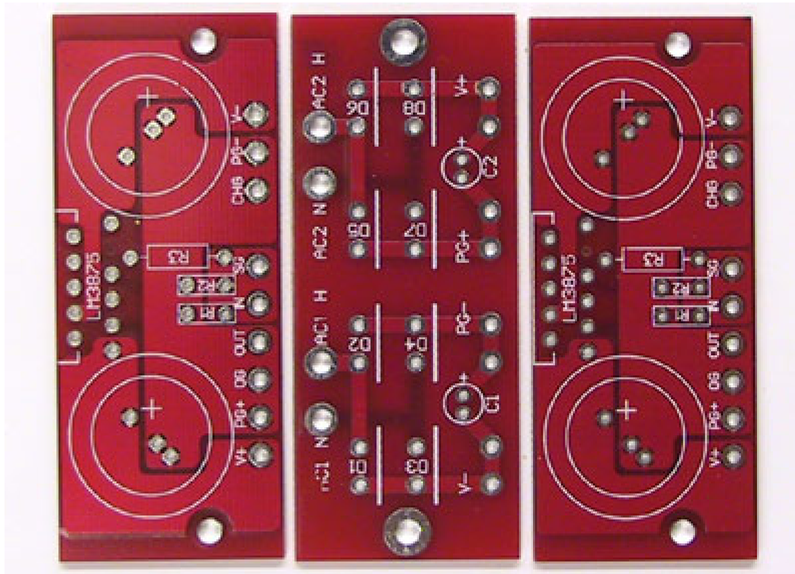
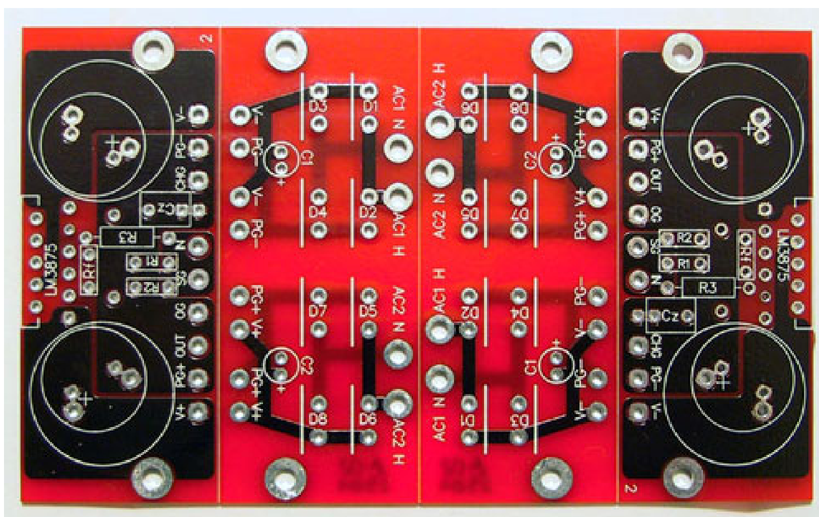
Users Guide for the Non-Inverted LM3875 Kit
(also known as the Gainclone kit)



Picture of Rev A PCB set



Picture of Rev B PCB set

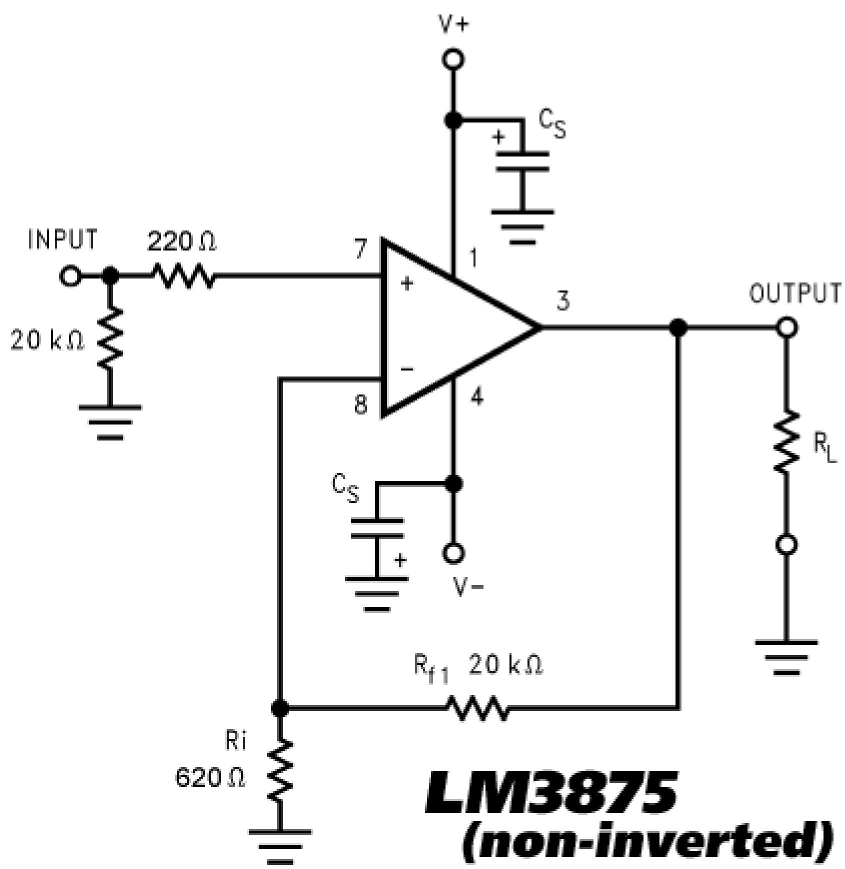
1 Introduction

Thanks for participating in the Non-Inverted LM3875 kit/PCB group order. The goal of this kit is to get easy to build, good sounding DIY amplifiers into the hands of as many people as possible. I am tired of seeing countless people start projects and never finish them. I have done this myself, but this kit should be simple enough to build, that you shouldn't have an excuse for not finishing it in a reasonable amount of time.

It should be noted that the currently available kit uses a Rev B board. Additions to this board have been taken from feedback from those who purchased the Rev A boards, as well as the design guidance from members of the diyAudio.com community. Major changes to the Rev B board include the addition of a second rectifier board, allowing the construction of true monoblok amplifiers without purchasing additional boards, the ability to mount the feedback resistor on the PCB directly and an area for a Zobel network to be integrated directly on the board. Most of the pictures contained in this document are of the Rev A board, but changes where pertinent are included for the Rev B board. A comparison is shown on the previous page of the 2 boards.

1.1 History

The term gainclone is based on the 47 Laboratory 4706 Gaincard. After rave reviews of this amplifier in Japan, it was determined that the main component of this amplifier was a readily available $5 chip, made by National Semiconductor. This chip, the LM3875, when properly implemented creates a high quality amplifier. This started a new trend in DIY amplifiers, as this is the first time that such a high quality amplifier could be made so easily and for such little money.



1.2 Terminology used in this document

Non-Inverting – Operational Amplifier (opamp) topology used in this kit. An opamp has 2 input terminals. If the input goes into the +input, then it is generally non-inverting, meaning that the output is in phase with the input.

Negative Feedback (NFB) – A technique used in most power amplifiers to set the voltage gain. The output is fed back into the negative input on the amplifier. The effect of negative feedback is to cancel out distortion and negate the effect from non-ideal characteristics. The NFB loop should be as short as possible. This leads to the NFB resistor not being placed on the PCB, and soldered directly to the device pins. On the Rev B board, the option of including the NFB resistor is included, but not required and the builder is encouraged to still solder the NFB directly as described in the following instructions.

LM3875 – Amplifier IC used in this kit. It is available in 2 different packages, insulated LM3875TF, and un-insulated LM3875T. The insulated package is usually preferred, due to the ease of use. Some people say that the insulated package also sounds better. The actual Gaincard uses the LM3875TF.

PCB – abbreviation for printed circuit board

Gain – Ratio of output divided by Input, expressed by: $A_v = \frac{v_{out}}{v_{in}}$

Gainclone – Amplifier using a chip amplifier, such as the LM3875 used in this amplifier

2 Building Instructions for the Kit

2.1 Premium Kit Contents

Description	Quantity
PCB containing board for 2 channels and 2 rectifiers	1
LM3875TF – National Semiconductor chip amplifier	2
1500uF 50v – Panasonic FC Capacitor	4
22k ohm – Caddock MK132 resistor (R_2 and R_{NFB})	4
680 ohm – Riken Ohm 0.5w resistor (R_3)	2
220 ohm – Caddock MK132 resistor (R_1)	2
MUR860 – On Semiconductor fast diode	8
4.7uF 50v – Black Gate N-type capacitor	2

2.2 Basic Kit Contents

Description	Quantity
PCB containing board for 2 channels and 2 rectifiers	1
LM3875TF - National Semiconductor chip amplifier	2
1500uF 50v - Panasonic FC Capacitor	4
22k ohm – Phoenix SFR16S resistor (R_2 and R_{NFB})	4
680 ohm – Panasonic Carbon Film 0.5w resistor (R_3)	2
220 ohm - Phoenix SFR16S resistor (R_1)	2
MUR860 - On Semiconductor fast diode	8
4.7uF 100v – Panasonic FC capacitor	2

The PCB for this project is scored, so it will easily break apart in to the boards required for a 2-channel stereo amplifier.

2.3 Soldering Tips

(You are welcome to skip this section if you know how to solder)

My recommendation, if you have never soldered before, is to find a friend to teach you, as experiencing it first hand would be the best way to learn. It is important to have a decent soldering iron, as you don't want one that gets too hot or not hot enough. If your soldering iron is too hot, it can ruin the components on the boards, so if you are using an unregulated iron, I would not exceed 30 watts. I like to use a regulated Weller WPTCT soldering iron, but it can be a rather expensive purchase if you won't be soldering a lot. Also make sure that your iron gets hot enough, as cold solder joints cause bad connections.

Recommended soldering procedure:

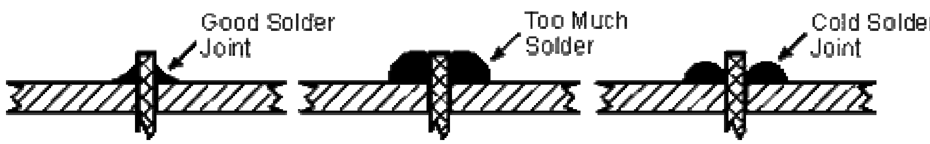
1. Simultaneously apply the tip of the iron and the solder to the circuit board so that they touch both each other and the wire being soldered to the solder pad. (working on the bottom side of the board)

2. When the solder begins to flow, remove the solder and hold the iron on the joint until the solder flows and bonds to the wire and the pad.

3. Pull the tip of the iron up so that it slides up the wire, leaving a nice smooth soldering joint.

4. Next, you will want to cut the excess wire off. Be sure to trim the wire just above the solder joint. You do not want to cut into the solder joint.

Here is a picture that I found on the web of a good soldering joint, and 2 bad ones:



Additional tips:

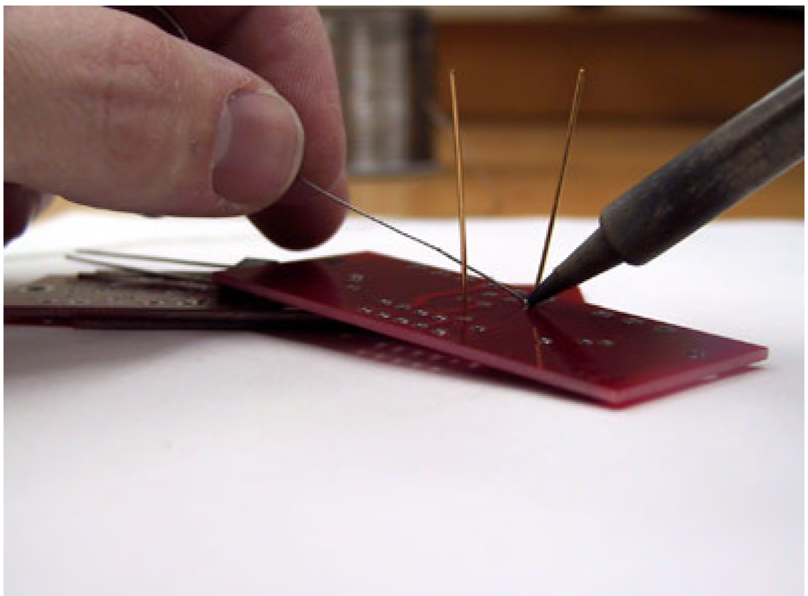
From Peter Daniel: Liquid rosin flux might also work well. For anything that didn't come out nicely, you can touch with a brush wetted in flux and touch with the soldering iron. This will make it look nicer and remove any solder excess.

From Sandy: If mistakes are made, or too much solder is used, desoldering braid is a very helpful tool. Simply hold the desoldering braid up to the soldered joint, and apply heat with the iron for removing unwanted solder. I prefer to use flux along with this to make the process go more smoothly.

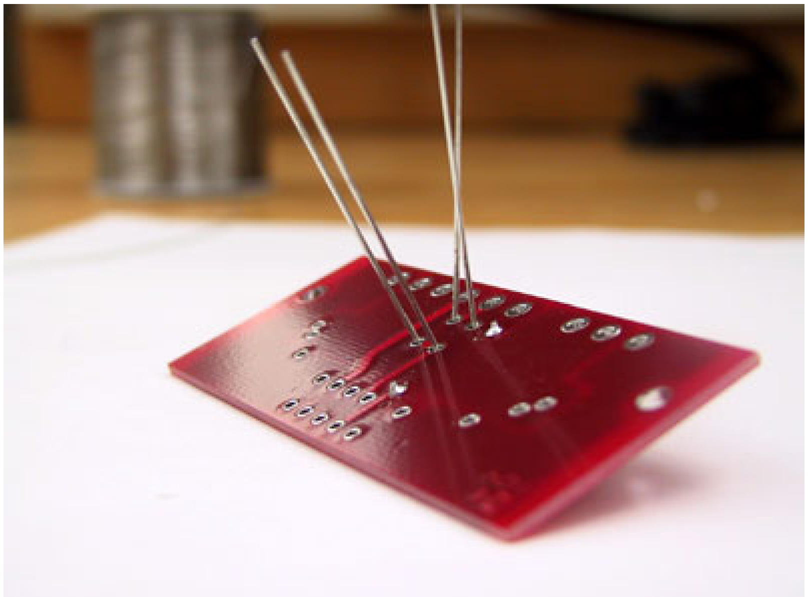
2.4 Assembling the Amplifier PCB

There are a total of 6 components to be soldered to the PCB (3 resistors, 2 capacitors and one LM3875), and one resistor to be soldered directly to the pins on the LM3875 or to the PCB when using the Rev B boards. Here is the recommended procedure for putting the PCB together:

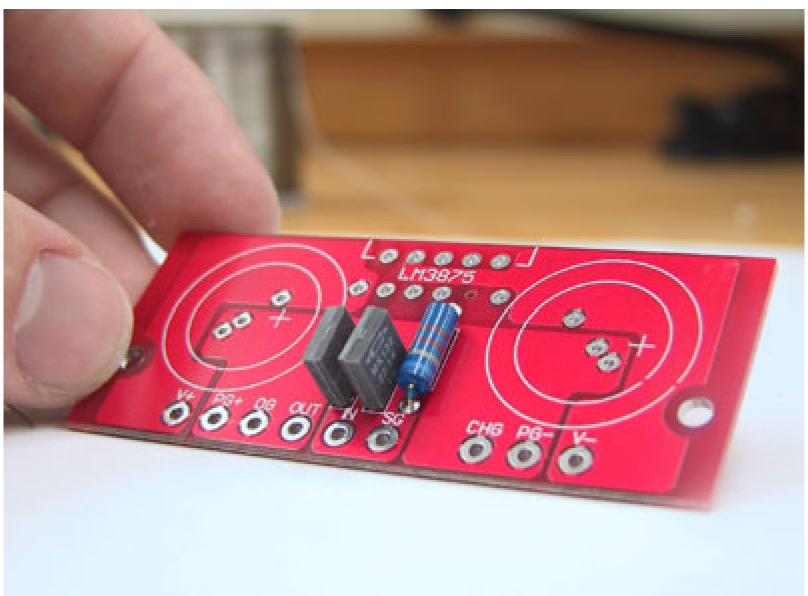
1. Since R_3 has the lowest profile on the PCB, start with this component first, following the soldering process detailed above.



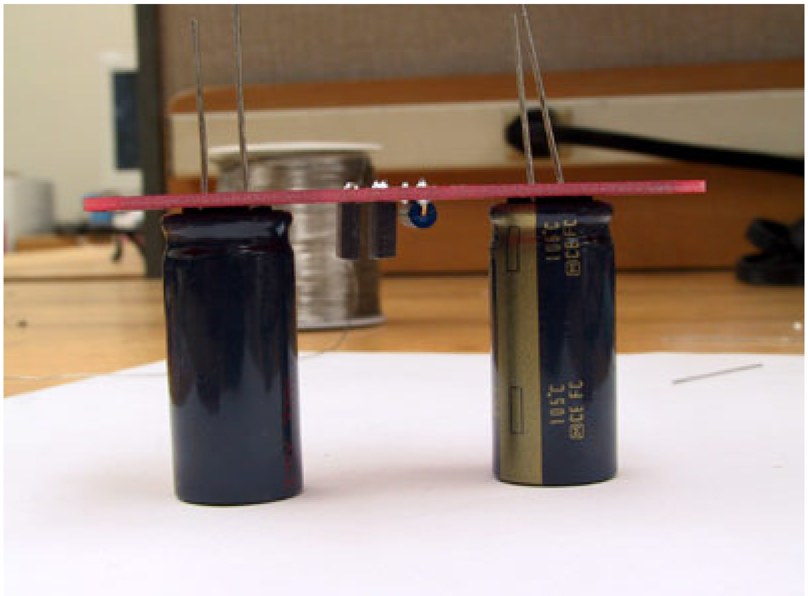
2. Next, solder R_1 and R_2 onto the pcb:



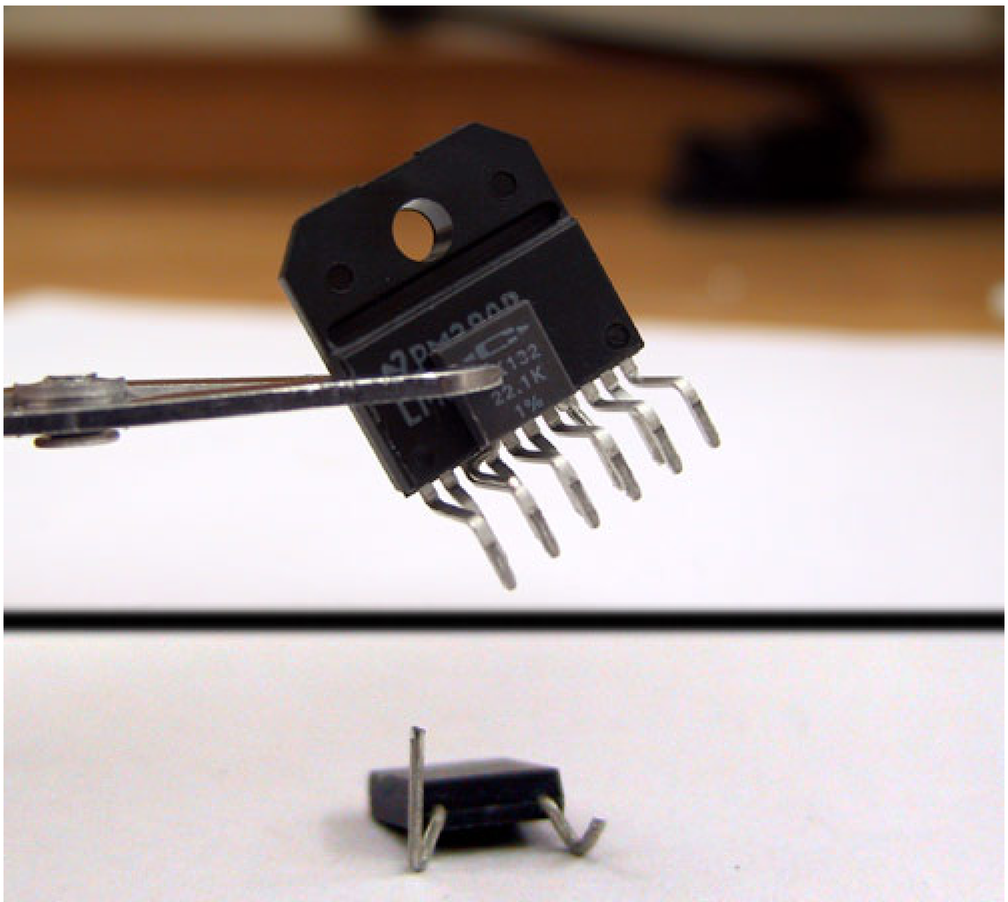
3. The PCB should now look like this:



4. Next, install the capacitors C1 and C2 (not labeled on PCB) using the correct polarity as noted by the '+' on the board. Note that there are three holes instead of two due to optional capacitors having different lead spacing. Use one negative and one positive hole which fits the capacitor being used.



5. The next step is to connect the negative feedback resistor (NFB), R_{NFB}. This resistor may not placed on the PCB, as it is the **most** important component in this amplifier, and the shortest possible signal path for this component will result in the best possible performance of the amplifier. For the Rev B boards, the option of installing this resistor in the board is available, but direct connection is still recommended. There are two options for attaching the NFB resistor directly, place it directly on the LM3875 package, or install it on the underside of the PCB, connecting to the leads of the LM3875 protruding through the PCB. The resistor, R_{NFB} is connected from **pin 3** to **pin 8**. The first method is more difficult, but provides a shorter signal path. Beginners may way to stick to the second method or to attaching it on the Rev B PCB. Here are pictures of how I attached the resistor to the LM3875:

   Attaching the resistor to the LM3875

   As you can see, I used a clamp to hold the resistor to the LM3875, while I soldered it to the package. A normal alligator clip will work fine for this.

I had my fiancé put a board together. She had never soldered before, and had no problem getting the board assembled correctly. I had her solder the RNFB resistor directly to the bottom of the PCB. Here is a picture of her board:

The components used in the picture above are less expensive components, but the build process is the same. Soldering the resistor to the bottom of the board or attaching in the Rev B board will be easiest for most beginners.

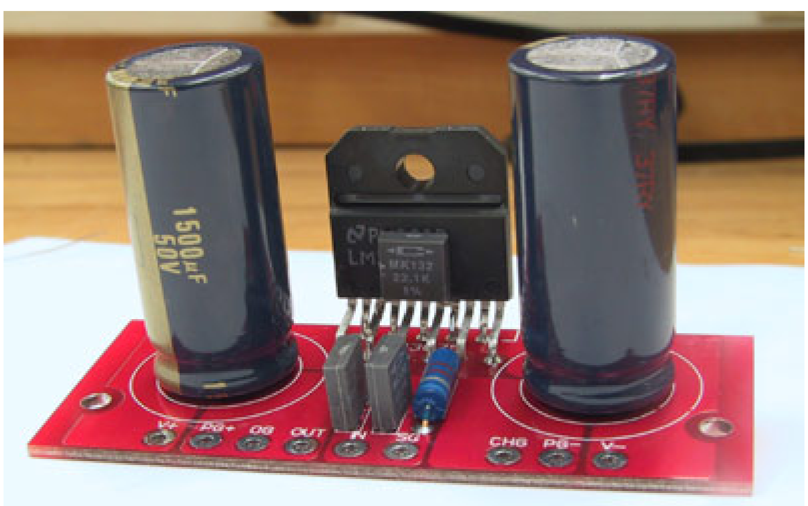
6. The next step is to solder the LM3875 to the board. On the PCB, pin 9 is a bit smaller than the rest of the holes. This is an unused pin on the LM3875, which will be used to make the soldering process easier for the LM3875, by holding the chip in place while it is being soldered. The plating was also removed so that there was more clearance for the trace running next to the hole (+input trace).

   Insert the LM3875 IC into the PCB, applying pressure on pin 9. Pin 9 will be a tight fit, but will help position the chip properly. Keep the chip elevated a bit from the board. Here is a picture:

   With the board upside down, supported by the capacitors as shown above, solder the LM3875 into place.

7. Your board should now be finished and look like this:

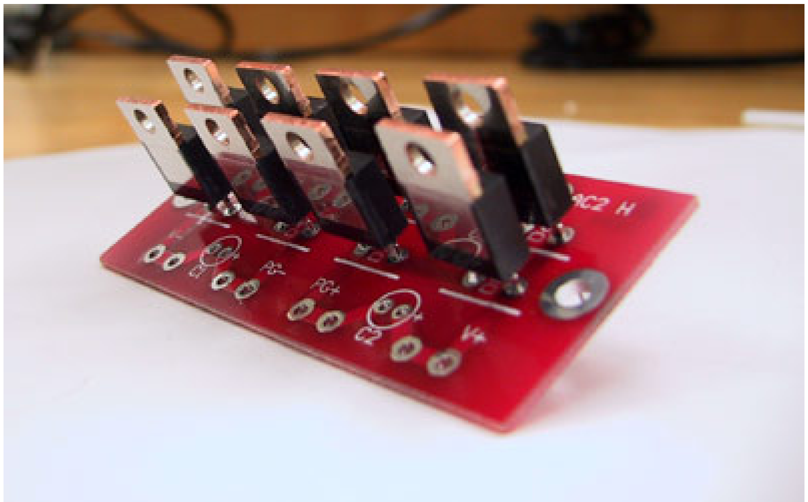
2.5 Assembling the Rectifier PCB

The rectifier is quite simple to install, as there are just 8 MUR860 diodes and 2 capacitors. If assembling monobloks using the Rev B board, this procedure will be duplicated for both rectifier boards. Here is the procedure that I followed:

1. On the PCB, the white line indicates the metal tab on the package. It is important to line up the package correctly, or it will not work properly.

   On the MUR860 diode, the left side is tied to the cathode, which is also tied to the back tab of the case. The right side is the anode. For soldering, I like to solder the right anode side first, since you can hold onto the case without burning yourself. Solder the right side, then position the diode in place properly, while heating the right pad. Once you get the positioning correct, solder the left side.

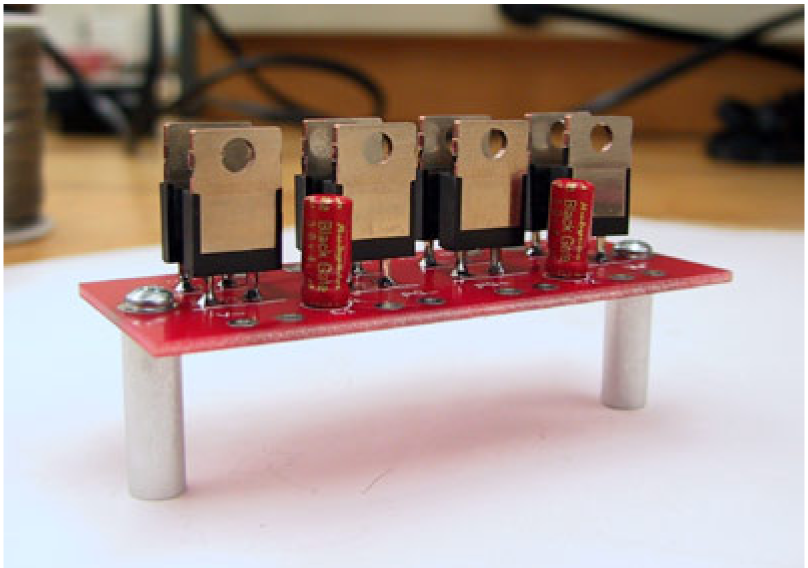
   Here is a picture of the diodes, soldered in place correctly:



2. The last 2 components, the capacitors need to be soldered in next. Note that the longer lead indicates the positive side, which is indicated on the PCB by a + symbol.

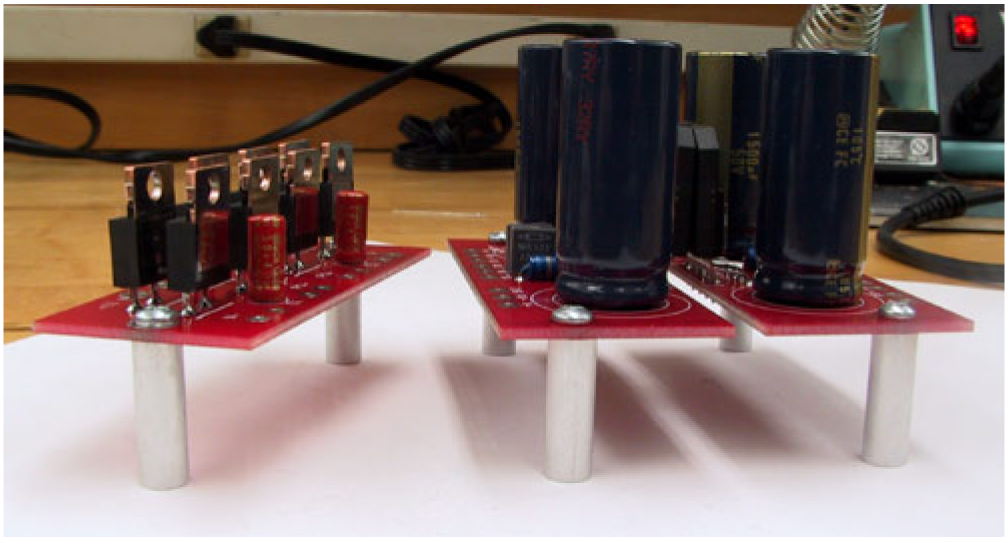
   Note: Peter Daniel suggests that the BG N non-polar capacitors be installed with the short lead in the + side (essentially reverse polarity for a non-polar device, not causing any risks). Do NOT do this for the Panasonic FC capacitors, as they will possibly explode, since they are polar capacitors.

3. You should now be finished with the rectifier PCB, and here is a pictures of a finished board:



2.6 Picture of finished PCBs

You should now have all your boards finished for your amplifier and now be ready to assemble them in to a working amplifier. Here is a picture of a set of finished PCBs, ready for assembly:



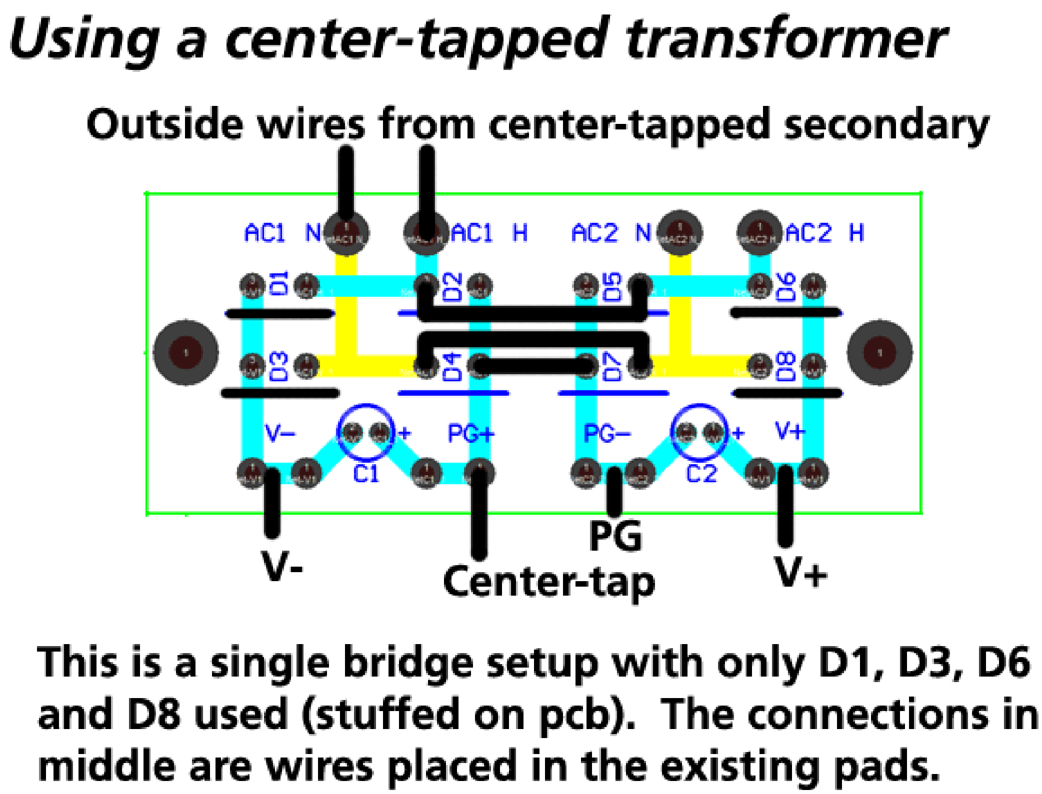
Once the soldering process is complete, I like to clean them up with some rubbing alcohol and a plastic brush. This will remove the rosin flux from the solder, and make the board look better.

3 How to Turn the Assembled Kit into a Working Amplifier

3.1 Choosing a transformer

There are many different options for producing a power supply, but using the supplied diode bridge gives three main options when choosing a transformer. A transformer with dual secondaries, true center tapped secondaries and two transformers with single secondaries. The original standard application of this kit uses the first, a transformer with dual secondaries. For this application, the primaries are attached to the mains, while V+ and 0+ are attached to V+ and PG+ and V- and 0- are attached to V- and PG-. Note that for a stereo amp, one diode bridge can be used for both channels or for monoblok applications, both bridges and two separate transformers can be used.

For the application of a true center tapped transformer, one with only three wires, V+, V- and 0, the following alternate arrangement can be used.



Finally, if two separate transformers are connected such that one transformer connects it V to V+ and 0to PG- and the second transformers V to V- and 0 to PG+.

Using the first example, one can choose a variety of VA ratings and rail voltages. Keep in mind that after rectification, the rail voltages are somewhat higher than the non-rectified AC secondary rating of the transformer. The secondary voltage averages to 1.4 * the AC voltage, minus diode losses. The transformer regulation is also a factor, dependent on the size and regulation characteristic of the transformer in question. Suffice it to say that a commonly used 18V transformer results in approximately 25V rectified, while a common 22V supply produces around 34V rectified.

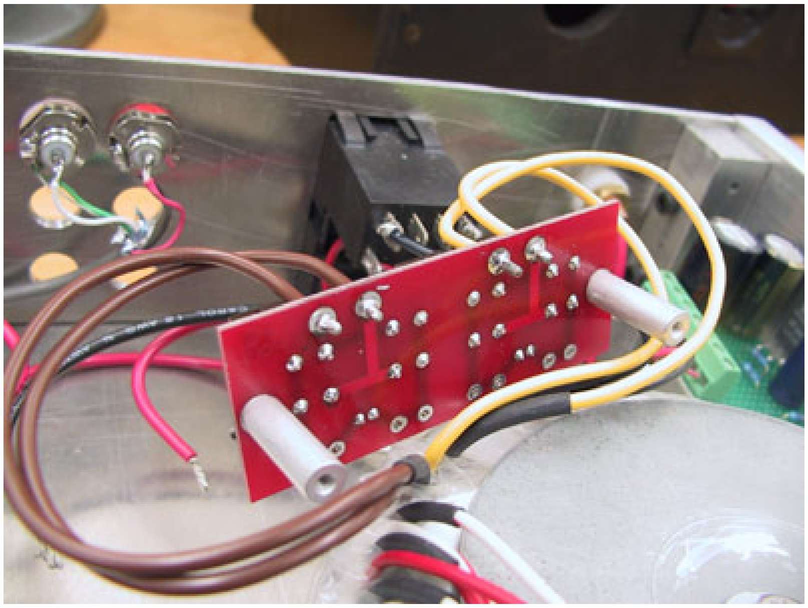
Referencing the National datasheet LM3875.pdf, you can find the practical maximum rail voltage rating for the average impedance of your speaker. The Output Power vs. Supply Voltage chart on page 9 is a good indication of the maximum rail voltages for a given speaker that you will design to. When looking over the curves a speaker with a nominal impedance of 4 ohms tend toward 25V rails with a reasonable margin of safety, while voltages above 35V are still well within the range for 8 ohm speakers. This shows that transformers with 18-22V secondaries are well within reason for many common commercial and DIY speakers. A transformer with 25V transformer secondaries can also be successfully with less of a safety factor.

The VA ratings on transformers is also a consideration. Within reason, a larger transformer has more constant regulation under load, but this chip operates very successfully without extremely large transformers. Many have successfully used 160VA transformers, while the 220VA range seems to be adequate for almost all stereo implementations, not straining the transformer. The price point between 220VA and 330VA, however might lead one to purchase the larger of the two. Anything above this could be considered frivolous for a stereo pair, unless one happens to be on the shelf or in a surplus vendors stock. Don’t be tempted to buy an extremely large transformer such as used for Class-A applications as it is simply not required. The fuse required will be dictated by the size of the transformer, due to the inrush current when power is first applied. A 2 amp slo-blo typically works fine for transformers around 220VA or less, while a 3 amp slo-blo fuse might be required if using a larger transformer.

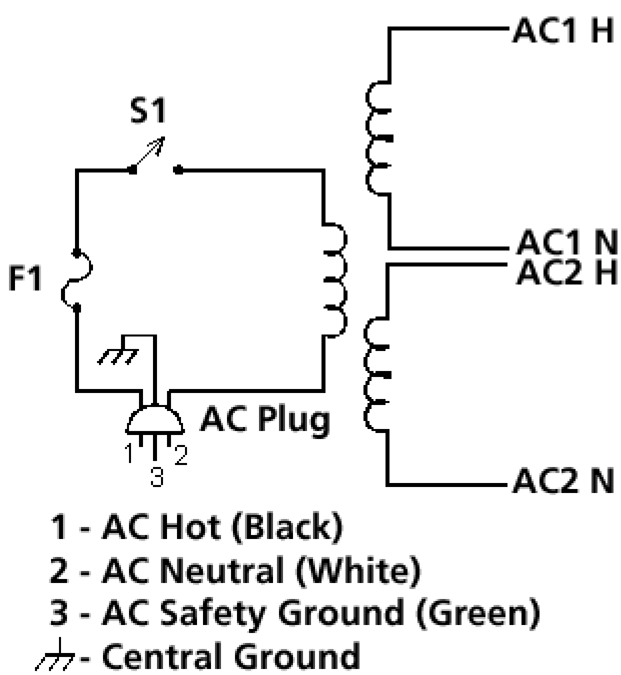
Using an EI transformer or a torrodal transformer is up to the end user. Both have their advantages, but the common availability of torroids and their successful use in many designs makes them a common choice. If you have access to a good quality EI, don’t be afraid to use it.

3.2 Wiring the power supply and installing into the chassis

1. Wire the secondary windings from the transformer into the rectifier PCB.

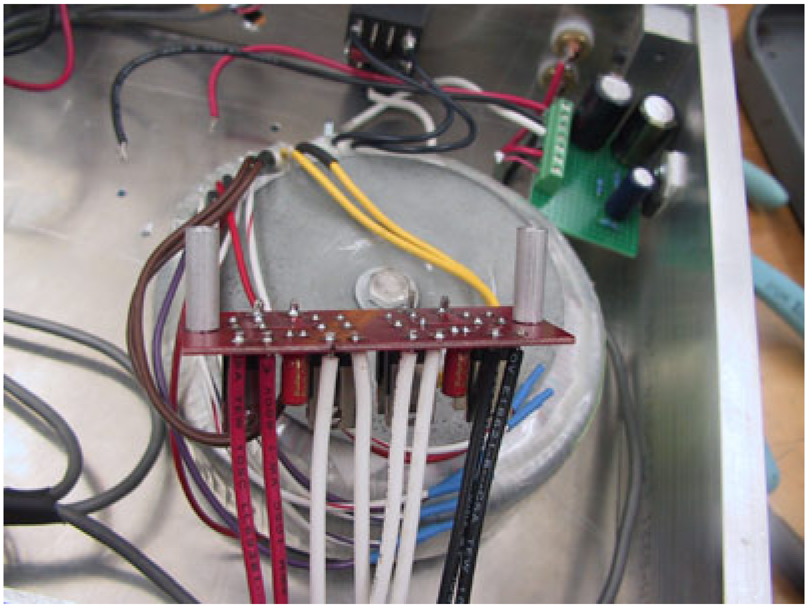


The primaries of the transformer should be wired as shown below with a fuse and power switch. If constructing monobloks, this procedure will be duplicated using one transformer and rectifier board per monoblok.

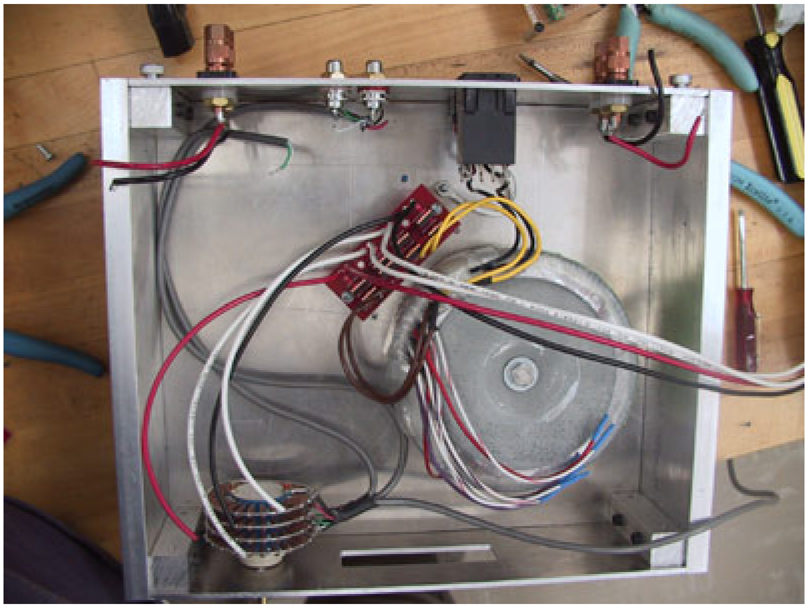


IMPORTANT: Electricity is dangerous, so be careful not to have any loose wiring, and measure the voltage with a working multimeter.

2. Once you have verified that the voltages from your rectifier board are correct, by measuring from V+ to +PGND and V- to –PGND (should be the same), solder wires to the rectifier board for running the power supply voltages to each channel. It is helpful to choose a color scheme for your wiring, so that things are not improperly wired. Reversing the voltage supply rails on the amplifier can ruin the LM3875 IC. I choose Red for +V, White for PGND and Black for –V. Here is a picture of the output wires soldered to the rectifier board:

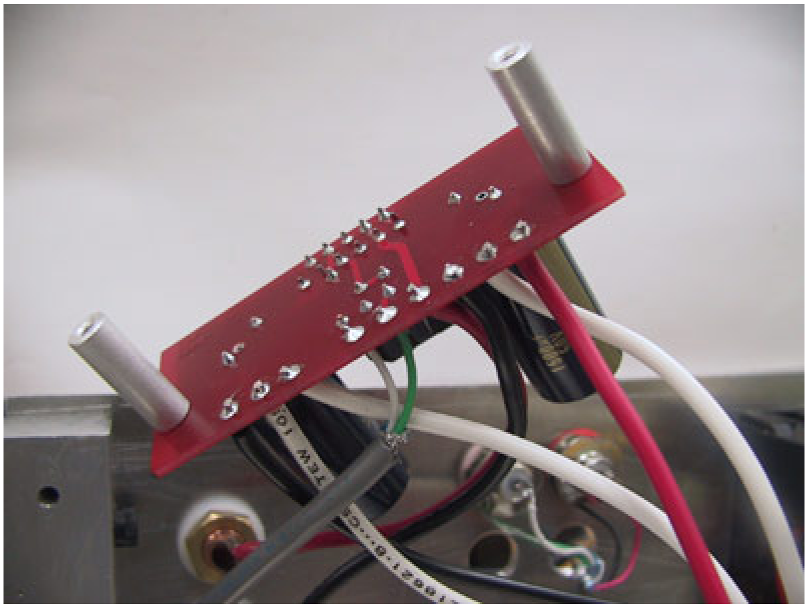


3. Next, route the power supply wires in your chassis. You will also need to run a ground wire from your chassis ground to each channel (not shown):

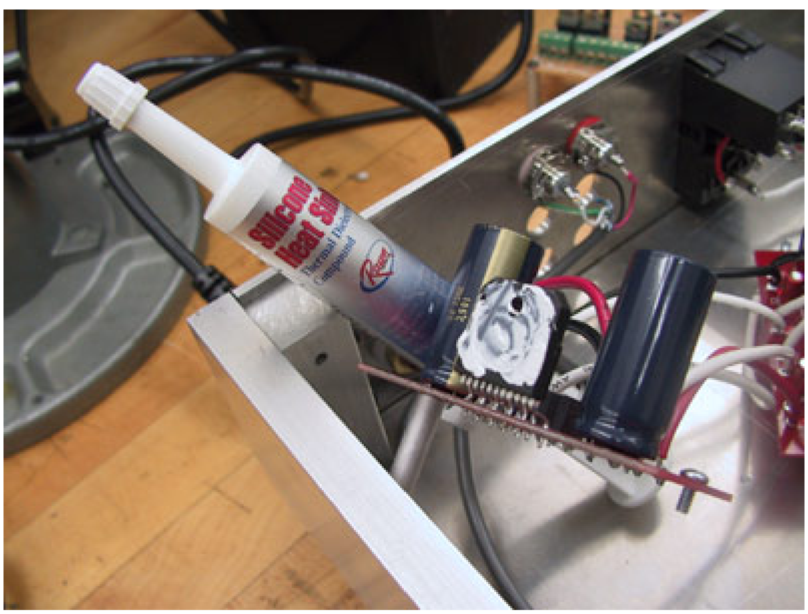


The chassis ground essentially go to what could be referred to as the star ground. There will be 3 wires tied here, one each channel, and AC ground connection.

4. Next, solder the power supply wires, along with the input and output wires into the PCB for each channel. Be sure to wire the polarity of the input and output correctly. Wiring the input backwards might cause the input to be shorted to ground.

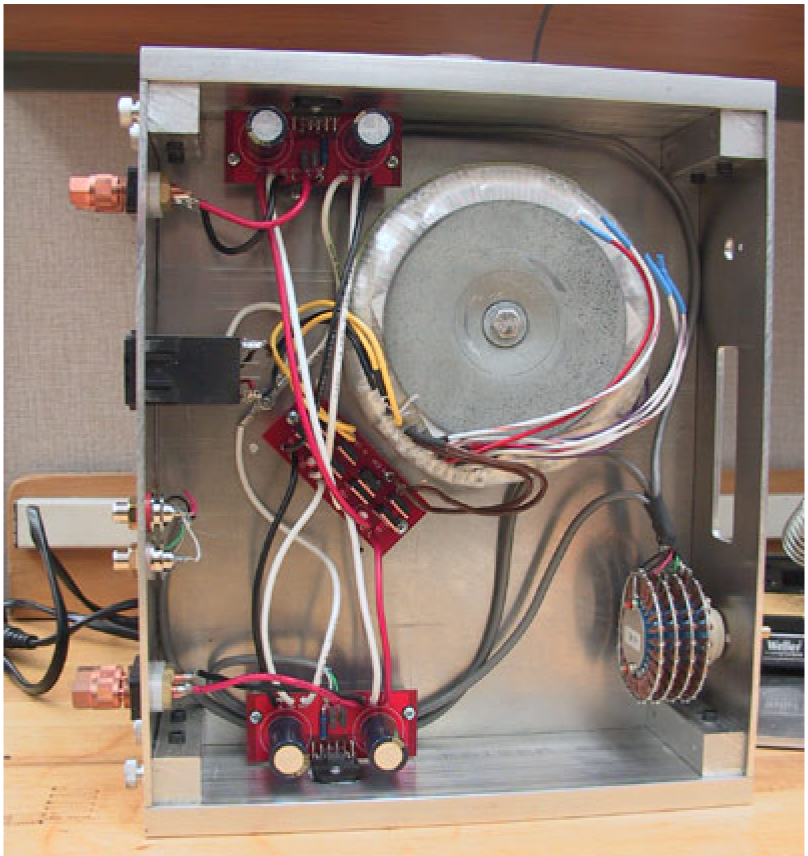


5. Now it is time to mount the boards to the chassis. For the standoffs on the PCB, I used regular 1” standoffs with #4 screws. Heatsink compound is required for mounting the LM3875 to the chassis.

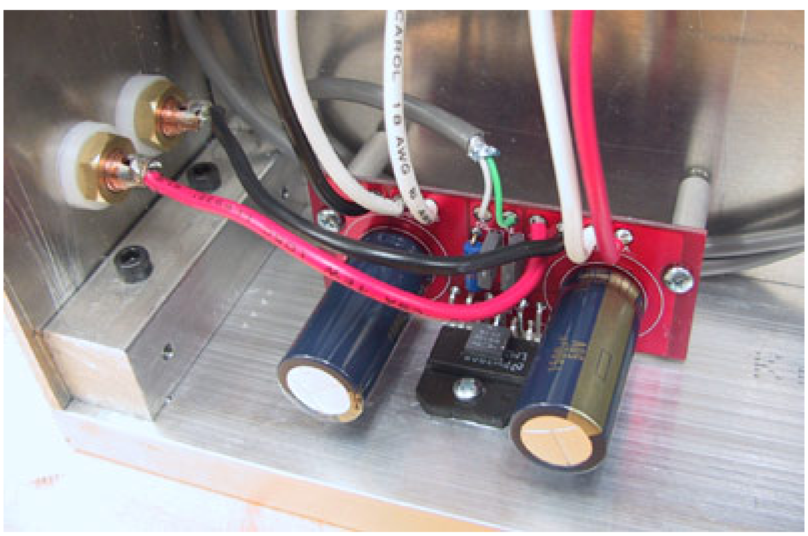


If you are using the LM3875T IC, make sure to use the provided Aluminum Oxide insulator and the screw flange. #4-40 screws is recommended for mounting the boards.

6. Here is a picture of my chassis with the boards mounted:



And a close-up of the amplifier PCB:



Assuming that you took care of tying the input and output to the board, it should now be a functional amplifier.

3.3 Chassis Considerations

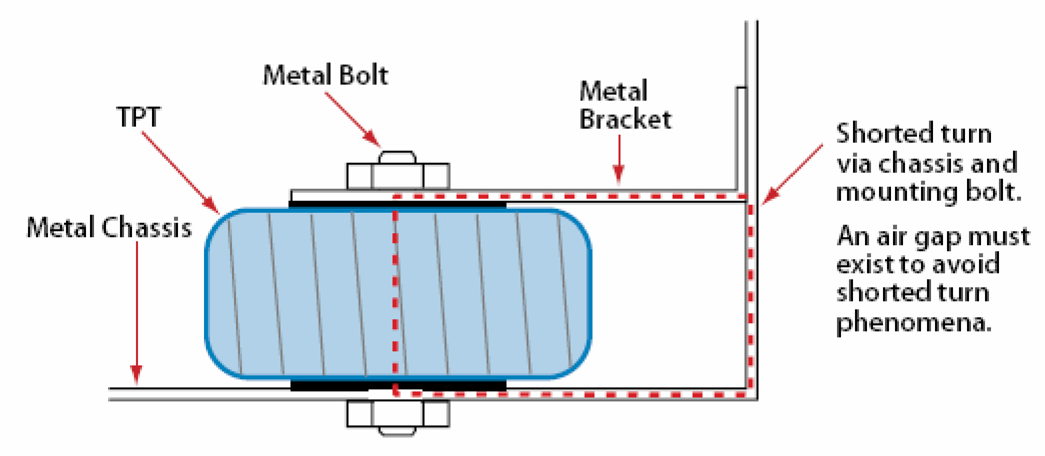
There are many options on chassis design. Much of this us up to the builder's taste, tools, budget and time. One fundamental issue which is the same, regardless of final design, is safety. The chassis keeps dangerous voltages from harming anyone or anything that is in the area of the amplifier.

The chassis must be properly grounded to the mains ground to prevent possibly lethal voltages from being seen on the chassis in the event of failure. Ground loops, which commonly are a source of hum in situations may still be alleviated by proper chassis layout, shielding and the use of a star ground. Stopping the source of hum is the only option, instead of disconnecting the chassis ground. At times, when connected to a complex system, hum can be caused by interfacing to coax cable used for video. Try disconnecting this cable from the system entirely and if this solves the problem, consider an isolation transformer for cable systems. Chassis layout also can affect hum. Keeping signal level wiring away from the transformer is a good place to start.

Just about anything which can safely isolate the user from the amplifier could be a viable chassis. Some great and creative examples can be found on www.briangt.com as well as other sites on the Internet. Some designs focus on aesthetics, while others offer simplicity and others focus on optimising various materials to provide a stable platform for the amp to subjectively sound its best.

The last consideration for chassis construction is heatsinking. This amplifier does not require huge heatsinks. Successful implementations have used computer heatsinks without fans, solid pieces of bar stock or aluminum angle and others rely on standard large heatsinks. An aluminum plate around 3" x 3" x ½" should be plenty for most applications if allowed to circulate in free air.

When attaching a torrodal transformer to the chassis, care must be taken to not create a shorted turn. This occurs when a ferrous (metal) object creates a loop through the center of the transformer. In the following example, eliminating the top metal bracket and using only the washer supplied with the transformer would eliminate the shorted turn.

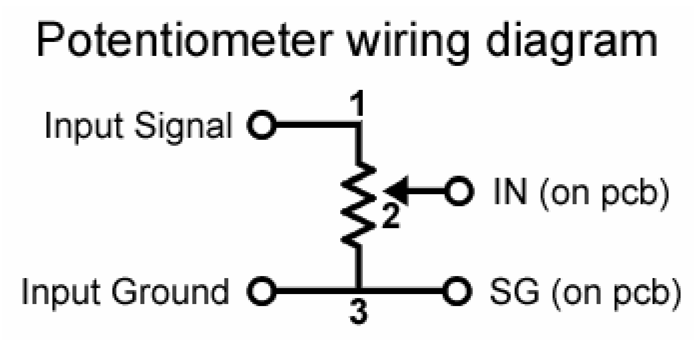


3.4 Volume Control

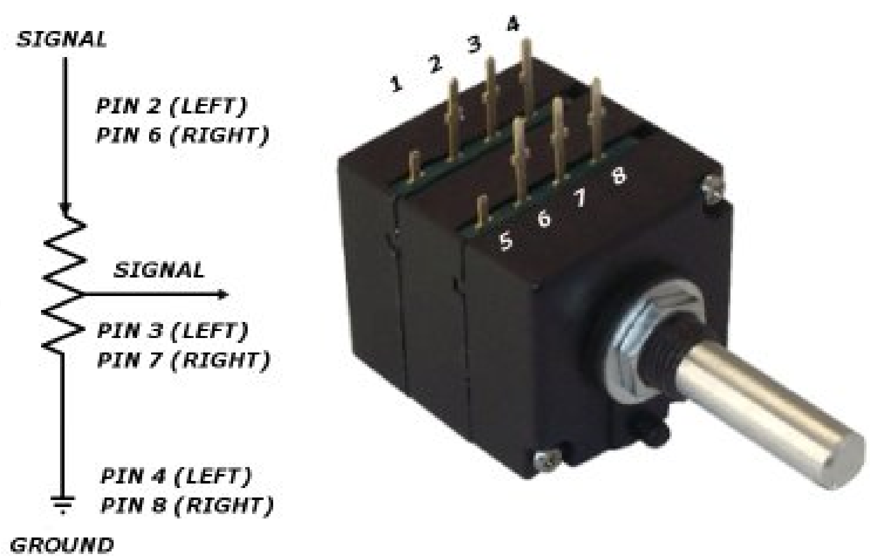
Volume control can be achieved by use of a receiver or pre-amp or via signal attenuation. An attenuator can be made using a potentiometer with a shunt resistor or a multi-position switch with various combinations of resistors to attenuate the signal to appropriate levels. If a switch is used, a make before break switch should be used to to eliminate the possibility of popping when switching from one setting to another.

If a receiver or pre-amp is used and has sufficient gain, the amplifier could be overpowered and clipping could occur. If the volume control on the pre-amp gets extremely loud with small variation, lowering the gain of the amplifier is in order. Lower the gain of the amplifier by varying R3 per the equation Gain = 1+ Rf/R3. The standard supplied gain set by R3 is 33dB. Gain should remain above 10dB to reduce the chance of the amplifier going into oscillation. To lower the gain, increase the value of R3.

The following diagram shows the wiring scheme for a typical three-leg potentiometer:



Here is an example of a Noble stereo potentiometer:



Pins 1 & 5 are cut to prevent accidental short

4 Miscellaneous Information

4.1 The Zobel Network

A brief description of the implementation of a Zobel network and why this would be done follows. Please note that this is per the recommendation of some experienced designers and is included in the National datasheet. As of this time, I have not tried the implementation of a Zobel with this amp and have suffered no apparent ill effects. I have included this option, as it may prove useful to some users.

What does the Zobel do? It helps prevent high frequency oscillation, which may occur with difficult loads/capacitive cables. Basically, it creates a low pass filter, with the 2.7uF cap and 1 ohm resistor creating a pole at 1/(2*pi*r*c), which can solve issues with cables, as well as reducing RF interference. According to some people, the Zobel can have a negative effect on the sonic performance. I haven't tried it myself to see. Builders who are concerned about the presence of oscillations induced by cables should indeed experiment with this addition and are encouraged to post their results!

4.2 The Bridged LM3875

The following is a brief explanation of options for bridging the LM3875 chip. There are many different options, including buffering, which would result in a different topology than the one alluded to below.

The first thing you need to determine is the load you want to drive. Keep in mind the effect of bridging and paralleling of amplifiers:

- When you bridge an amplifier, this means that you will invert the signal phase and send it into one channel, while sending the normal phase into the other channel. This effectively creates an input signal of twice the amplitude. With the two channels bridged, each channel sees 1/2 the load impedance, meaning that if the speaker was 8 ohms, each amp would see 4 ohms. Since this is true, you need to make sure that you are not driving too low of a load with a bridged set-up. Note that the normal phase amp is driving the + input for the speaker, while the inverted amp is driving the - input to the speaker.

- When you parallel 2 channels, you are sending the same signal into both channels, causing each amplifier to see twice the load, meaning that a 4 ohm speaker would cause each amp to see an 8-ohm load. Note that the outputs of the paralleled amplifier are tied together, and each should be in series with a 0.22ohm resistor.

Now, to figure out the power, simply figure out how much load each amplifier is driving. If you are driving 8-ohm speakers and bridging, the power will more than double. If you are driving a 4-ohm speaker with a paralleled set-up, the power will increase, but not quite double.

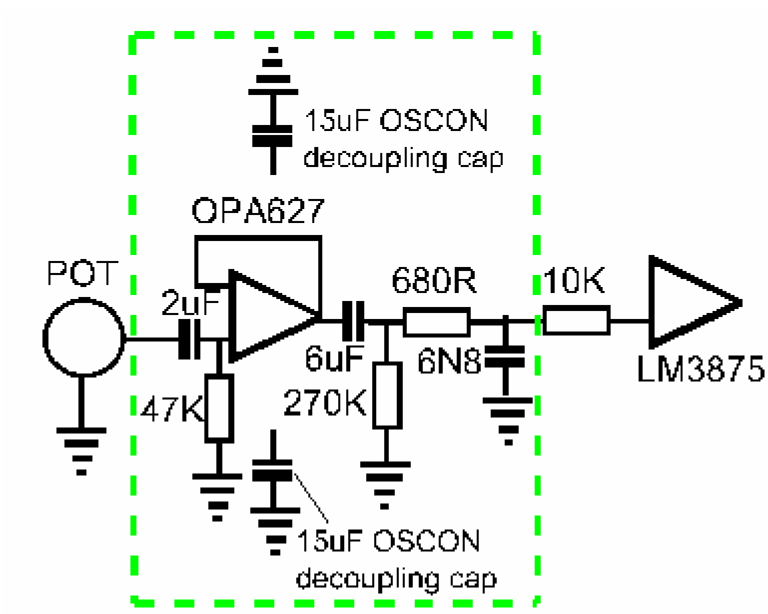
The ultimate configuration for power would be a bridged-parallel configuration with 4 amplifiers per channel.

Paralleling is easy, with the addition of the 0.22 ohm resistors in series with the output. Bridging will require a circuit that will invert the phase of one channel, such as a simple circuit using the DRV134.

Feel free to search for more on your favourite website for more background on bridging this amp or look into the LM4870 chip discussions.

4.3 An Input Buffer

Nick Whetstone at Decibel Dungeon has compiled some information from other sources regarding the addition of an input buffer. It is a good place to start to determine if this addition is something you would like to try. The following diagram is posted with Nick's permission and is one way to achieve a simple buffer for the circuit.



If this addition interests you, be sure to check out Nick's website http://www.decdun.fsnet.co.uk , as well as those of his references and the typical DIY sites for additional information.

5 Frequently Asked Questions (FAQ)

Here is a list that I compiled of frequently asked questions:

5.1 What started this group order?

I was teaching my friend how to use some PCB layout software for his projects, and I gave him the assignment to make a PCB for the gainclone, as it is the simplest circuit that I know of, yet still functional. I had been playing around with the gainclone for the last year, trying different versions, but never actually taking it seriously. The circuit was based off of Peter Daniel's simplified version that he posted on the forum. My friend returned to me a board layout that was twice the size of the current board, and had long unholy traces running everywhere. I decided that I would make my own layout and show him what I thought would be the best implementation. I spent a night working on it, and posted the result the next day to DIYaudio to see if others had comments on it. I received several e-mails requesting boards, and a recommendation to offer a kit, as I had chosen a specific set of components for the design.

Anyway, I worked on the design more, and started a new thread a couple of days later to see how much interest a group order of pcbs for this would generate. There was quite a substantial interest, so I created a rectifier PCB also, and decided to put 2 channels + rectifier on a single PCB. I worked with Peter Daniel to refine my PCB layout, and get a design that would be work best, for chassis mounting, and optimal signal paths. I researched components and availability and put up an order page.

From here, I generated orders for well over 100 kits.

5.2 Where did you first find out about the gainclone?

There was a discussion on DIYaudio over a year ago about the term Frugal-phile™, which Dave (planet10) started by saying:

"It is relatively easy to spend big-bucks and get a reasonable sounding hifi (but not as easy as you might think, given the number of mega-buck audio-pile systems that are not very listenable). The goal here is to spend as little as possible and get a hifi that is musically enjoyable."

From this, I got curious about what kind of amplifiers would be considered Frugal-phile™. I asked about it, and the term GainKlone came up. I found a reference to the Chip Amp Forum. From here, I saw Thorsten's design, and decided to try it. I made one, and never took it seriously, until I saw Peter Daniel's latest amplifiers. At this point, I was hooked on using the gainclone.

5.3 I just ordered the PCB set, what premium components should I use with it?

If you want to use premium components, like I used with the kits, here are a list of vendors that carry these parts:

Riken resistors: Michael Percy: www.percyaudio.com, Angela: www.angela.com
Caddock resistors: Michael Percy: www.percyaudio.com,
pcX: www.partsconnexion.com
Blackgate caps: Michael Percy: www.percyaudio.com,
pcX: www.partsconnexion.com,
Reference Audio Mods: www.referenceaudiomods.com
Rest of components: Digikey: www.digikey.com

The parts list for the boards is listed above in section 2.1.

5.4 I would rather start with basic components, what works with the board?

If you just want to use standard components, the board will work fine, and you can always upgrade later on. Here is a parts list for Digikey:

#	Part Name/Description	Digikey Part Number
2	LM3875TF	LM3875TF-ND
4	1500uF 50v Panasonic FC Capacitor	P10334-ND
4	22.1k ohm resistor (min quantity = 5)	PPC22.1KXCT-ND
2	680 ohm resistor (min quantity = 10)	P680BBCT-ND
2	220 ohm resistor (min quantity = 5)	PPC20.0KXCT-ND
8	On Semiconductor MUR860 fast diodes	MUR860OS-ND
2	4.7uF 50v Panasonic FC Capacitor	P10315-ND

If you are just buying these components for yourself from Digikey, in small quantities, expect to pay about $35 for all these parts, plus shipping from Digikey. The budget kit will cost around $35, including all of the parts above and a PCB set.

5.5 I have a suggestion for material to be added to this manual

Feel free to drop an e-mail to: manual@chipamp.com and I will try to look over this account once and a while to make revisions to this manual.

Thanks to Sandy for all of his help in preparing this manual.