

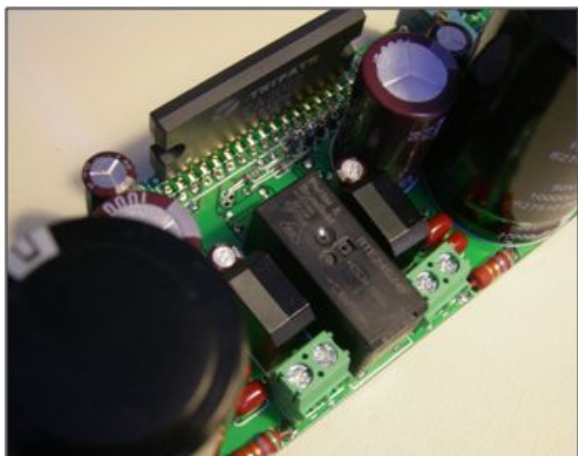
AMP-10-PS-Basic Assembly Instructions

Number of Views: 667

0 Comments Related Sub-Forum: [AMP-10](#).

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

Introduction



Thanks for selecting the AMP10-PS-Basic kit from 41Hz Audio.
This document will help you build the kit. It has been updated for revision 1.05 boards

MAKE SURE you have read the Assembly Instructions for AMP-10-PS (not basic) as well !

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

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

Please read "Considerations before building" FIRST (towards end of this AI).

Building/Assembly instructions

IMPORTANT Components packaged in a shielded, aluminized bag should be considered ESD sensitive and should be handled with ESD care. The Tripath chips use MOSFET outputs which by nature are sensitive to ESD (Electro Static Discharge). Use ESD precautions. Preferably work on a conductive, grounded "ESD mat", and avoid touching the chip leads with your fingers. Discharge yourself before working with the components.

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, pre-amplifiers etc or by the amplifier inputs.

It is therefore recommended that some precautions 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 J1 connector ground, not to the housing or power supply ground.

The speaker returns should lead to J8/J9 ground, not to the casing or power supply ground. It is strongly recommended that hookup cable for the signal input is shielded and as short as possible so that it does not pick up noise from the outputs.

Input cables should lead away from the outputs as far as possible.

Speaker and power cables should be twisted to limit EMI radiation.

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.

Note; while the speaker returns are "ground" you should connect them to the speaker connectors on the PCB, and not to the amp housing.

The same is true for signal inputs; lead the grounds of the inputs to the input connector on the PCB rather than to the amp housing.

The TA2022 copper slug on the back of the chip is connected internally to the chip ground.

Therefore, it is not required to have electrically insulating mounting of the chip to the heat sink. However, it may be better to use insulation for two reasons; one is to avoid ground loops and the other is to reduce EMI from the heat sink.

Some experimentation is recommended and feedback on this would be appreciated.

Important

The 5V regulator has a back metal side that is NOT connected to ground, but to +5Vdc. If you mount this chip to a heat sink, then the heat sink must be isolated.

While you can mount the components in any order, this is a recommended sequence.

1. Start by building the 5V section.

The components are Q5, C51, C150, R100, R80, R81.

J10 connects 5V to the rest of the board. R100 connects the V5 section to the positive power rail. Leave J10 open until you have tested the 5V section and verified that the output voltage is near 5V. If V5 is higher than 5.5V, the Tripath chip can be damaged.

You can test the 5V by connecting the positive rail to any positive source of 10V to 30V.

2. Solder all components on the bottom side of the PCB

Except C44 and C45 which will be mounted later with the Tripath chip.

Note that the diodes have a direction!

3. Continue with the smallest components on the top side.

I.E. resistors first (read the Considerations section before doing this).

Note that all electrolytic capacitors (round cans) have a polarity. Generally, the minus is marked on the capacitor, while the positive lead is usually longer than the negative. The positive pole is indicated on the PCB is quadratic when possible, while the negative is usually round.

Work yourself up to larger components.

I suggest you save the large capacitors until last.

4. Mount the Tripath chip, together with C44 and C45.

These are 100V rated capacitors that are indicated by circles and lines on the bottom side of the board.

These two capacitors have 5 mm lead spacing and should be mounted in the same holes as the Tripath chip, pin number 4+8, 9+12 respectively.

These capacitors are critical and need to be mounted as close as possible to the chip, after cutting their leads short. Make sure the Tripath chip is horizontally and vertically aligned to the PCB.

5. Mount the large capacitors, cables / connectors and any other component you have forgotten earlier.

Power up and test

Do not connect or solder anything while the power is connected.

When you have verified that the 5Vdc is OK, you can close J10 by soldering a short wire in place.

Use a low fuse value or current limited power supply for testing. The amp needs no more than 200mA per rail for testing.

1. Connect the power to the two power connectors (but not speakers and signal source yet).

Turn on the power. If fuses are OK, measure the voltage on each big bulk capacitor to see that it is OK.

The relay should have clicked when power was supplied. The amp should now draw around 100mA on each rail

2. You can mute and open the speaker relay by closing J2.

If you want, you can wire this to a mute switch on the front panel of your amp box.

When J2 is closed, the relay should click and the current go down a bit.

3. If all seems OK, then measure the voltage on the output (after removing J2).

By turning the trimmers R54 and R55 you can zero the output voltage. It should be possible to have less than 10 mVdc, max 25mVdc on the speaker outputs.

4. If all is OK, connect speakers and signal input, turn on the power and enjoy the music!

Minimum requirements to build a working amplifier

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

- A suitable power supply, typically a Toroid with 2x18Vac 150VA upto 2x22Vac, 300VA
- Heat sink, connectors

Additional components

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

- Transformer.

A toroid with a nominal voltage of 18VAC to 24VAC is recommended. The fuse should not be higher rated than the transformer maximal

nominal load.

- 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 which are included.
- A mute/un-mute switch is recommended for thump-less power-on.

You can wire this to a switch on your front panel. Optionally use a 2.5 mm jumper (50 mil) on the board.

- You can optionally fit a volume control

Considerations before building

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

Power supply

The AMP10-BASIC requires a dual rail power supply plus a stabilized +5V 100 mA supply.

The rail voltages could be +/-20Vdc to +/-30Vdc. Higher voltage gives a higher possible maximum power output.

With 4 ohm, +/-30Vdc is the maximum recommended by Tripath, to avoid tripping the over-current shutdown at high power.

With 8 ohm loads, +/- 35Vdc can be used without tripping the over-current limit.

Toroid transformers are available with standard values. Nominal voltages of 18VAC to 22VAC are common. When rectified, this will give about +/- 24VDC to +/-30VDC.

Note that transformer voltages are given for full rated power. At low power, the voltage will be slightly higher. For maximum power a 2x22VAC transformer would be about ideal.

Mains power is **lethal!**

If you are not professionally qualified to work with mains power, get help from someone who is!

On the board there is a regulated supply for the +5Vdc.

It uses a LM317 voltage regulator and taps from the positive main supply. The +5Vdc should not draw more than about 60 mA.

A jumper, J10, can break the +5Vdc supply which can be useful when testing. The +5Vdc is used for the analogue input amplifier and the in-chip digital parts of the TA2022.

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.

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 output signals, the input stage may clip.

The amplifier input stage in the Tripath chip is of the inverted operational amplifier type. The gain is calculated as: $\text{Gain} = -1 * R_{\text{feedback}}/R_{\text{in}}$.

On the board,

R3+R4 are the Rin

and

R43+R44 are the Rfeedback.

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 type. I recommend metal film resistors.

<i>R-in</i>	<i>R-feedback</i>	<i>Input gain V/V</i>	<i>Suitable signal source</i>
22kΩ	47 kΩ	-2.1	Direct connection of portable MP3/CD player with built in volume control or via a passive volume pot.
22kΩ	22 kΩ	-1	General use
47kΩ	22 kΩ	-0.47	(Pro) preamplifier with fairly high output signal

Table 1: *Input stage gain setting recommendations*

Modulator gain

The "modulator gain" is the power stage voltage gain. You can select this to match your power supply voltage.

The modulator feedback resistors are R31, R37, R41 and R52. The supplied value for these is 8.2kΩ.

These are suitable for rail voltages 25-30V. There are also 10K supplied, suitable for rail voltages 27-33V.

If you use other resistor values than the supplied, you should use 1% tolerance resistors, preferably of the metal film type.

For details on selecting modulator feedback values, see the TA2022 chip datasheet from Tripath, , but the spreadsheet in this post will be -very- convenient to help out:

[Input and modulator gain](#)

Summary on selecting the gain.

- Select input gain to match the music source signal level
- Select the modulator gain to match the supply rail voltage
- The total gain is the input stage gain multiplied by the modulator gain.

Just remember that music signals are very dynamic by nature, signal levels are approximate and impossible to predict accurately.

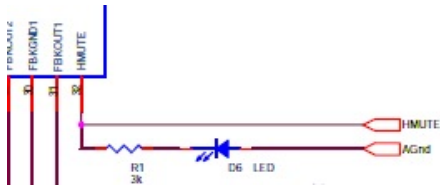
Mute and Mute Status (from forum entry: <http://www.41hz.com/forums/showthread.php?p=10&Basic-Mute>, not checked yet)

1. TA2022 MUTE pull-down resistor (R1)

This should be 1kΩ or less (absolutely NOT 56kΩ as given in the Bill of Materials).

The reason is that the MUTE input is a logic level input which is expecting 0V or near it or 5V or near it in order to function correctly.

2. The MUTE circuit which drives the Speaker relay and MUTE/UNMUTE LEDs



This circuit takes its input from the TA2022 HMUTE output, which is a logic output.

When the amplifier is muted this output is high, which turns on Q2 causing it to pull down the voltage across Z1 and C60, which turns Q1 off and the associated LED off.

It also causes Q3 to turn off resulting in current flowing through R6 and the associated MUTE LED.

When the amplifier is unmuted the HMUTE output goes low, turning Q2 off and allows C60 to charge up through R2 until its voltage is clamped by Z1.

This causes both Q1 and Q3 to turn on, thereby allowing current to flow through R7 and through the UNMUTE LED and also causing the relay latching current to flow to ground, thus activating the relay and connecting the amp to the speakers (it also turns off the MUTE LED).

With this circuit operation in mind, it becomes clear where the component values need to be altered from those given in the Bill of Materials:

1. Zener series resistor and time delay resistor (R2)

This resistor performs 2 functions: it charges up capacitor C2; and it provides sufficient current to allow Z1 to behave as a zener diode (ie clamping the voltage).

With a 25V nominal supply rail, there will be about 5V across Z1 once C2 has charged up and been clamped. With a value for R2 of 1kΩ, this gives a zener current of $20V/1000\text{ohms}=20\text{mA}$, which is greater than the 5mA minimum required for correct operation.

In fact R2 can be anything between 1k and 3k9 but if it is as high as 56kΩ which I have seen mentioned elsewhere in the forum, it will ensure that the zener does not operate correctly.

The second function of R2 is to provide a time delay in conjunction with C2 to ensure that the speakers are not connected immediately on power-up or unmute.

With R2 value of 1K and C2 of 47uF, this gives a 0.3 second delay (to 25V which it never gets to because of the zener) which is probably insufficient. With R2 of 3k9, the delay is over a second, which is better but still not very long so I have increased C2 to 220uF as well.

2. MUTE LED series resistor (R6)

When Q3 is off (muted) current flows through the relay coil, R5 (120R), R6 (8k2) and then the MUTE LED. This current must be sufficient to light the LED but not to latch the relay. With a 25V supply there will be about 2.5mA flowing through the LED which is on the low side.

If you solder a 10kΩ resistor across the existing 8K2, this increases the LED current to about 5mA which makes the LED considerably brighter but doesn't latch the relay.

3. UNMUTE LED series resistor (R7)

When Q1 is on (unmuted), current flows through R7 (8K2) and the UNMUTE LED. This means there is approx 3V across 8K2 which corresponds to an LED current of about 0.3mA, which explains why the LED isn't very bright! For decent LED brightness, with a current of 5mA, R7 should be about 1kΩ.

If you don't want to remove the existing 8k2 you can solder a 1kΩ resistor across the existing one.

3. Speaker Click when AMP-10-PS-BASIC is un-muted

I would be interested to hear if anyone else suffers with this.

I suspect it is caused by the relay coil after the amplifier is unmuted and could be fixed by using a suppression diode directly across the coil.

Troubleshooting

Read article under Tech Info, Building Kits.

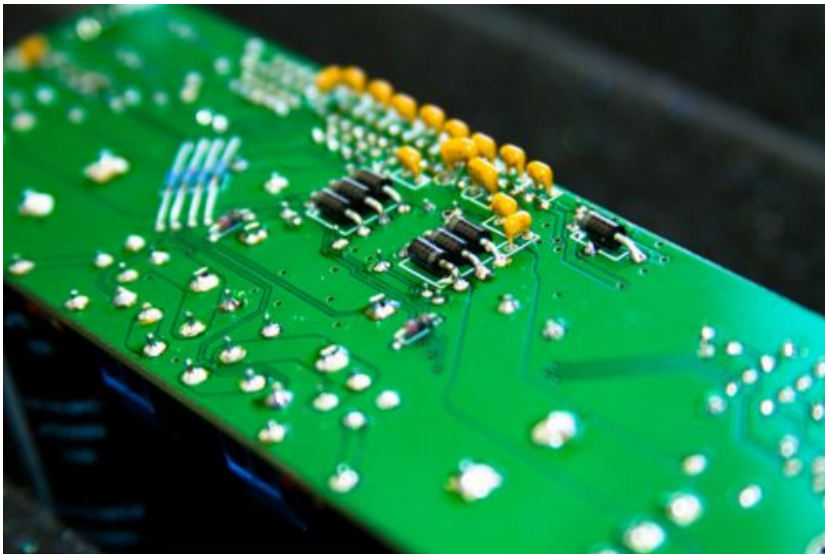
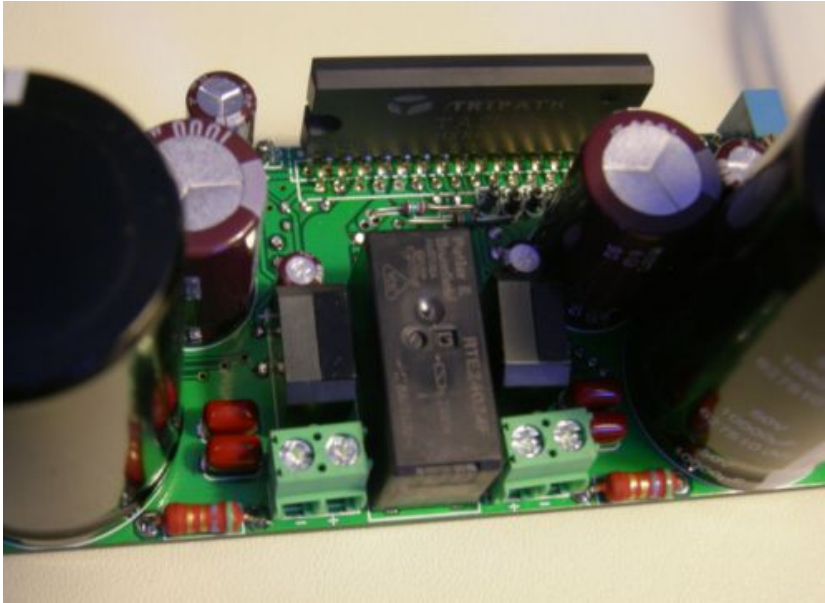
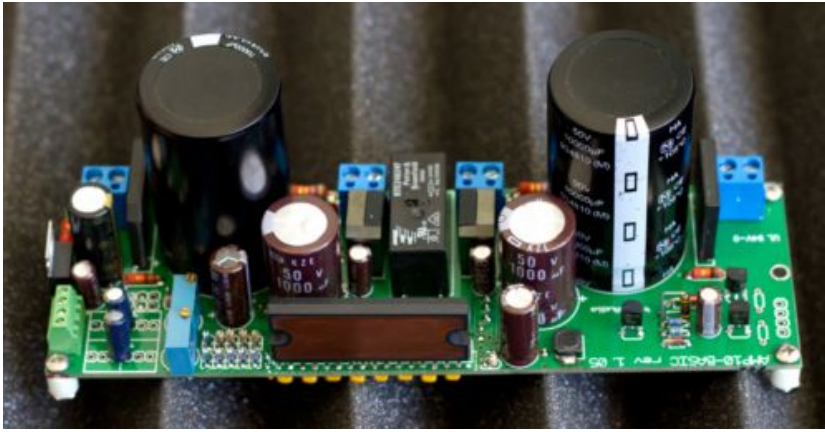
AMP10-basic tweaks + tips

....

Revision history

- 1.0
- 1.....

AMP-10-Basic views



Categories: AMP5, AMP10

