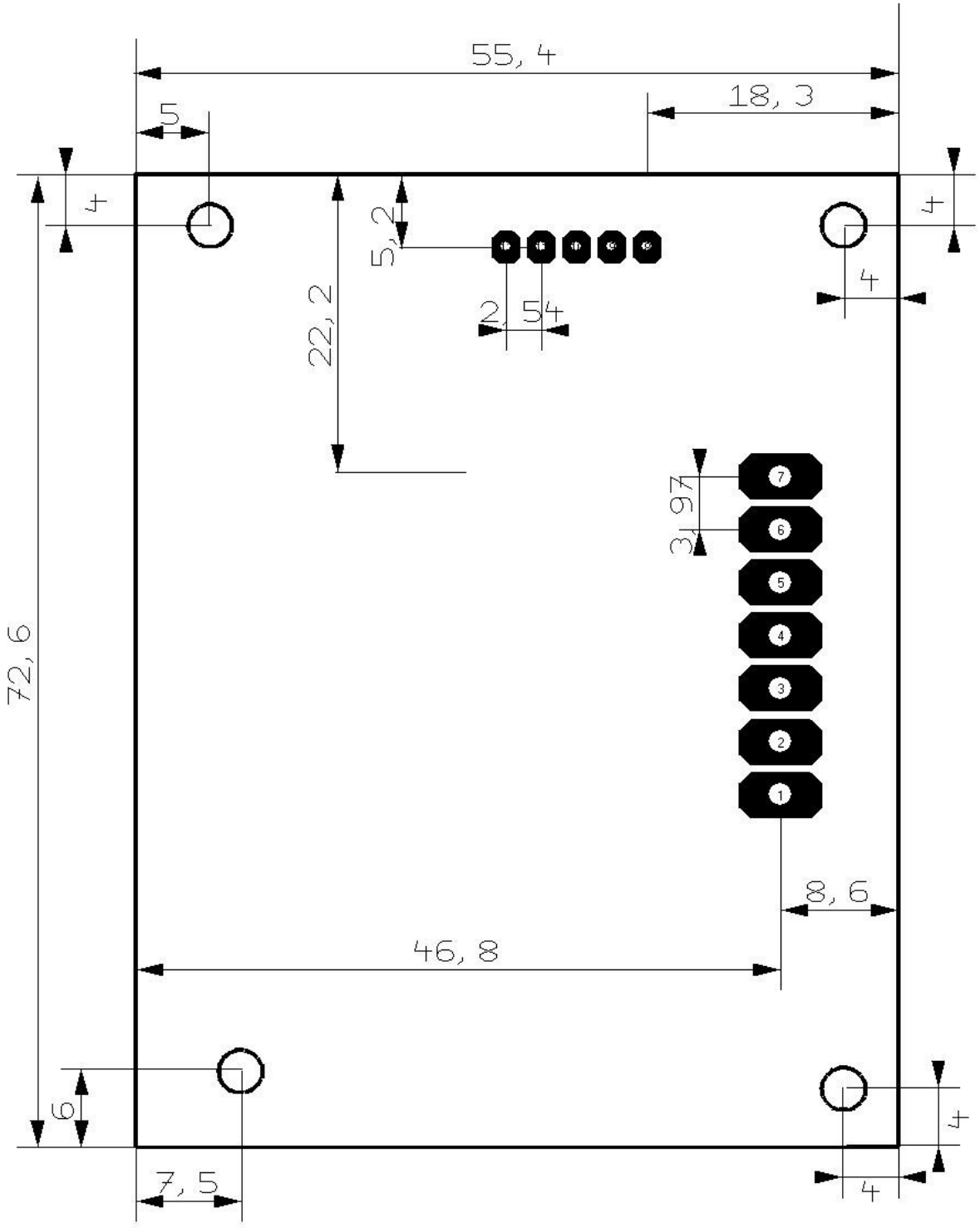
PCB Top View



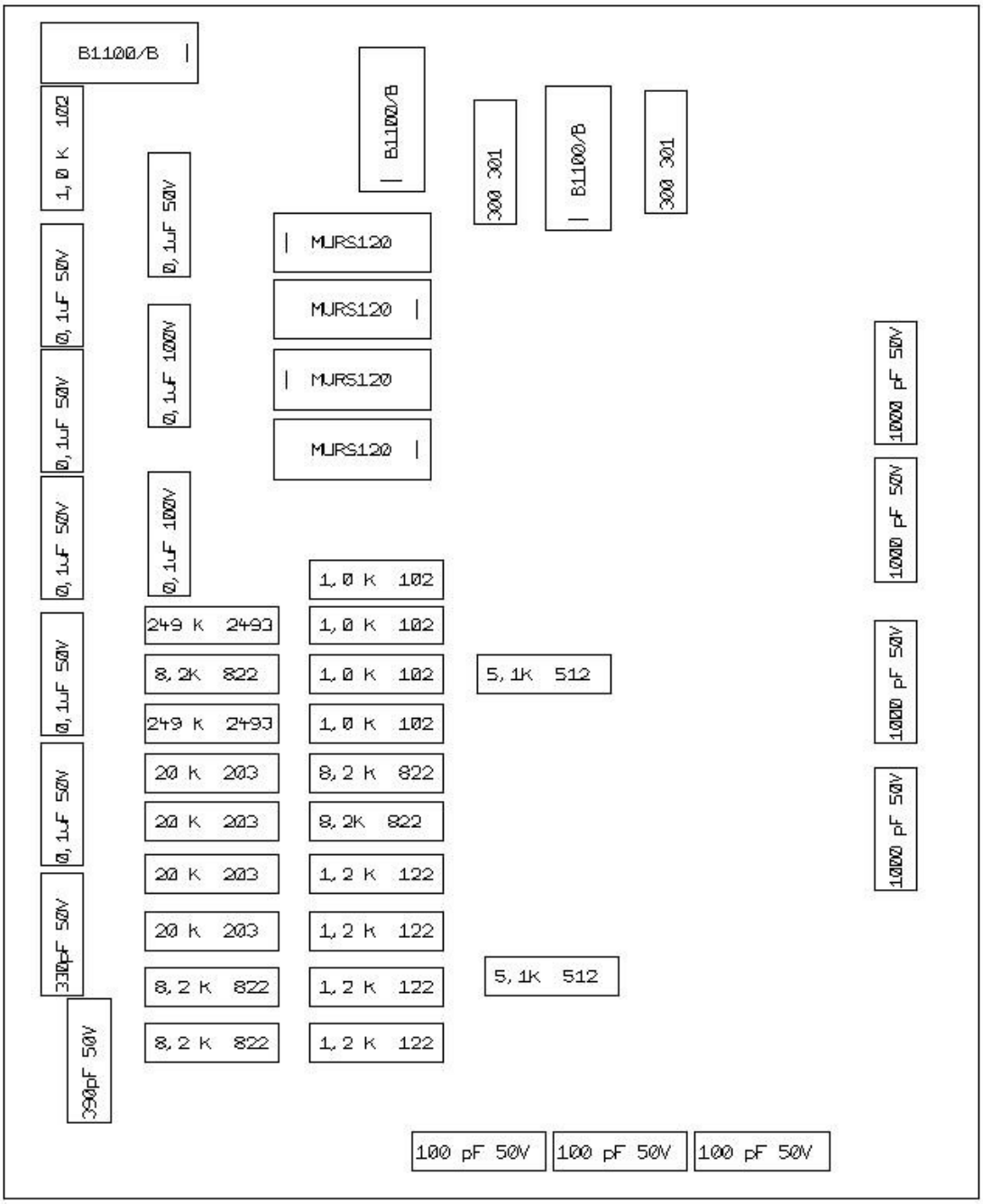
PCB bottom view



Footprint of the board



Bottom side component placement with markings commonly used on the components



APPENDIX 1 BOM

Quantity	Reference	Part
6	C33,C41,C42,C43,C48,C49	0.1uF;50V
2	C44,C45	0.1uF;100V
4	C1,C2,C68,C75	1000pF;50V
3	C3,C30,C76	100pF;50V
1	C47	390pF;50V
1	C46	330pF;50V
2	C35,C36	3,3uF;25V
4	C37,C38,C39,C40	0.22uF;50V
2	C29,C32	220uF;50V
1	C34	100uF;35V
2	C27,C28	47uF;16v
1	D1	B1100/B
6	D5,D8,D9,D11,D12,D13	MURS120
1	L4	100uH
2	L5,L6	Toroid
2 m	Insulated copper wire 0.6 mm	For toroids
4	R32,R43,R44,R51	20KΩ
2	(R32,R43,R44,R51 see table 1)	36KΩ
5	R30,R33,R34,R35,R50	1KΩ
4	R36,R38,R39,R40	1.2KΩ
2	R54,R55	50kΩ Pot
5	R31,R37,R41,R42,R52	8.2KΩ
2	(R31, R52 see table 2)	10KΩ
2	R56,57	5.6Ω;2W
2	R45,R46	249KΩ,1%
2	R47,R48	300Ω
2	R49,R53	5.1 KΩ
1	U2	TA2022

Note. The resistor values are printed on the components. The resistor value is given with three or four digits. The last digit always tells you the number of zeroes to add to the value. Example: **100** means 10 with no zero = 10 ohms. **105** means 10 with 5 zeros = 10 00000 = 1M ohm. **363** means 36 with three zeroes = 36 000 = 36 Kohms.

Surface mount capacitors do usually NOT have any value printed on them, so you must keep the different values apart.