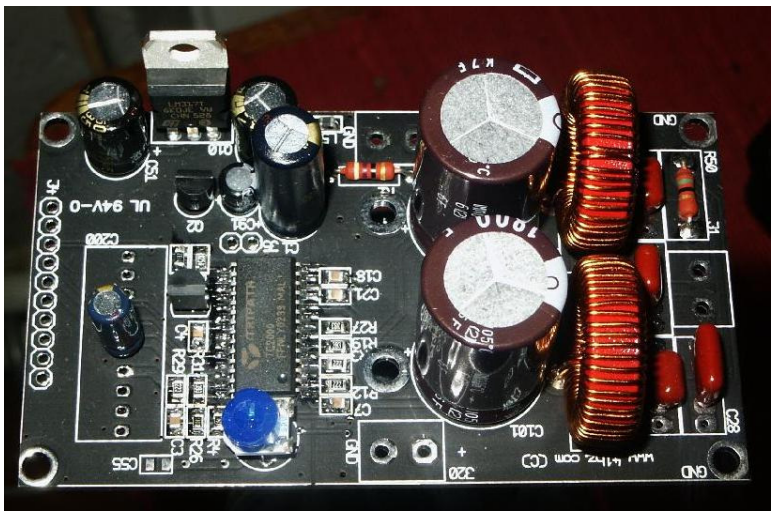
AMP11 ASSEMBLY INSTRUCTIONS

Assembly Instruction | **Revision 1.0 November 2009**

Document ID: **88**

Written by Jan Fredriksson



AMP11 FEATURES

- Bridged mono output
- Signle rail supply, max 32V
- Regulated supply not required
- Footprint, 50x80 mm
- Line level analogue audio input.
- Sensitivity and gain adjustable with external resistors
- Power supply for +5V on the board.
- Mute function for click-less on-off
- Over / under voltage turn off
- Over current protection, temperature overload protection
- The module is suitable for monoblock amplifiers and active speakers

TK2050 FEATURES

- Class-T architecture
- High Power >100W @ 4Ω, 10% THD+N
- 0.005% THD+N @ 70W 4Ω
- High efficiency means small heat sink and moderate transformer requirements
- Mute Input
- Over-current protection
- Over and under-voltage protection
- Bridged output
- Dynamic Range >100 dB

Table of Contents

Introduction

Read through the whole assembly instruction briefly before assembling anythingIntroduction

The assembly instructions are always under development.

The assembly instructions are gradually moved from files like this, to the TecDoc wiki site http://wiki.41hz.com/ You can contribute your own ideas and improvements there! Please do!

You where sent the latest downloadable 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 that may be needed

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. In many cases, if you mount the Tripath chip to an aluminum amplifier casing, this is sufficient to cool the chip. The Tripath chip does not need to be electrically insulated, as the back of the chip is internally connected to ground.

- Mute/un-mute switch. A jumper on the board can be used to mute / un-mute the amplifier. Preferably wire this to a switch on your casing panel. Muting the amp before power on minimizes the turn on thump and is recommended.
- A power supply. AMP11 is intended to be used with an external single rail DC power supply or a 24V battery. The maximum recommended voltage is 36V, being more comfortable values. Typically, you would use a 150VA or more power supply. You can use power supplies with lower power rating, but the amplifier may not perform as well and the power supply may get overloaded.

More information on various considerations and possibilities can be found in the appendixes. It is recommended that you read through these quickly before mounting any components.

Errata

1. There is rail voltage range sensing in the TC2000 chip. On early schematics the voltage sensing thresholds sensing resistors value of R52 and R54 are given as 22K and empty. With these values the amp will work OK, but voltage sensing is disabled. If instead both R52 and R54 in place, as in later BOMs / schematics, the voltage sensing is active. With 200 K for both R52 and R54, the voltage sensing limits are about 17 to 32.5V and with 220K 19V to 35V. 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. *Note that the over-voltage limit sensing gives no hard protection! It just gives an indication, by muting the amp, if the voltage is out of bounds.*
2. In some deliveries, the value of R22 and R30 (marked 1K on bag nr 4) was delivered as 750 ohm, but must be changed to 1K.

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

Building and Soldering tips

The boards for AMP11 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

The most important advice:

- Do nut rush. Let the construction take its time. Have patience and do it well. If you rush, you will most likely be disappointed with the result. Accept that building electronic circuits requires craftsmanship, which takes time to learn and years to master.
- Get good tools. It is frustrating to use low quality tools.
- If you do not have a good temperature controlled soldering iron, then borrow one.
- Get some fine solder wire, some solder flux and some solder wick. It saves your nerves and eliminates a lot of frustration.
- Read through this entire manual, at least briefly, before actually building the board.

AMP11 basics

AMP11 is a mono amp with potentially excellent data. It is a mono amp without power supply. Everything
The final result should be an amp with excellent sound, if built properly. AMP11 is a pure mono amp, suitable for plain mono amps or multi-channel true “monoblock” amps.

At the core of AMP11 there are two chip; Tripath TC2000 and TP2050. The TC2000 is the analogue input section, digital signal modulator and supervision. The TC2000 works with no voltages above 5V. The TP2050 is the output / power stage. It takes the modulator input from the TC2000 and contains output power transistors. The TP2050 chip is rated 40V absolute maximum, with 36V being the very maximum recommended rail voltage. A four volt safety margin is needed to account for overshoots, power supply instabilities etc etc. The TP2050 has two paralleled power stages, with a total current handling capacity of about 10A. This is quite sufficient for 4 ohm loads at up to 36V or 2 ohm loads at up to 20V.

Note that the standard AMP11 is shipped with 35V close-up capacitors, which should not be used above 32V rail voltage. The 35V capacitors can be substituted for 50V capacitors on request (before shipment!). The 50V capacitor will have lower capacitance than the 35V equivalent.

On the AMP11 board there is a voltage regulator for +5V, so no external +5V is needed. The +5V is generated in two stages, the first taking rail voltage, dropping it to about 9V, the second stage drops voltage from 9V to 5V. It is very important that the +5V is stable and clean, and the two stage design helps achieving this. The +5V regulator is current limited to 100 mA. The amp needs about 70-80 mA.

Mounting the components

There are basically two different approaches you can use.

The fastest is to build the amplifier in any order you find convenient, and when everything is ready; hook up test. This method is recommended only to very experienced builders, and requires a lot of experience in troubleshooting in case something goes wrong.

A slower and safer way is to build and test along the wa. The instructions below are structured that way.

Wind the toroid inductors

This may be tiresome to the fingers so you may just as well start now, and take a break now and then.

There is a separate document about winding toroids available on the www.support.41hz support site.

With the T80-2 size inductors included in the AMP11 kit you should use about 55 turns of wire to get 15 uH (14-16 uH is acceptable). Wind as tight as you can. Tight winding minimizes HF signal leakage from the inductors. With the supplied 0.6 mm wire, you can wind the 55 turns in a single layer before coming back to the starting point. If you have an inductance meter, measure the value and adjust if required. Do not mount the toroids to the PCB yet.

Mount the Tripath chip

Mounting the chip is best done now when the board is free from other components. The chip are probably the hardest part to solder so you might as well do it at the beginning, when it is easier to repair, and at worst, you can get a new chipset and replace it.

Mount the surface mount decoupling capacitors

Mount/solder all the surface mount capacitors.

Mount the surface mount resistors

Now mount /solder all surface mount resistors.

Connect the two ground planes

The power and analogue ground planes are connected at one point only. This is done via the choke L5. Solder it to the PCB.

Build the +5V section

The +5V is stabilized in two stages. The first stage takes it input from the main rail and has an output of around 9V. The second is from 9V to at 5V. The +5V is separated from the board by the J5 jumper, so you can test the +5V section separately (optionally, you can connect en external +5V at J5). To get the +5V section up and running, you need to mount the following components.

+9V

R2, Q10, R56 and R57. When these are in place you can test the +9V. R2 connects to the rail, R56 and R57 set the voltage. Connect a voltage source of between 12V to to 32V. **The power must be with correct polarity. Connecting power with wrong polarity will destroy the Tripath chip.** Measure that you get 9V out of Q10 (the heat sink of Q10 is also output). (The main power connectors are J20 or J21; these two are equivalent. The polarity is printed on the PCB).

+5V

Mount Q2, R110 R111. Now you should be able to test and have +5V out of Q2, for example over the pads for C1. Do not proceed any further until you have +5V working. It is vital for the function of the amp.

When +5V is OK, you can mount the capacitors for the 9V and 5V. These are C50 , C51, C1 and C91.

Mount the +5V supervisor U10. This supervisor ensures the amp can not be unmated before the +5V is up and stable at startup.

Output section components

The output section consists of a filter that removes high frequency switching patterns from the speaker output. As the output is bridged you have two filter sides to ground. Mount C36, C32, C25 and C28. The R50 resistor is mounted vertically. Prepare the toroids by cleaning off the insulation at the wire ends. A good way is to use a small gas torch and then clean the wire ends. Pre-tinning them with some solder is a good idea as this reveals if the wire is not well stripped. Put the toroids in place, apply solder flux, hold the wire ends with pliers when soldering so you can pull the wire ends and pull the toroid snugly to the PCB. You can also put a small amount of temperature resistant silicone glue or similar under the toroids prevent them form moving and working loose.

Finishing the board

Mount and solder the trimmer resistor. On the 1.00 revision boards the positive pad of C101 and C102 are not properly connected but floating. You can check this by measuring from this pad to the positive rail connector a J20 or J21. The positive rail is a wide lead, straight from J20 to J21 under the chip. The fix is to scrape a little of the solder stop varnish around the C101 and C102 positive pads (the rectangular pads) on the *bottom side* of the PCB and make a solder bridge there.

Mount the input capacitor. A 3.3 uF Panasonic FC cap is provided but there is plenty of space for alternate capacitors of your choice.

If you will use the status LED outputs, solder the status LED drivers Q4, Q5, Q6 in place.

Trimming and testing

Always de-power the amp before doing any work! Never connect or disconnect signal connector or speaker cables with power connected as this can damage the amp. The Sleep jumper is the ONLY exception.

Powering up for the first time

See the appendixes for connector pin-outs.

1. Connect power to J20 or J21, via a fuse of 500 mA or less. **MAKE SURE POLARITY IS CORRECR. Connecting power with wrong polarity will damage the board components and destroy the Tripath Chipset.**
2. Do not connect speaker / load yet.
3. Turn on the power. If the fuse blows, check the power and output section until the problem has been solved.
4. Make sure the +5V section is tested and OK. Close J5 by soldering a wire or jumper in place.
5. Turn on power. The HMUTE output / LED should now be on.
6. De-mute by connecting MUTE to MUTE_REF (via a switch). The HMUTE output should now toggle to low / OFF. If not, there is an error detected bt the chip. This could be an over / under voltage or other internal errors.
7. If all is OK, de-power, close the signal input to signal ground (i.e input signal=0.000). Still with no speakers / load connected, turn on the power and, turn the trimmer until the speaker output signal is below 10 mV from zero.
8. De-power. Connect speakers. Connect signal source.
9. Turn on, de-mute and enjoy the music!

APPENDIXES

PROBLEM SOLVING

There is s separate generic troubleshooting guide available in the support web site.

If you have any questions, comments or feedback, please write specific questions on the support site http://www.support.14hz.com. Other amp builders can also help via the forum on the web site http://www.41hz.com. You can of course also contact us at jan@41hz.com

Considerations

1. On the board there is an input capacitor, C200. It is required, as the amplifier is internally biased to about +2.5V. A 3.3 uF capacitor is provided with the kit. The board provides space for input capacitors with hole spacing from RM 2.54 (100 mil) to RM22.5 (800 mil). The input capacitor forms a high pass filter together with the input resistor Rin. The cutoff frequency is $F=1/(2*\pi* Rin*Cin)$ For example, with Rin = 22 Kohm and Cin = 3.3 uF, the cutoff frequency is $F=1/(2*3.14*22*0.0033) \approx 3$ 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. If there is a separate woofer in your system, you could use input capacitors of 1 uF or even smaller. Input impedaces of 10K to 100K are common in amplifiers. A lower input impedance puts higher current requirements on the signal source. A higher impedance can mean increased THD+N.

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=$-1*R_{feedback}/R_{in}$ [V/V]. The minus sign is due to the fact that the input stage is inverting. On the board, R29 is the R_{in} and R31 is the $R_{feedback}$. With the kit, there are 22 KΩ resistors and 56 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.

Rin	Rfeedbk	Input Gain	Suitable signal source
22 KΩ	56 KΩ	-2.5 V/V	Portable MP3/CD player with built in volume control.
22 KΩ	22 KΩ	-1 V/V	General use
56 KΩ	22 KΩ	-0.4 V/V	Preamplifier with fairly high output signal

Table 1. Gain setting recommendations. R29 is the R_{in} and R31 is the $R_{feedback}$

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 mute function. Therefore, using a switch for the mute function is recommended rater than using a power switch.

Connectors of AMP11

J1

Speaker connector. Note that the output is bridged, and no speaker connector should be connected to ground!

J20, J21

Power connectors. The tow connectors are equivalent, you can use either. Note the polarity marked on the PCB. **Connecting power with the wrong polarity WILL damage components, including the Tripath chipset.**

J4

Signal inputs / outputs

1. Signal input
2. Signal Ground
3. MUTE switch reference (Output from U10, not equivalent to ground)
4. MUTE (to switch)
5. +5V (To external circuit, max 20 mA, or from external +5V supply, in case onboard +5V is not used)
6. HMUTE, mute indicator LED
7. OVRLDB indicator LED (indicates input overload)
8. FAULT LED (indicates faults in power stage)
9. THWARN (Warns for power stage over-temperature, at above 130ºC. Power stage shuts off at 150ºC)
10. AGND. Ground for LEDs

J5

JSeparates +5V regulator section from the rest of the board. Leave open until +5V section has been tested and confirmed. Then close with soldered wire.

Schematic of AMP11

The schematic is available on the download page of the web site. Please also see the Tripath reference design data sheet and the TK2050 datasheet.

BOM (Bill Of Materials)

The BOM is available on the downloads page of he web site

Component placements

Top side and bottom side pictures are in a separate document

AMP11 Specifications

Table with typical data. For full specifications, see the Tripath data sheet for the TK2021B chip

	Min	Typical	Max
Supply voltage[1]	9V?	24-32V	36V
Over-current shutdown[2]	7A	12A	16A
Signal/Noise Ratio[2]		103 dBA	
THD+N[2]		0.005% (70W / 4 ohm / 30V)	
Channel separation		95 dB	
Efficiency[2]		85% -90%	
Output power 4 ohm[2] @30V rail		75W <0.01 THD+N, 115W 10% THD+N	
Output power 8 ohm[2] @30V rail		35W <0.01 THD+N, 60W 10% THD+N	
Offset voltage		Adjustable below 10 mV	
Output noise[2]		135 µV	
Input impedance		Set by external resistors. Nominally 22 to 56 K	
Power stage gain		Set by external resistors depending on rail voltage	
Thermal warning[2]		130ºC	
Thermal shutdown[2]		150ºC	

[1] Supplied capacitors are 35V. For rail voltages above 32V these should be changed to 50V capacitors.
[2] Tripath specifications for the chipset.

Comments and feedback to this assembly instruction is highly appreciated!!!!