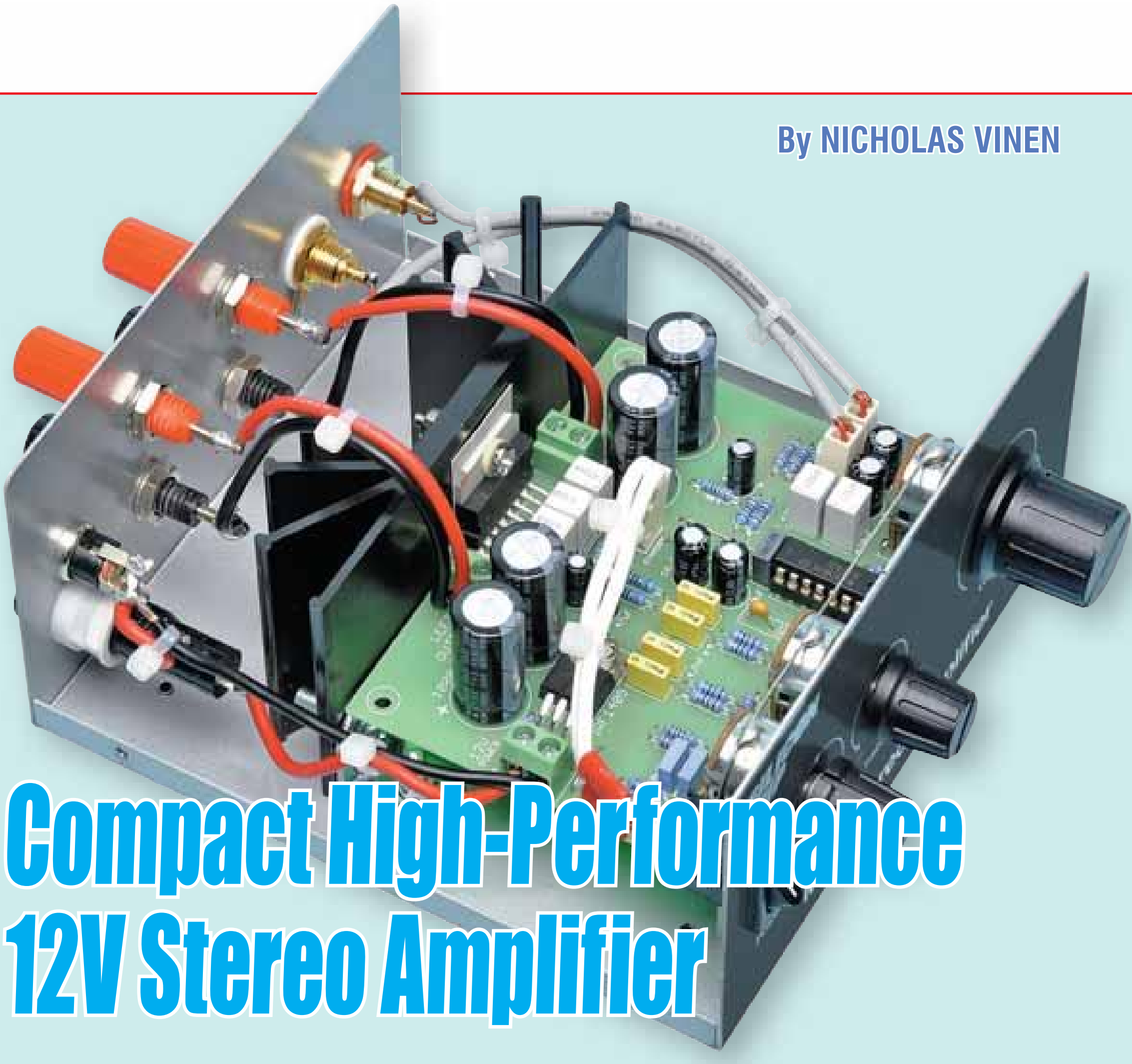
By NICHOLAS VINEN



Compact High-Performance 12V Stereo Amplifier

Amplifiers that run from 12V DC generally don't put out much power, and they are usually not 'hifi' either. But this little stereo amplifier ticks the power *and* low distortion boxes. With a 14.4V supply, it will deliver 20W per channel into 4Ω loads at clipping, while harmonic distortion at lower power levels is typically less than 0.03%.

THIS IS an ideal project for anyone wanting a compact stereo amplifier that can run from a 12V battery. It could be just the ticket for buskers who want a small but gutsy amplifier, which will run from an SLA (sealed lead-acid) battery. Alternatively, it could be used anywhere that 12V DC is available – cars, recreational vehicles, or remote houses with 12V DC power.

Since it runs from DC, it will be an ideal beginner's or school project, with no 230V AC power supply to worry about. You can run it from a 12V battery or a DC plugpack. But while it may be compact and simple to build, there is no need to apologise for 'just average' performance. In listening tests from a range of compact discs, we were very impressed with the sound quality.

The bottom line – this amp has a THD under typical conditions of just 0.03% or less (see Performance figures on page 63). Also, its idle power consumption is low – not much more than 1W. Therefore, if you don't push it too hard, it will run cool and won't drain the battery quickly. And because the IC has self-protection circuitry, it's just about indestructible. It will self-limit or shut down if it overheats, and the outputs are deactivated if they are shorted.

Obtaining enough power

With a 12V supply, the largest voltage swing a conventional solid-state power amplifier can generate is ±6V. This results in a meagre 4.5W RMS into 4Ω, and 2.25W RMS into 8Ω, without even considering losses in the output transistors. Even if the DC supply is around 14.4V (the maximum that can normally be expected from a 12V car battery), that only brings the power figures up to 6.48W and 3.24W for 4Ω and 8Ω loads respectively – still not really enough.

There are three common solutions to this problem. The first is to boost the supply voltage using a switchmode DC converter. This greatly increases the cost and complexity of the amplifier, but it is one way of getting a lot of power from a 12V supply. However, we wanted to keep this project simple and that rules out this technique.

The second method is to lower the speaker impedance. Some car speakers have an impedance as low as 2Ω, which allows twice as much power to be delivered at the same supply voltage. However, we don't want to restrict this amplifier to 2Ω loudspeakers.

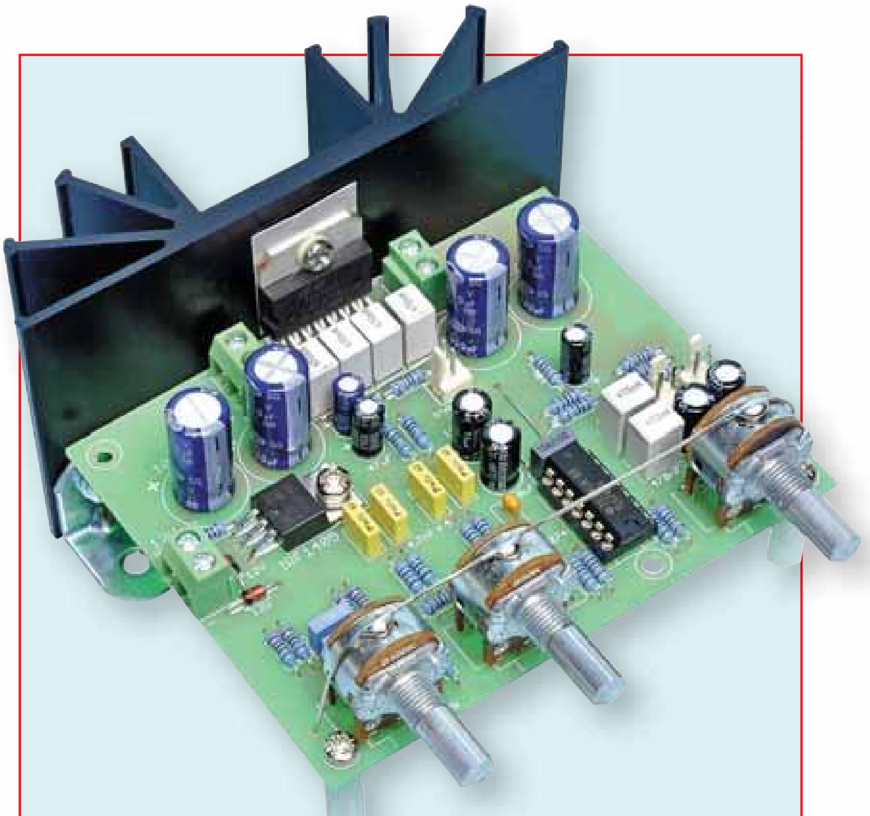
Bridge mode

The remaining solution is to use bridge mode, also known as bridge-tied load (BTL). It requires two amplifier circuits per channel. The TDA-7377 IC is ideal for this purpose – it contains four amplifiers in a single package and is intended for a bridged stereo configuration.

In the TDA7377, two of the four amplifier circuits have inverting inputs, so all we need to do is to feed the same signal to one of each type and the outputs will swing in opposite directions – when one voltage goes up, the other will go down, and *vice versa*.

Instead of connecting the speakers between the amplifier output and ground, we connect them between the two outputs. This doubles the voltage across the speaker and multiplies the maximum power delivered by four ($P = V^2/R$). It also eliminates the AC-coupling capacitor at the output, which is needed with a standard single supply amplifier.

Practically speaking, virtually any 4Ω or 8Ω speaker is suitable for use with this amplifier; the more efficient, the better. Avoid anything less than 4Ω, as that would be asking each amplifier circuit to drive a load under 2Ω, which the IC is not rated for.



It may be small, but the High-Performance 12V Stereo Amplifier puts out up to 20W per channel into 4Ω loads at low distortion. It uses just two ICs and is very easy to assemble.

Circuit description

The full circuit diagram is shown in Fig.3. As can be seen, it's based on the aforementioned TDA7377V monolithic stereo BTL amplifier (IC2) plus a TL074 quad FET-input op amp package (IC1). The latter provides the tone control stages in both channels.

As shown, the input signals are fed via 4.7µF non-polarised (NP) capacitors to a 10kΩ dual-gang potentiometer (VR1), which serves as the volume control. From there, the signals are AC-coupled via 470nF capacitors to op amps IC1a and IC1b. These act as unity-gain buffer stages to provide a low source impedance for the following Baxandall tone control stages based on IC1c and IC1d.

In operation, IC1c and IC1d, and their associated potentiometers (VR2 and VR3), provide bass and treble boost of ±15dB, with a centre frequency of 700Hz. The frequency response is very flat when the pots are centred (see Fig.11).

To understand how the tone control stages work, let's consider the bass and treble sections separately. We'll concentrate on the bass sections first, but will initially ignore the 10nF capacitors. This leaves us with an inverting amplifier (IC1c or IC1d), where the resistors (including the pots) form the feedback network and thus control the gain. With the bass pot (VR2) turned all the way clockwise, the gain is set at 122kΩ/22kΩ, or about 5.5. If it is turned in the opposite direction, the gain is 22kΩ/122kΩ, or 0.18.

Adding the 10nF capacitors across VR2a and VR2b adds a low-pass filter to each gain network, so that turning the knob affects low frequencies more than high frequencies. As a result, we can adjust the gain of the bass and hence achieve bass boost/cut.

The treble section (VR3a and VR3b) works similarly, except that the capacitors (4.7nF in this case) are in series with the resistors, thus forming a high-pass filter instead.

The 10pF capacitors on the inverting inputs of IC1c and IC1d reduce their gain at high frequencies, thereby

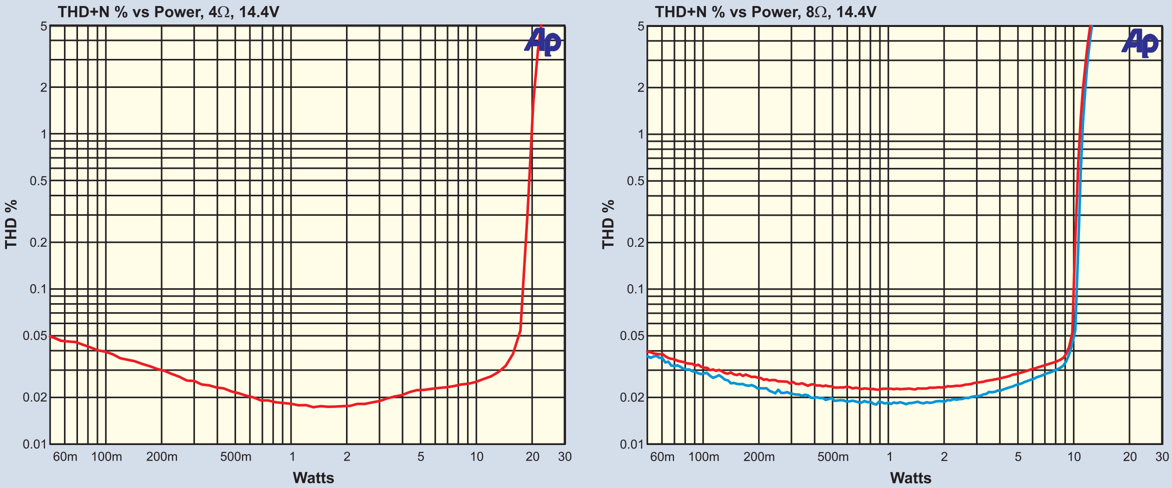


Fig.1: THD+N vs output power at 4Ω (one channel driven only). The supply is 14.4V and the measurement bandwidth is 20Hz to 22kHz. The distortion increase below 1W is due to noise.

Fig.2: THD+N vs output power at 8Ω with both channels driven. The supply is 14.4V and the measurement bandwidth is 20Hz to 22kHz. The disparity between the channels is primarily due to tone control pot tracking errors.

preventing oscillation in case there is RF pick-up in the filter network. Similarly, the 10Ω resistors at the outputs of IC1c and IC1d attenuate any RF signals which may make it through before they go into the power amplifier (IC2).

Power amplifier

Only a few external components are required by the single TDA7377V quad power amplifier IC (IC2). It's very clever – not only does it contain the four low-distortion amplifiers we need to drive stereo speakers in BTL configuration, but it has virtually rail-to-rail swing on the outputs and is inherently stable with a fixed 26dB gain.

We have used its standby pin (pin 7) to switch the amplifier on and off. This avoids having high current passing through on/off switch S1. In fact, S1 only switches the power to pin 7 of IC2 and to the quad op amp IC1. Hence, the power supply and IC1 remain energised as long as the supply voltage is present, but only the capacitor leakage and standby current are drawn, a total of around 100µA. Switching the amplifier on raises the quiescent (no signal) current to around 100mA.

As soon as switch S1 is turned on, the 100µF filter capacitor is charged via diode D1. The standby pin (pin 7) has a low-pass filter consisting of a 22kΩ resistor and 1µF capacitor, so that the power amplifier is not enabled until the op amp is on. This avoids turn-on thumps.

Similarly, when you switch S1 off, the 22kΩ resistor at the anode of diode D1 pulls down the standby pin voltage, turning the power amplifier IC off almost immediately. This avoids switch-off thumps from the loudspeakers.

Reverse polarity protection

The main power supply components are the four 2200µF 25V electrolytic capacitors, plus two 470nF MKT capacitors in parallel for high-frequency filtering. MOSFET Q1 provides reverse polarity protection for this section.

Although the TDA7377 IC can withstand negative supply voltages, the electrolytic capacitors cannot. So, in this circuit we have connected an IRF1405 MOSFET in series with the supply ground lead.

In essence, the MOSFET acts like a diode with a very low forward voltage, typically less than 25mV at 5A (we measured 8.7mV at 2.5A). This compares with around 1V at 5A for a standard rectifier diode. This means that the amplifier can deliver significantly more power, about 15% more, in fact, than if a standard diode had been used.

Power source

The maximum current consumption depends on the speaker impedance and how far you turn up the volume. As a rough guide, full power with a 14.4V supply and 8Ω speakers requires at least 3A. For 4Ω speakers, the current consumption can exceed 6A.

At a minimum, use a 7.2Ah SLA battery for 8Ω speakers, or a 12Ah SLA for 4Ω. These should last 2 to 24 hours, depending on how hard you're driving the amplifier (larger batteries will last longer).

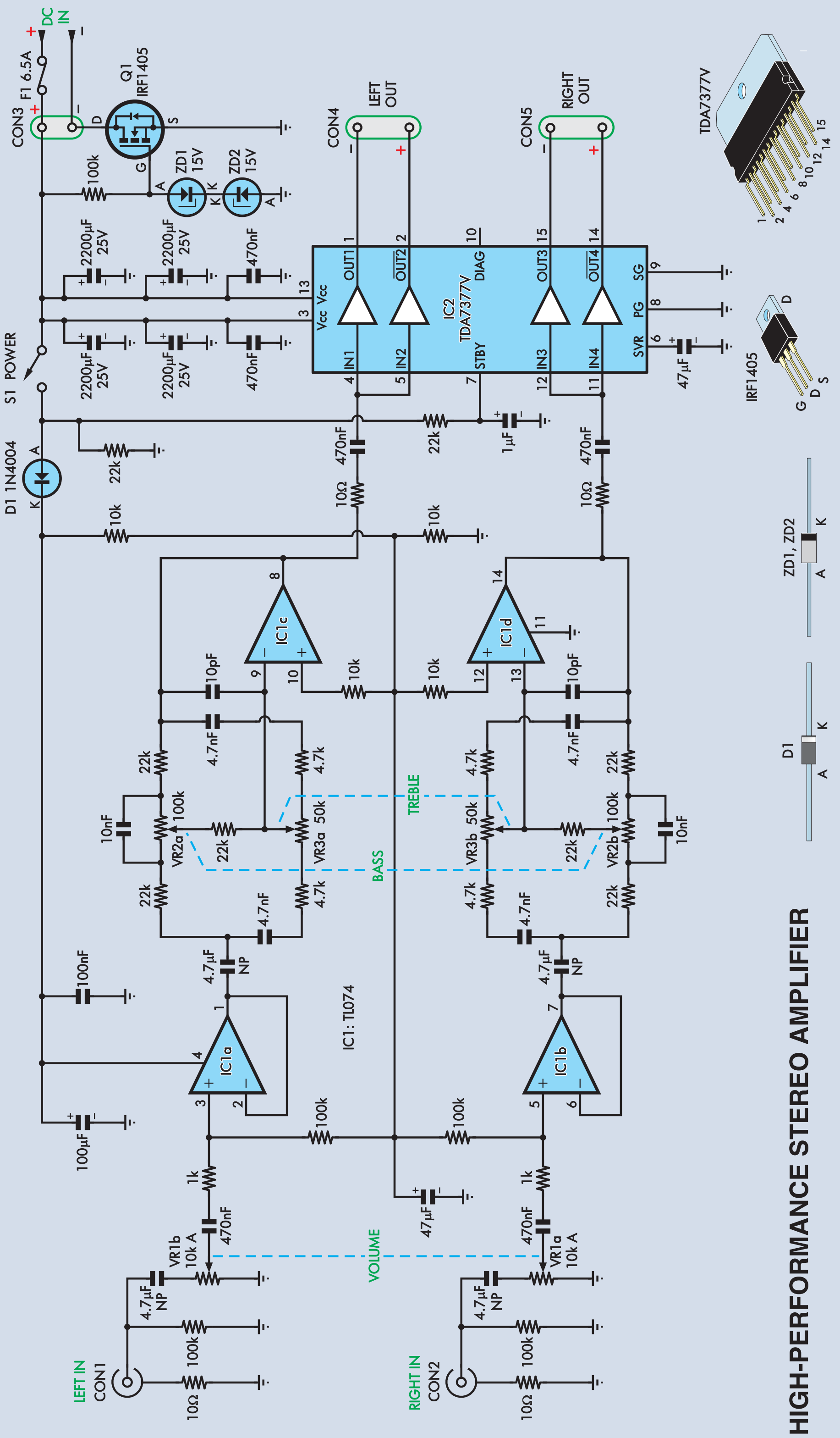
You can charge the battery while using the amplifier, although this may slightly prejudice the sound quality due to the supply ripple that charging introduces. Power supply rejection is >50dB at 300Hz, and thanks to the large supply bypass capacitors, the additional noise should be kept to a low level.

If you want to run the amplifier from a mains power supply, both linear and switchmode types are suitable. A 6A linear supply is likely to be large and expensive, so switchmode is probably the way to go. A higher supply voltage (ie, up to 16V) will give more power.

The absolute maximum operating voltage is 18V, so make sure whatever you use can never exceed that.

Construction

All the circuit components, including the potentiometers for the volume and



HIGH-PERFORMANCE STEREO AMPLIFIER

Fig.3: the complete circuit is based on a TL074 quad FET-input op amp (IC1) and a TDA7377V quad power amplifier (IC2). IC1a-IC1d and their associated stereo potentiometers (VR2-VR3) form a Baxandall tone control circuit, and this drives IC2, which is wired in bridged stereo mode. MOSFET Q1 provides reverse polarity protection.

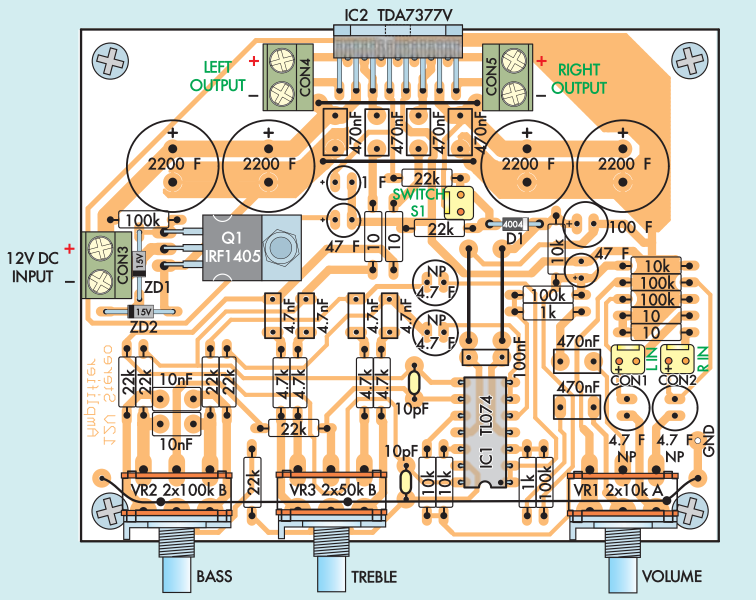


Fig.4: follow this layout diagram to build the PC board. Make sure that all polarised parts are correctly oriented and don't get the pots mixed up.

This view shows the completed prototype. The potentiometer bodies are connected together using a length of 'tinned' copper wire, which loops across them and is terminated in pads on either side of the board.

tone controls, are mounted on a single PC board measuring 97mm × 78mm and coded 843. This board is available from the *EPE PCB Service*. The PC board is housed in a compact metal case with an aluminium base and steel lid.

Fig.4 shows the component layout on the PC board. Start by checking the board for defects, such as shorts or breaks in the copper tracks and undrilled holes. If it's OK, start the assembly by installing the four wire links using 0.71mm tinned copper wire. Make sure they are straight and flat before soldering, since some of the links pass near exposed component legs.

Next, install the fixed value resistors. It's a good idea to check each value with a DMM, as the colour codes can be notoriously hard to read.

After that, solder in the two Zener diodes. They are identical, but do make sure that you get their orientation correct.

Once the Zeners are in, bend the MOSFET's leads down at right angles, about 5mm from its metal tab, using small long-nosed pliers. That done, insert its leads into the PC board and check that its mounting hole lines up. Adjust the leads if necessary, then secure the tab to the PC board using an M3 × 6mm machine screw, spring washer and nut. Once it is firmly in place and cannot move, solder and trim the three leads.

Next, install the three terminal blocks. Push them all the way down so that they sit flush with the board and check that they are correctly oriented before soldering their pins. The three polarised headers (CON1, CON2 and S1) can then be installed, again taking care with their orientation.

Follow with the MKT capacitors and the two ceramic types. The polarity doesn't matter here, but don't get the values mixed up. The four bipolar electrolytic capacitors can now be installed, followed by the four small polarised electrolytics. Don't install the larger 2200μF units just yet.

Next, install the TL074 IC, making sure it goes in the correct way around. We used a socket in our prototype, but there's no reason why it cannot be directly soldered to the PC board.

Installing the TDA7377

It's now time to install the TDA7377V. You must do this slowly and carefully since it's difficult to remove if it's misaligned.

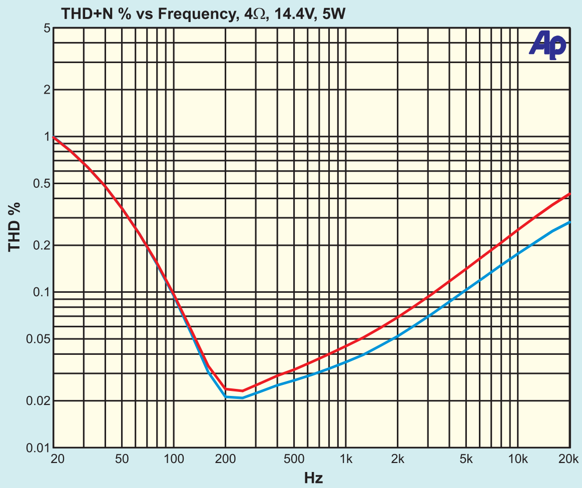


Fig.5: THD+N vs frequency for 5W into 4Ω. The supply is 14.4V and the measurement bandwidth is <10Hz-80kHz. The reading at 1kHz is slightly higher than in Fig.1 due to the wider measurement bandwidth.

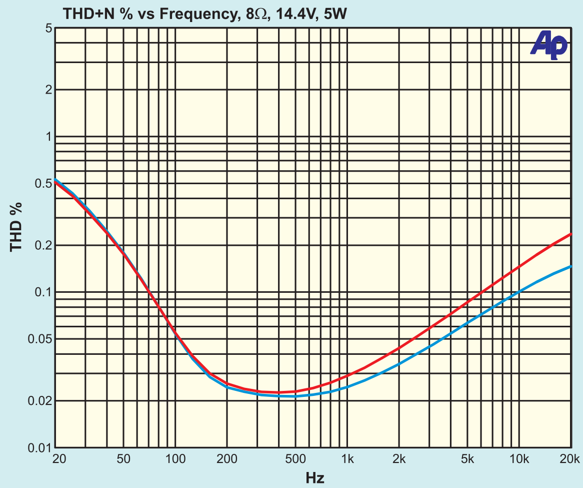


Fig.6: THD+N vs frequency for 5W into 8Ω. The supply is 14.4V and the measurement bandwidth is <10Hz-80kHz. The reading at 1kHz is slightly higher than in Fig.2 due to the wider measurement bandwidth.

Start by gently inserting its pins through the board, taking care not to bend any of them. You may need to adjust them using needle-nose pliers if they have been bent during transport, so that they line up properly.

Once the pins have been pushed all the way down, place the metal tab of the IC on a flat, horizontal surface with the PC board held in a vertical position. Hold the IC down against the surface and adjust the angle of the board so that they are exactly at right angles. That done, check that the pins are all still properly inserted and that the package is parallel with the edge of the board.

When it is all straight, solder a couple of leads and recheck its orientation before finishing the job.

The four 2200µF electrolytic capacitors can now be installed. Make sure that each of these sits flush against the PC board, and is oriented correctly.

Short cut

Before fitting the potentiometers (VR1 to VR3), it's necessary to cut their shafts to length so that the 'D'-shaped sections (ie, the flat sections) are about 10mm long. This is done by clamping the end of each shaft in a vice and then cutting it with a hacksaw. Deburr the ends when you have finished, so that the knobs can be fitted later.

Pots VR1 to VR3 can now be installed on the board. They each have a different value, so be sure to mount each one in the correct location.

Earthing

To prevent noise pickup, it's necessary to 'earth' the bodies of the pots. This is done using a length of tinned copper wire, which loops across the top of the pots and is terminated at both ends to copper pads on the PC board.

To install this wire, first solder one end to the pad immediately to the right of volume control VR1. That done, stretch the wire across the tops of the three pots and feed the free end into the pad to the left of the bass pot. Finally, pull the wire down tight and solder it in position, then solder the wire to the top of each pot body.

Note that it will be necessary to scrape away the passivation material on each pot body in order for the solder to take. You will also need to use a hotter-than-normal soldering iron in order to heat the pot bodies sufficiently for soldering.

In practice, the pot bodies will later all be in contact with the bare metal of the case, so it should not be necessary to connect the GND pad on the PC board to the case itself. **However, if you elect to house the board in a plastic case, it will be necessary to connect the GND pad to the ground (–) terminal of CON3.**

The PC board assembly can now be completed by attaching an M3 ×12mm

Table 1: Resistor Colour Codes

❏	No.	Value	4-Band Code (1%)	5-Band Code (1%)
❏	5	100kΩ	brown black yellow brown	brown black black orange brown
❏	8	22kΩ	red red orange brown	red red black red brown
❏	4	10kΩ	brown black orange brown	brown black black red brown
❏	4	4.7kΩ	yellow violet red brown	yellow violet black brown brown
❏	2	1kΩ	brown black red brown	brown black black brown brown
❏	4	10Ω	brown black black brown	brown black black gold brown

Follow this photo and the text to complete the wiring inside the case. Note the use of shielded cable to connect the RCA-type phono input sockets.

spacer to each corner, secured by M3 × 15mm machine screws. That done, thread M3 × 9mm spacers over each end so that you effectively finish up with four 21mm spacers. Note: the spacers at the rear of the board are later removed when it is attached to the heatsink.

Initial checks

If you have a bench supply, set it to 12V with a current limit of 200mA. Otherwise, use a 12V plugpack or similar supply – if possible, one which is too small to provide much current.

First, connect your DC supply to the power block (CON3), with a DMM wired in series and set to read current. Now switch the supply on and check the current reading. With no power switch attached, the current should be negligible (<1mA) and the DMM will probably read 0 (once the 2200μF capacitors have charged).

If it reads more than a few milliamps, switch off and check the board for mistakes.

Now short the switch header pins (ie, for S1) together using a piece of wire. The current should now increase to around 100mA and possibly as high as 160mA. If you remove this short, the reading should drop back to 0mA within a second or so.

If you have made a mistake with the MOSFET or Zeners, it's possible no current will flow at all. If that happens, check that area of the board. It could also be a problem with the standby RC filter components (22kΩ and 1μF).

Drilling the case

The drilling details for the front and rear panels of the case are shown in Fig.7. This can be photocopied and used as a template.

Start by attaching the front panel drilling template section to the case. Once that's done, centre-punch the location of each hole, then remove the template and drill a small pilot (eg, 2mm) hole at each location. Before going further, place the board assembly in the case and check that the three right-hand holes line up correctly with the centres of the pot shafts.

Once you are satisfied that everything is correct, drill the three potentiometer holes to 6.5mm, then check that their shafts (including the threaded portions) go all the way through). If necessary, enlarge the holes using a tapered reamer until it fits correctly.

The switch cutout is made by first pilot-drilling the two marked positions, then enlarging them to 5.5mm and drilling a third hole between them. The centre piece can then be knocked out and a small flat file used to gradually enlarge the cutout to the marked rectangular outline.

Slowly enlarge it in each direction until the switch snaps into place and is locked in by its plastic tabs. It will take a good 10 to 15 minutes of patient filing, so take it slowly and make sure you don't make the hole too large or crooked.

Installing the PC board

In order to later secure the PC board, it's necessary to drill mounting holes in the base of the case for the front (but not the rear) spacers. The two holes should each be marked on the underside of the case, and are positioned 6mm in from the front panel and 21mm in from the sides. Drill them to 3mm, then slide the board into position and fit the mounting screws.

Don't worry about securing the pots to the front panel at this stage – that step comes later. For the time being, just check that everything lines up correctly, then remove the board and power switch so that the rear panel can be drilled.

Rear panel drilling

Eight holes have to be drilled in the rear panel – two for the insulated RCA-type phono input sockets (D), four for the loudspeaker terminals (C), one for the DC power socket (C) and one for the fuse (E).

Begin by securing the drilling template accurately in position (use tape), then centre-punch each hole location, and drill small pilot holes. The template can then be removed and the

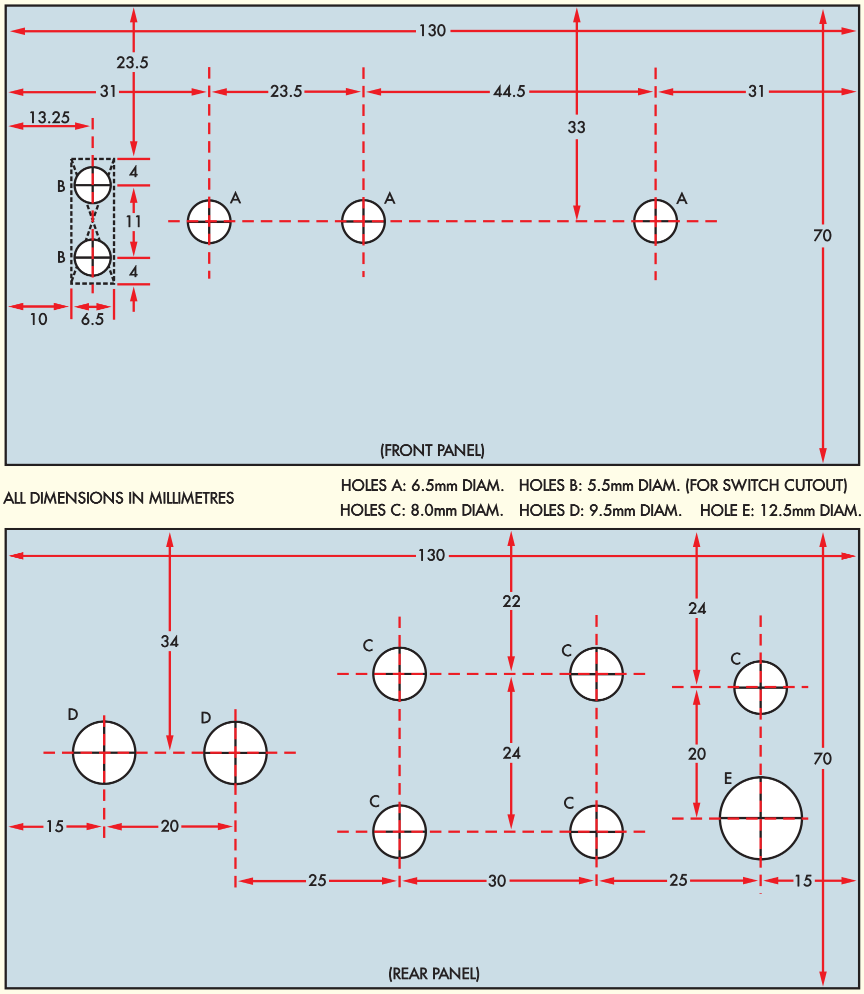


Fig.7: full-size drilling templates for the front and rear panels. Centre-punch each hole position and drill small pilot holes before carefully enlarging them to size using progressively larger drills and a tapered reamer.

holes enlarged to the sizes indicated using drills and a tapered reamer (ie, 8mm for the binding posts and DC connector, 9.5mm for the phono sockets and 13 to 15mm for the fuseholder, depending on the exact type).

Use an oversize drill to deburr the holes, then install the rear panel hardware as shown in the photos. Make sure all the nuts are tight so the components can't rotate. By the way, *insulated* RCA-type phono sockets are mandatory if you want to get low distortion.

Pay attention to the orientation of the holes in the binding posts. The upper two (red) should have the holes vertical, while the lower two (black) should be orientated with the holes 30° to 45° from vertical, so that you can insert the speaker leads from the side.

Attaching the heatsink

The specified heatsink is a 55mm × 105mm 'fan' type. It is quite heavy, so it will need to be attached to the base of the case using two right-angle steel brackets (see the photo two pages on – homemade *matching* aluminium or sheet steel will work perfectly well).

To do this, stand the heatsink vertically on a flat surface and place a bracket flush against the flat side at one end. Mark the centre of the mounting hole, then repeat this procedure at the other end. The two holes are then centre-punched and drilled to 4mm. Remove any swarf from around the holes using an oversize drill, then attach the brackets using two M4 × 10mm machine screws, spring washers and nuts.

Now remove the rear spacers from the PC board, install it in the case and slide the heatsink up to it so that it sits flush against IC2's metal tab. Check that the heatsink is correctly centred, then mark the mounting holes for the heatsink brackets on the bottom of the case. The heatsink and PC board are then removed and the marked locations drilled to 4mm.

The next step is to drill a mounting hole in the heatsink for IC2's metal tab. That's done by first reinstalling the PC board in the case and securing

Using a MOSFET as a diode

In this project we have used a MOSFET instead of a diode for reverse polarity protection, for the reasons explained in the article.

Fig.8 shows how an N-channel MOSFET is typically used for motor control, lamp flashing or any other task where a high current DC electronic switch is required. Because a MOSFET's source is generally connected to the substrate, a parasitic diode known as the 'body diode' is present. This is shown in the symbol and it cannot be avoided.

Because its source (S) is connected to ground, Q1 is on whenever the gate (G) voltage is above the MOSFET's on-threshold (usually 2V to 4V). The body diode is reverse biased and does not conduct unless the load is inductive and switch-off causes a large enough positive voltage spike to trigger reverse breakdown (avalanche).

What we want to do, though, is use a MOSFET to prevent current flow if V_{CC} becomes negative. In the case of Fig.8, if this were to happen, the body diode would conduct and it would be impossible for the MOSFET to provide reverse polarity protection. Hence, we must reverse the MOSFET and connect it so that the source is positive with respect to the drain (D), as shown in Fig.9.

Note that the body diode is now forward biased when V_{CC} is positive. If we also bias the MOSFET on, all of the current will flow through the channel (ie, source to drain) instead. The channel path will have a much lower voltage drop than the body diode.

We achieve this by connecting a resistor between the supply input and the gate. When the supply voltage is positive, the MOSFET is turned on and if it becomes negative it will be turned off and of course, the body diode will be reverse-biased!

Because the source is no longer connected to ground it may seem that we can no longer turn the MOSFET on. In fact, the source is pulled to ground via the body diode.

The final refinement adds two back-to-back Zener diodes between the gate and source terminals (see Fig.3). They are included to prevent a supply voltage spike of more than ±20V destroying the MOSFET.

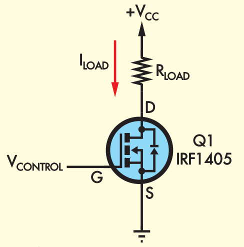


Fig.8: using a MOSFET as a switch (typical connection).

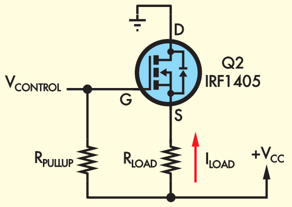


Fig.9: using a MOSFET as an active rectifier.

the heatsink to the base using two M4 × 10mm machine screws and nuts. It's then just a matter of marking the hole location, then removing the heatsink, centre-punching the marked location and drilling to 3mm.

Now use an oversize drill to carefully deburr the mounting hole. **This step is most important – if there's any metal swarf around the hole, it could punch through the insulating washer that's used to electrically isolate IC2's tab from the heatsink and create a short circuit.** Basically, it's just a matter of checking that the mounting area is perfectly smooth by running your finger over the hole.

Attaching IC2 to the heatsink

IC2's tab must be electrically isolated from the heatsink using an insulating bush and washer – see Fig.10. It's just a matter of fitting the heatsink back in the chassis, then attaching IC2's tab as shown. It's secured using an M3 × 15mm machine screw, spring washer and nut. Do the screw up firmly to ensure good thermal contact, then use your multimeter (set to a low ohms range) to confirm that IC2's tab is correctly isolated from the heatsink.

If you use a mica washer rather than a thermal insulating pad, be sure to smear both sides of the washer with thermal grease before bolting the tab down.

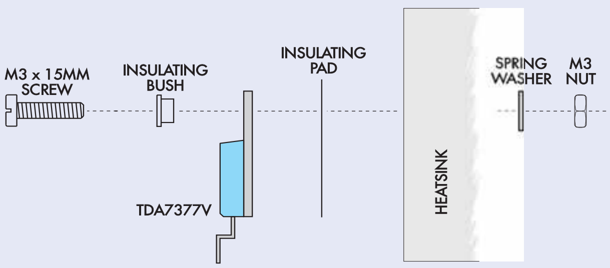


Fig.10: this diagram shows how the TDA7377V amplifier IC is attached to the heatsink. It must be electrically isolated from the heatsink using an insulating bush and pad.

Front panel

A full-size, suggested front panel design is given in Fig.11. This can be photocopied, trimmed and attached to the panel using double-sided tape. This should be done with the PC board and heatsink assembly removed from the case. You will also have to temporarily remove the rocker switch if it's in place.

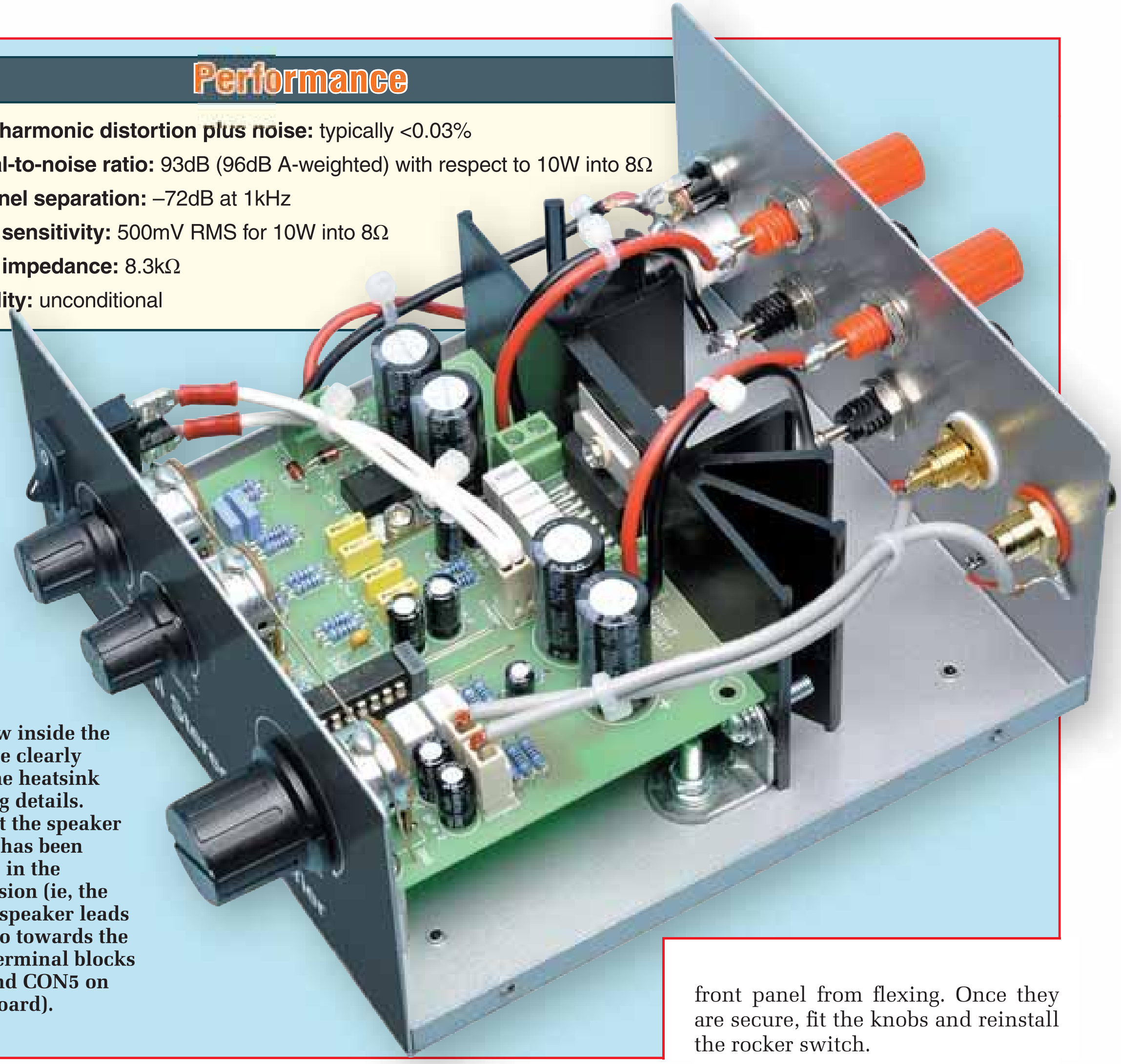
Once the front panel is in position, cover it with some wide strips of adhesive tape, then cut out the holes for the switch and pot shafts using a sharp knife. The adhesive covering will protect the label from scratches and finger marks and provide a durable finish.

The PC board and heatsink assembly can now be permanently installed in the chassis. Before sliding it in, fit a nut onto the threaded boss of each pot and wind it all the way up to the pot body. That done, place the assembly in the case and secure it via the heatsink brackets and the screws that go into the front spacers.

Now wind the pot nuts forward until they contact the rear of the front panel, then fit three more nuts to the pots from the front. The six pot nuts can now all be tightened to lock the pots firmly in place and prevent the

Performance

Total harmonic distortion plus noise: typically <0.03%
Signal-to-noise ratio: 93dB (96dB A-weighted) with respect to 10W into 8Ω
Channel separation: –72dB at 1kHz
Input sensitivity: 500mV RMS for 10W into 8Ω
Input impedance: 8.3kΩ
Stability: unconditional



This view inside the prototype clearly shows the heatsink mounting details. Note that the speaker polarity has been reversed in the final version (ie, the positive speaker leads should go towards the rear of terminal blocks CON4 and CON5 on the PC board).

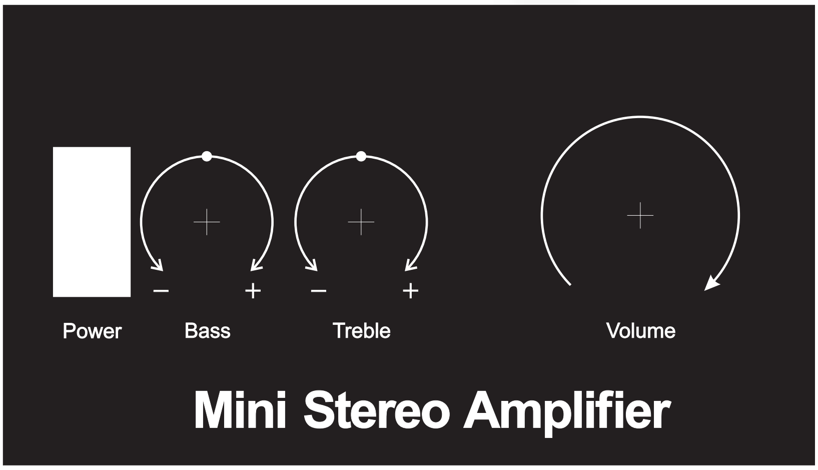


Fig.11: Full-size front facia and lettering.This can be photocopied and attached to the front panel using double-sided tape

front panel from flexing. Once they are secure, fit the knobs and reinstall the rocker switch.

Chassis wiring

All that's left is the chassis wiring. First, cut a short length of red, heavy-duty hook-up wire, strip the ends and solder it between the centre pin of the DC socket and the middle tab of the fuseholder. A 70mm length of red wire is then run from the end fuseholder tab to the +12V input terminal on the board (ie, at CON3).

Now connect the two remaining tabs on the DC socket together and run a 90mm length of black heavy-duty wire to the ground (–) terminal of CON3. In fact, the easiest way to do this is to strip the insulation from the hook-up wire back at least 15mm and wrap the wire around both these tabs before flowing solder over it. Because one of the tabs goes

Parts List – High-Performance 12V Stereo Amplifier

1 PC board, code 843, available from the *EPE PCB Service,* size 97mm × 78mm
1 vented aluminium case (Jaycar HB-5444)
1 55mm 'fan'- type heatsink (Jaycar HH-8570)
1 SPST mini rocker switch (Jaycar SK-0975)
2 4.8mm female spade lugs
2 small steel brackets (steel or aluminium)
4 M4 × 10mm machine screws
4 M4 nuts
4 M4 spring washers
1 2.1mm ID chassis-mount DC power socket (Jaycar PS-0522)
1 low-voltage M205 chassis-mount fuseholder (Jaycar SZ-2020)
1 M205 6.5A fast-blow fuse
2 red insulated binding posts (Jaycar PT-0453)
2 black insulated binding posts (Jaycar PT-0454)
1 red insulated RCA-type phono socket (Jaycar PS-0276)
1 white insulated RCA-type phono socket (Jaycar PS-0278)
2 16mm knobs (Jaycar HK-7762)
1 24mm knob (Jaycar HK-7764)
3 2-pin terminal blocks (5.08mm spacing)
3 2-pin polarised headers (2.54mm spacing)
3 2-pin polarised header connectors (2.54mm spacing)
1 TO-218 mica or silicone insulating washer (with bush)
5 M3 × 6mm machine screws
1 M3 × 10mm machine screw
4 M3 × 15mm machine screws
2 M3 spring washers
2 M3 nuts
4 M3 × 12mm tapped nylon spacers
4 M3 × 9mm tapped nylon spacers
1 500mm length of red heavy-duty hook-up wire
1 500mm length of black heavy-duty hook-up wire
1 300mm length of medium-duty hook-up wire
1 400mm length of single-core shielded cable
1 300mm length of 0.71mm tinned copper wire
Heatsink compound (if using a mica insulating washer)
8 100mm cable ties
3 additional nuts for pots

Potentiometers
1 100kΩ linear dual-gang 16mm potentiometer (VR2 – B100k)
1 50kΩ linear dual-gang 16mm potentiometer (VR3 – B50k)
1 10kΩ log dual-gang 16mm potentiometer (VR1 – A10k)

Semiconductors
1 TL074 quad op amp (IC1)
1 TDA7377V quad power amplifier (IC2) (available from: **www.littlediode.com**)
1 IRF1405 MOSFET (Q1)
2 15V 1W Zener diodes (ZD1,ZD2)
1 1N4004 rect. diode (D1)

Capacitors
4 2200µF 25V electrolytic
1 100µF 25V electrolytic
2 47µF 16V electrolytic
4 4.7µF non-polar (NP) electrolytic
1 1µF 25V electrolytic
6 470nF MKT
1 100nF MKT
2 10nF MKT
4 4.7nF MKT
2 10pF ceramic

Resistors (0.25W, 1%)

5 100kΩ	4 4.7kΩ
8 22kΩ	2 1kΩ
4 10kΩ	4 10Ω

to the metal thread of the DC socket, this connects the negative rail to the case and improves the shielding.

Next, connect the speaker outputs, again using heavy-duty red and black hook-up wire (see photos). These leads run from CON4 and CON5 to the binding post terminals on the rear panel and should be made long enough so that they don't touch the heatsink.

Note that we've reversed the output terminal polarity compared to our prototype to compensate for the inverting preamplifier stage. Ultimately, though, it doesn't matter greatly, as long as both pairs of binding posts are connected the same way around (ie, the loudspeakers are not in anti-phase with each other).

The power switch wiring is next. This can be run using two 95mm lengths of medium-duty hook-up wire. Begin by stripping about 8mm from one end of each wire and crimping them to two polarised header pins using pliers. Once you have crimped them, flow some solder into each junction so that it can't come apart.

After soldering, insert the two pins into one of the plastic header blocks then strip about 5mm from the other ends of the wires. These ends are then attached to 4.8mm female spade connectors (a ratcheting crimper will do the best job) which are then pushed onto the switch terminals.

Alternatively, solder the wires directly to the switch terminals if that's what you prefer – be careful not to overheat and damage the plastic switch body.

Wiring the phono sockets

All that remains is the wiring to the RCA-type input sockets. These are connected using two lengths of shielded/screened cable, which run back to two polarised pin headers situated behind the volume control.

Begin by cutting two 150mm sections of shielded cable, strip 20mm of insulation from each end and twist the copper screening braid wires together. Then strip 5mm of insulation from the inner wires.

At one end, tin the shield and inner wires, then crimp them into polarised header pins and flow solder into the crimp pin so it can't come apart. Note that it's necessary to twist the shield wires tightly before soldering them, so that they fit into the header pins.

After soldering, insert the pins into the two remaining plastic header blocks. In each case, the inner wire of the shielded cable must go to the '+' side of the header block (see Fig.4). This means that when the headers are plugged in, the inner wire of each header must be to the left, as viewed from the front of the PC board.

The shield leads must be to the right, so that they connect to the earth pattern of the PC board when the headers are plugged in.

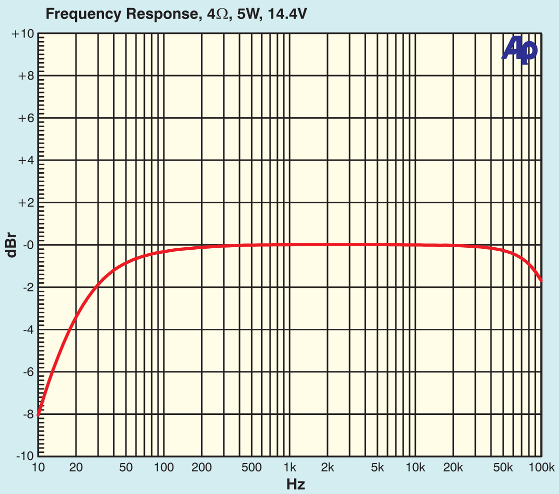


Fig.12: this shows the amplifier's frequency response for a 4Ω load with the tone controls centred. The –3dB point is around 25Hz. This is purposefully a little high to reduce the chance of 'motor-boating' with a sagging supply voltage under load.

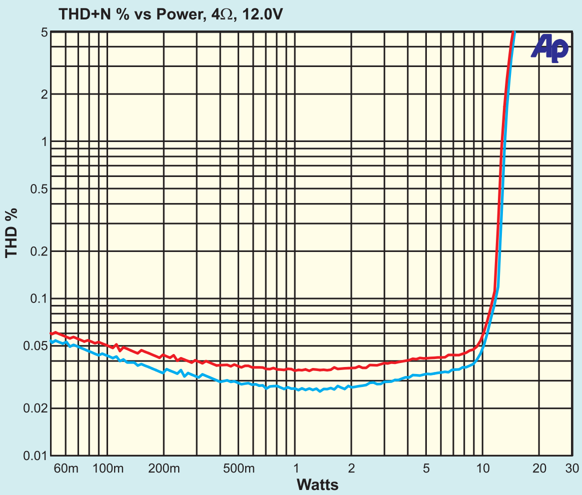


Fig.13: this graph of THD+N vs power is similar to that shown in Fig.1, except that the amplifier is powered from a 12V 4A switchmode supply. As you can see, performance doesn't suffer much, except that full power output is reduced due to the lower supply voltage.

Connecting the GND terminal

If you build the unit into the specified metal case, then it will not be necessary to connect the GND terminal to the case. That's because the circuit earth is connected to the case via the DC power socket, while the potentiometer bodies are earthed to the case via the nuts used to secure them.

In fact, if you do connect the GND pad to the case under these circumstances, you could get an 'earth loop'.

Conversely, if you elect to house the board in a plastic case, then it will be necessary to connect the GND pad to the ground (–) terminal of CON3. Alternatively, it can be connected to the negative terminal of the DC socket.

Similarly, if the pots are not directly secured to a metal chassis (ie, you don't fit the nuts), then the GND terminal should be connected to metalwork. You can do this by securing a solder lug to the base of the case and then running a short lead between it and the GND pad on the board.

The other ends of the shielded leads can then be soldered to the RCA phono sockets. In each case, the inner lead goes to the centre terminal of the socket, while the shield wire is soldered to the solder tag.

As stated above, it's necessary to use insulated phono sockets for the inputs. After connecting them, it's a good idea to check that neither phono socket surround is shorted to the case (if they are, the performance will suffer). You can do this by using your multimeter to check for continuity between the outside metal surround of each phono socket and the case. You should get an open circuit reading for both sockets.

If the meter does indicate a short, check that the shield wires are not touching the case at the metal tags. If they are, just bend the tags forward slightly until the short is cleared.

The assembly can now be completed by plugging the other ends of the shielded leads into the headers on the PC board. Don't get them mixed up – the left input (white socket) should go to the header on the left side of the PC board, and vice versa.

Once the wiring is complete, use some cable ties to secure the various leads as shown in the photos. This not only keeps them looking tidy but will also prevent them from coming adrift.

That's it – construction is complete.

Final testing

Now for a final test. Install the 6.5A fuse into the fuseholder and connect a signal source (eg, a CD player) and a pair of speakers. Be sure to connect the speakers in phase and don't cross-wire the leads.

Now turn the volume knob all the way down, switch on and slowly turn the volume up. If you hear audio loud and clear then all is well! If not, switch off immediately and check the chassis wiring carefully. If there's a problem, it's a good idea to first measure the voltage across the power terminal block, to make sure power is actually reaching the board.

If that doesn't solve the problem, you'll need to recheck the component placement and orientation, as well as the solder joints. If the fuse blows, then you likely have a short circuit in your chassis wiring, because the earlier tests on the board would have shown up any shorts on the board itself.

Assuming all is well, put on your favourite CD and enjoy the sound! ***EPE***