By Nicholas Vinen

High-performance stereo valve preamplifier

This stand-alone stereo valve preamplifier is based on the Currawong amplifier (November 2014-January 2015) but has a new power supply which runs off a low-voltage DC supply. It has very good performance, especially for a valve preamp, with low distortion and a very high signal-to-noise ratio of 105dB. It's easy to build too, with the preamp and power supply all on one PCB.

OUR FIRST VALVE preamplifiers were single-channel (mono) designs based on the 12AX7 twin triode (in the November 2003 and February 2004 issues). That design was also incorporated into the Currawong valve amplifier mentioned above. However, we have had a number of requests for a stereo version of the preamp and when we looked at the original mono design from 12 years ago, we realised that we could make a number of significant improvements.

So for a start, this new design is stereo so you don't need to build two separate units (which involved at least three PCBs). It also has a more compact and improved switchmode power supply which is on the same board as the rest of the components

Also, the earlier design had exposed components on the top of the board which operated at 250V DC, necessitating the application of silicone sealant to render it safe – not a very attractive option. The new design still "shows off" its components but they are visible through a clear acrylic case, protecting the user from electric shocks.

The overall performance is quite a lot better than the earlier design. Take a look at the graphs from our Audio Precision System Two, shown

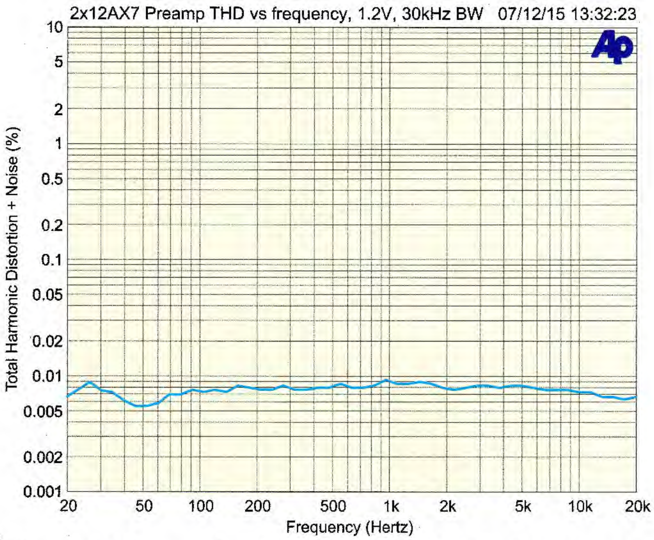


Fig.1: total harmonic distortion plotted against frequency for an input of 300mV RMS and an output of 1.2V RMS (full power for a typical power amplifier). The measurement bandwidth is 30kHz in order to chop out any residual switching artefacts from the power supply while still measuring some of the harmonics of higher audio frequencies. The result is essentially flat with frequency.

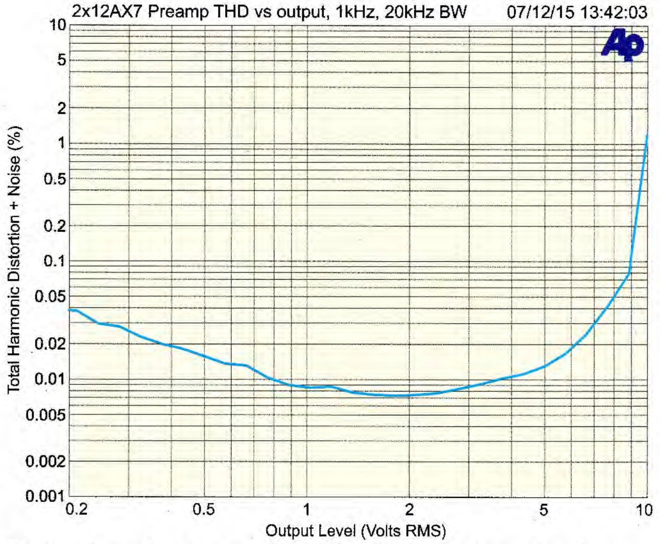


Fig.2: distortion versus output amplitude. For signals below 1V (ie, <250mV RMS input), noise starts to affect the measurement while for signals above 3V RMS out, the intrinsic second harmonic distortion of the valve begins to dominate. Distortion rises dramatically for outputs above about 9V RMS as parasitic capacitances interact with the higher slew rate.

in Figs.1-4. If you compare these to the graphs for the mono preamp in the February 2004 issue (pages 32 & 33), you will see that this is a big improvement with lower distortion across the board and no high-frequency rise. The frequency response is pretty flat, with a very slight rise in response at both 20Hz and 20kHz, due to reduced feedback effectiveness at these extremes.

One of the changes in our circuit is that we've put the volume control pot at the input end rather than the output end. This greatly reduces the chances of overload and gives lower output impedance and lower valve plate loading. In theory, it would increase the noise but in practice this design has ended up with a better signal-to-noise ratio.

Besides stereo music, another application for a 2-channel valve preamp might be for use as a musical instrument preamplifier, either with two mics on one instrument or two separate instruments. For this application, we have provision for a mixed output with a pot that controls how the two inputs are mixed. This pot, and its associated RCA connector, can be left off for stereo applications.

Since 12AX7 filaments are designed to run from 12.6V, the circuit has been designed to run off 15V DC, with an on-board regulator providing the correct filament voltage. However, we have tested the preamp with a 12V DC supply and it had little effect on performance so that is a valid option. A 12V automotive supply should be fine as it will normally be above 12.6V most of the time (assuming the battery charge state remains high).

Features & Specifications

- Stereo preamplifier with volume control
- Uses two 12AX7 dual triodes (socketed)
- Variable gain: -100dB to +12dB
- Low distortion: <0.01% THD+N @ 20Hz-20kHz, 1.2V output (see Figs.1 & 2)
- Flat frequency response: +1,-0dB 20Hz-20kHz (see Fig.3)
- Channel separation: >85dB @ 1kHz, >60dB @ 20kHz (see Fig.4)
- Signal-to-noise ratio: 105dB relative to 1V input (20Hz-20kHz bandwidth)
- Power & HT presence indicator LEDs
- RCA socket inputs & outputs
- Power supply: 13-15V DC @ 650mA
- Power supply reverse polarity protection
- Onboard power switch
- No transformer winding necessary
- Optional mixed output for use with musical instruments.
- Fits in a custom-designed clear laser-cut acrylic case

Circuit description

The full circuit is shown in Fig.5. Both channels are shown in full, along with the power supply, although the operation of the two channels is identical.

Looking at the left channel only, the signal comes in via RCA socket CON1 and passes through an RF-rejecting low-pass filter comprising a 100Ω resistor with a ferrite bead on one of its leads and a 100pF ceramic capacitor. The signal is then AC-coupled to 50kΩ volume control potentiometer VR1a via a 470nF MKT capacitor.

The attenuated signal is then AC-

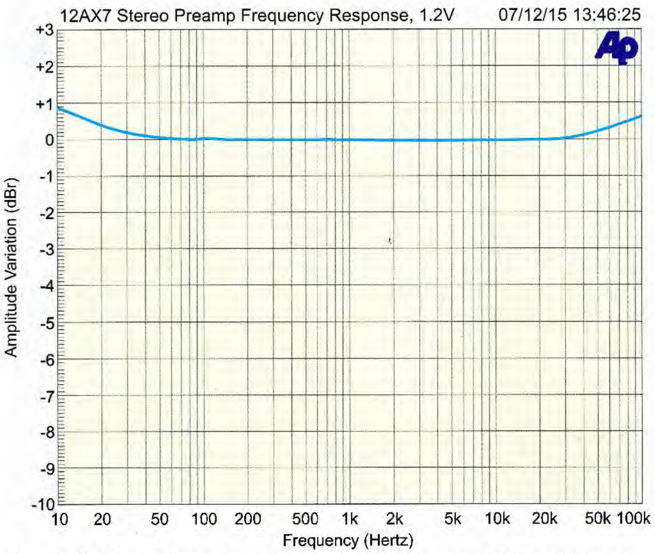


Fig.3: the frequency response for the preamplifier is quite flat but there is a slight rise in the response below 50Hz due to the increasing impedance of the feedback circuit; feedback starts to drop off, allowing the gain to rise. There is a similar rise above 30kHz, however this is well above the audio band. A small bump is visible at 100Hz due to low levels of mains hum being picked up.

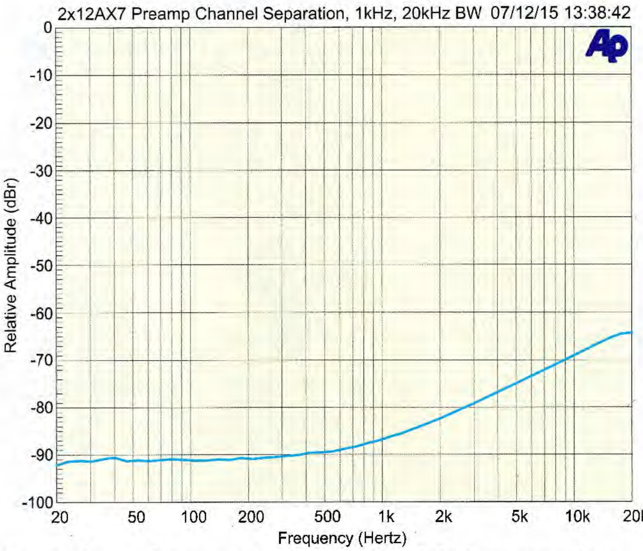


Fig.4: channel separation is very good, being more than 90dB below 400Hz, rising to around -65dB at the upper end of the audio band. This was measured with the other channel input terminated with a low impedance. The signal coupled through from one channel to the other at higher frequencies is relatively undistorted so should not result in undesirable intermodulation.

coupled to the grid of triode V1a via another 470nF MKT capacitor and a 22kΩ RF stopper. This stopper is quite important. Without is, a fair bit of hash from the power supply can couple into the valve and then be amplified. A 1MΩ bias resistor shunts any grid leakage to ground and biases the grid to near-0V.

V1a operates with a current of around 360μA, set by the combination of its 270kΩ anode resistor and 3.3kΩ cathode resistor. The amplified signal at its anode is coupled to the grid of V1b with a 220nF capacitor and the grid is biased with another 1MΩ resistor to ground.

Since V1b needs to handle a higher signal voltage, it runs at around 1.5mA, set by its 68kΩ anode resistor and 680Ω cathode resistor. The output at its anode is coupled to output connector CON3 via another 220nF capacitor, with a 1MΩ resistor setting the DC level to 0V.

AC-coupled negative feedback

The same output signal is also fed back to V1a's cathode via a pair of parallel 470nF capacitors and a 10kΩ resistor. The 10kΩ resistor forms a 4:1 voltage divider with V1a's 3.3kΩ cathode resistor. Say a 100mV positive step is applied to V1a's grid. This will turn V1a on harder, pulling its cathode negative and thus V1b's grid will be pulled negative. That will cut off V1b in turn, causing its anode voltage to rise. Once its anode voltage has risen by 400mV, the 4:1 divider will have caused V1a's cathode to increase by 100mV.

Since it's the grid-cathode voltage which determines how much current a valve conducts, the 100mV increase in V1a's cathode voltage effectively cancels out the 100mV increase in its grid, so it will be back to conducting roughly the same current it was initially. As its anode swing is a tiny fraction of the anode voltage of around 150V, it will therefore reach a steady state. Thus overall gain of the circuit is accurately set to 12dB by this negative feedback network.

Mixed & panned outputs

The preamp is intended to be used in stereo applications, with the two channels handling independent signals. However, it could be used as a musical instrument preamplifier. In this case, you can use it as two mono preamplifiers with the two outputs mixed together. For this configuration, VR2 and CON5 are installed and CON3/CON4 can be omitted.

In this case, the output of each channel is mixed by VR2. VR1 still controls the overall output level and with VR2 at mid-setting, an equal amount of each input signal is mixed into the output. As VR2 is rotated clockwise, the output contains more of the amplified signal from CON2 and less of that from CON1 and the opposite is true if it's rotated anti-clockwise.

Basically, VR2 can be regarded as a pan control, panning from one channel to the other.

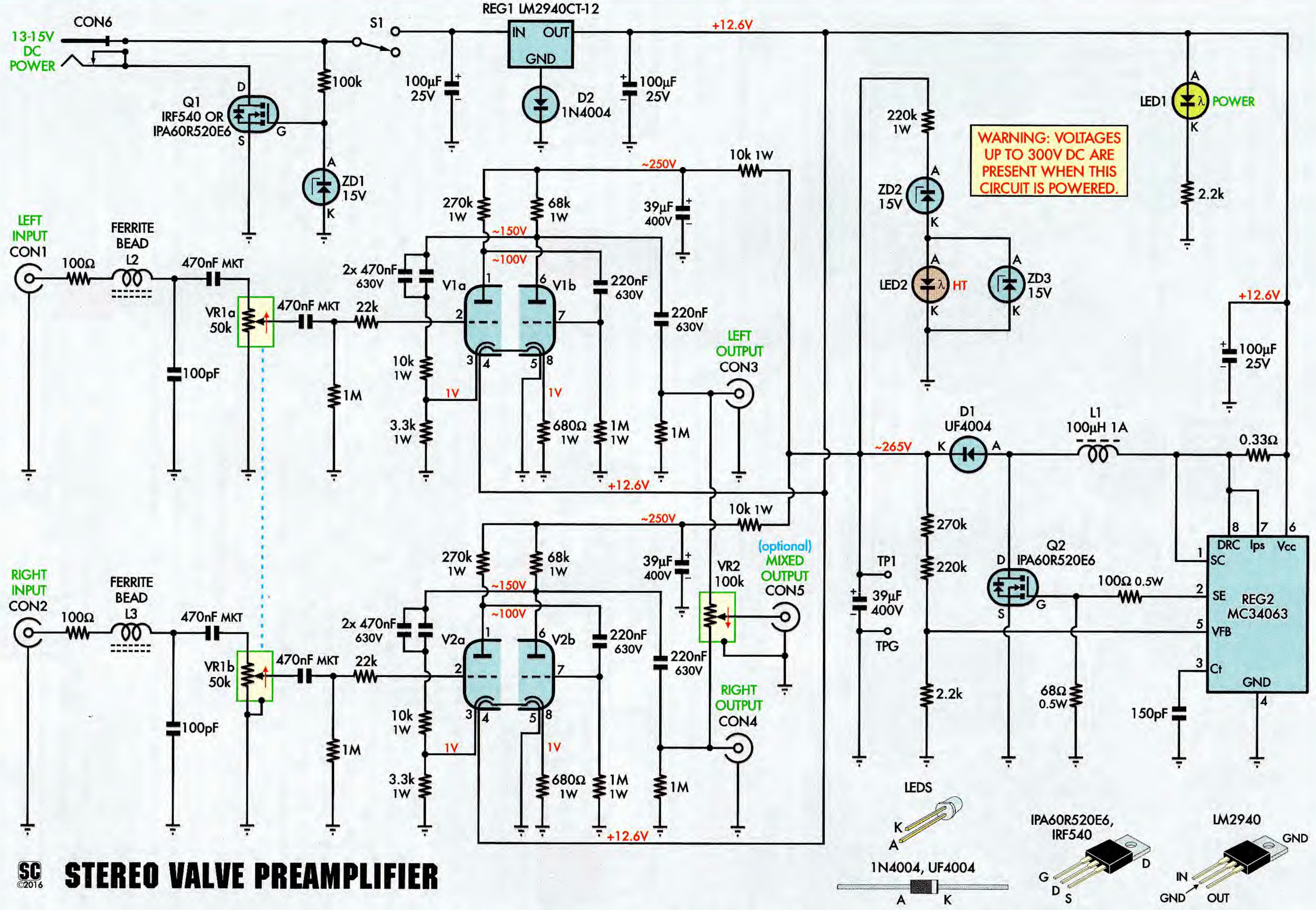
Note that if VR2 is fitted, V1b and V2b are loaded with around 50kΩ and the output impedance is increased. Still, as long as the device being fed

WARNING! HIGH VOLTAGES

High DC voltages are present in this circuit. In particular, the power supply produces an HT voltage of up to 285V DC and this voltage and other high DC voltages derived from it are present on various parts of the circuit.

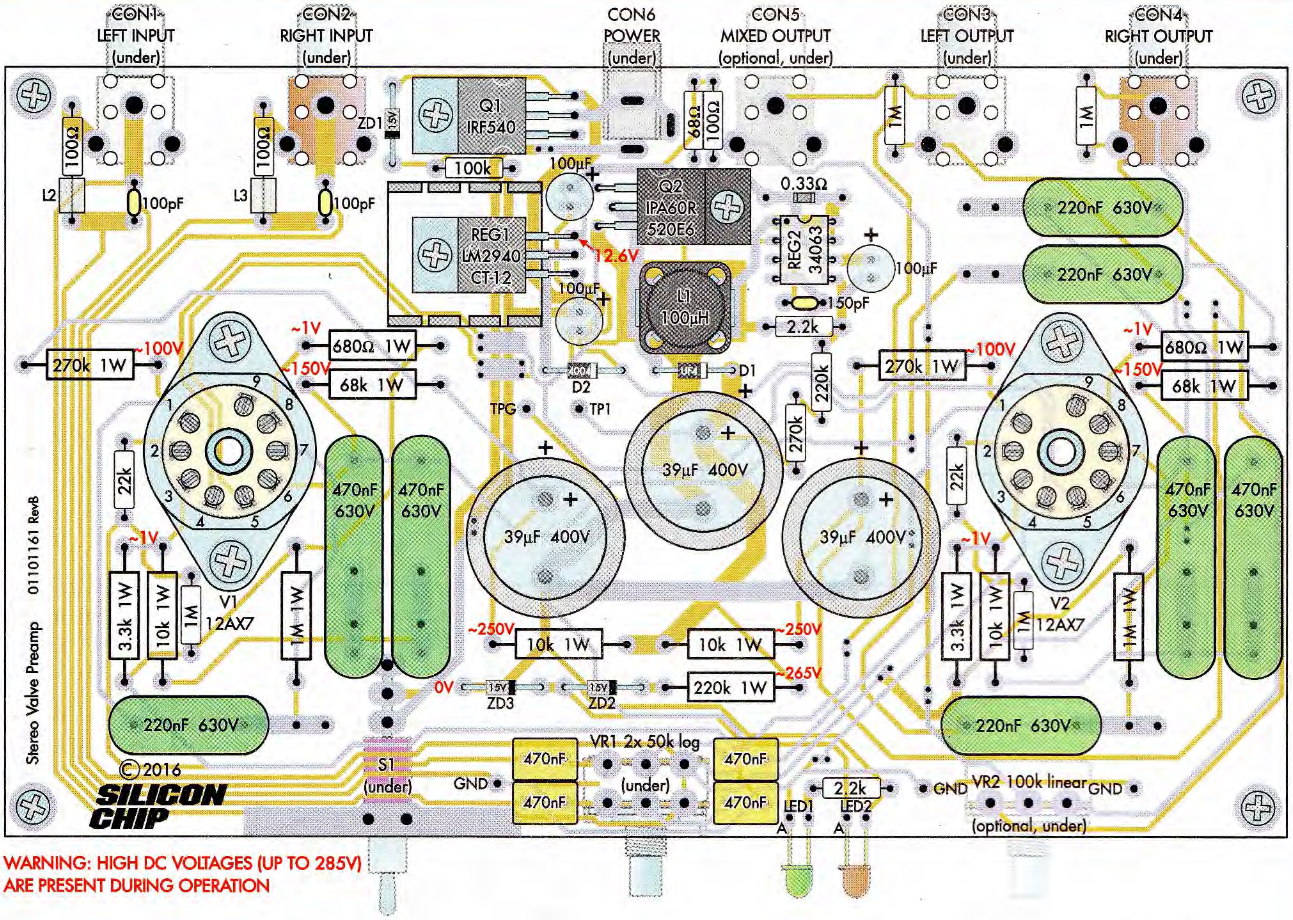
Do not touch any part of the circuit when power is applied otherwise you could get a severe or even fatal electric shock.

The red LED (LED2) in the circuit indicates when high voltages are present. If it is lit, the power supply and various parts on the PCB are potentially dangerous. **Before applying power, the completed preamplifier must be mounted in a suitable case and fitted with a Perspex cover as described in Pt.2 next month to ensure safety.**



SC ©2016 **STEREO VALVE PREAMPLIFIER**

Fig.5: the complete stereo valve preamplifier circuit. Each channel uses a 12AX7 dual triode with an overall gain of four times (12dB). Amplification is done in two stages, with negative feedback around both to set the gain and also cancel distortion. The circuit runs off a nominal 15V power supply which is regulated to 12.6V for the filaments, while a ~265V HT rail is produced by switchmode regulator REG2 and high-voltage Mosfet Q2.


CON1 LEFT INPUT (under)
CON2 RIGHT INPUT (under)
CON6 POWER (under)
CON5 MIXED OUTPUT (optional, under)
CON3 LEFT OUTPUT (under)
CON4 RIGHT OUTPUT (under)
Q1 IRF540
ZD1 15V
100k
Q2 IPA60R 520E6
0.33Ω
REG1 LM2940 CT-12
12.6V
REG2 34063
100µF
L1 100µH
150pF
2.2k
68Ω
100Ω
1M
100Ω
L2
L3
100pF
220nF 630V
270k 1W
~100V
~1V
680Ω 1W
~150V
68k 1W
D2
D1
TPG
TP1
220k
270k
39µF 400V
22k
470nF 630V
3.3k 1W
10k 1W
1M
1M 1W
V1 12AX7
V2 12AX7
~250V
10k 1W
~265V
0V
ZD3
ZD2
220k 1W
VR1 2x 50k log (under)
470nF
GND
S1 (under)
VR2 100k linear (optional, under)
LED1
LED2
A
Stereo Valve Preamp
01101161 RevB
© 2016
SILICON CHIP
WARNING: HIGH DC VOLTAGES (UP TO 285V) ARE PRESENT DURING OPERATION

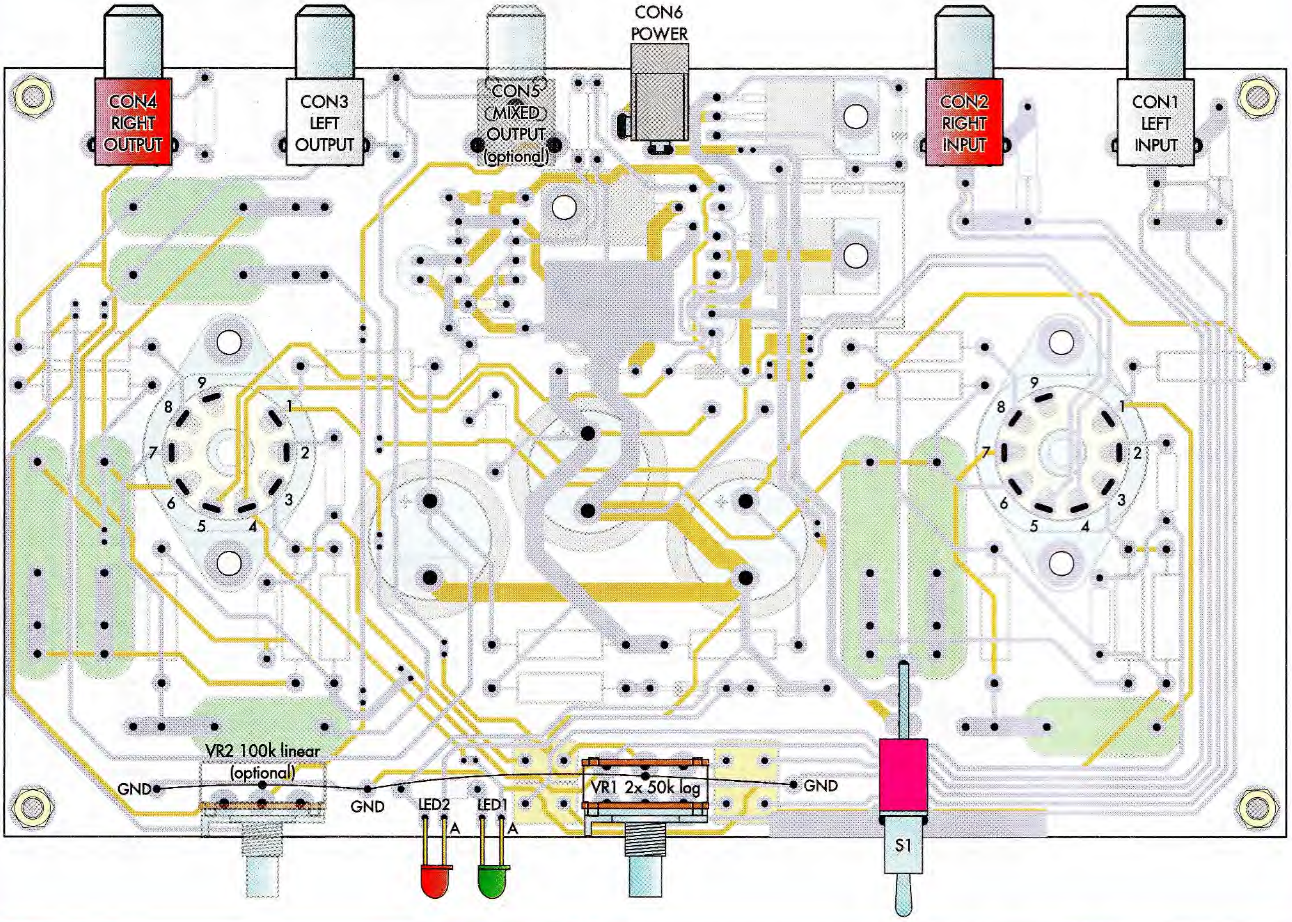

CON4 RIGHT OUTPUT
CON3 LEFT OUTPUT
CON5 MIXED OUTPUT (optional)
CON6 POWER
CON2 RIGHT INPUT
CON1 LEFT INPUT
VR2 100k linear (optional)
GND
LED2
LED1
A
VR1 2x 50k log
S1

Fig.6: top and bottom PCB overlay diagrams. Use these as a guide when assembling the PCB. Start by fitting the components to the top side, which is everything except the connectors, power switch, pots and LEDs. Note the wires used to earth the pot bodies to the nearby GND pads. Leave VR2 and CON5 out if building a stereo preamplifier. CON3 and CON4 are optional if VR2 & CON5 are fitted.

has a relatively high input impedance, this should not be a problem.

Power supply

A DC input of around 13-15V is required at CON6. As mentioned earlier, supply voltages down to 12V are acceptable however the filaments of V1/V2 will run at lower power than they are designed for.

Mosfet Q1 provides reverse polarity protection, with much lower voltage loss than a simple diode, even a Schottky type. If the supply polarity is correct, Q1's gate is pulled positive with respect to its source and so ground current can flow back to CON6 normally. However, if the supply polarity is reversed, Q1's gate is pulled negative and thus its channel will not conduct. Its body diode is also reverse biased in this condition so the only current that will flow is a few microamps through ZD1 and its series 100kΩ resistor.

ZD1 protects Q1 in case the supply voltage spikes above 20V for more than a very brief period.

Power switch S1 interrupts the supply to REG1, a low-dropout automotive 12V regulator. Its ground pin is "jacked up" by around 0.6V by diode D2, increasing its output to around 12.6V to suit the filament requirements of the 12AX7 valves. 100µF input bypass and output filter capacitors are provided and these should ideally be low-ESR types for supply stability.

LED1 indicates the presence of the 12.6V rail. As well as running the filaments directly, this rail also supplies switchmode regulator REG2 which is configured as a boost regulator to produce the HT supply.

When REG2's internal transistor is switched on, current flows through the 0.33Ω shunt, into pin 1 (switch collector), out of pin 2 (switch emitter) and through a voltage divider formed by 100Ω and 68Ω resistors. The voltage produced by this divider drives the gate of high-voltage logic-level Mosfet Q2.

So when REG2's internal switch is on, Q2 is biased into conduction and it pulls current through the 0.33Ω shunt and inductor L1 to ground. This charges up L1's magnetic field. REG2 has an internal oscillator that we've set to around 100kHz using a 150pF capacitor from pin 3 (Ct) to ground. L1 continues to charge either until the ~7.5µs period set by this oscillator expires or the current builds to around 1A, at which point the voltage across the 0.33Ω shunt exceeds the ~300mV current trip level, as sensed by pin 7 (Ips).

In either case, REG2's internal transistor is switched off and the 68Ω resistor quickly pulls Q2's gate to 0V, switching it off. This causes the magnetic field in L1 to begin collapsing, which continues to "push" current through the inductor in the same direction as it was flowing before it was interrupted.

Since the "input" side of L1 is still connected to the 12.6V supply, the only way for current to continue to flow is for high-voltage ultrafast diode D1 to become forward biased. As a result, the voltage at D1's anode increases dramatically. Before L1's magnetic field can collapse completely, the oscillator in REG2 causes its internal transistor to switch back on, recharging it and repeating the cycle.

When the circuit is first powered up, the voltage at D1's cathode will start at around 12V but as the three 39µF 400V capacitors charge up, this voltage will continue to rise until it reaches nearly 300V.

One of two things then happens. The voltage is either limited by the fact that the current limit enforced by REG2 prevents any more energy flowing into L1 in each cycle than is consumed by V1 and V2, or the voltage rises high enough that the voltage at the voltage feedback pin of REG2 (pin 5) rises above 1.25V. If this happens, REG2 will skip pulses until the output voltage drops, then it will switch back on to regulate said voltage to the set level.

However, we have designed this circuit so that it can't quite produce a high enough output voltage to regulate properly. This is because the pulse skipping that's used to regulate the output voltage causes sub-harmonics of the 100kHz switching frequency to

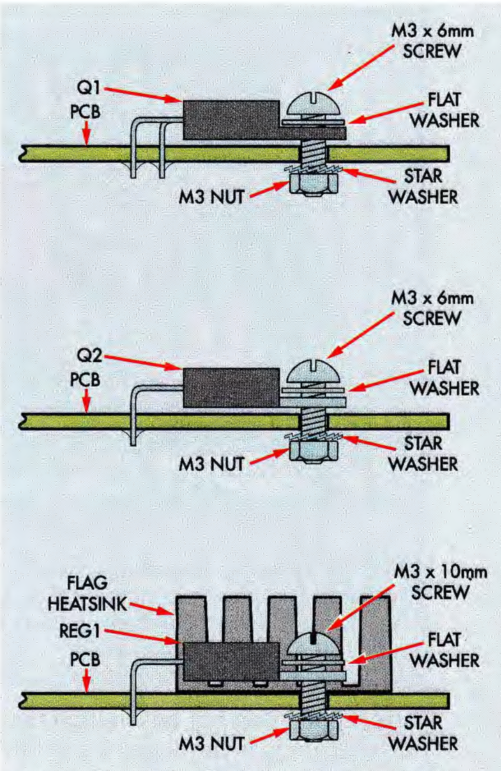


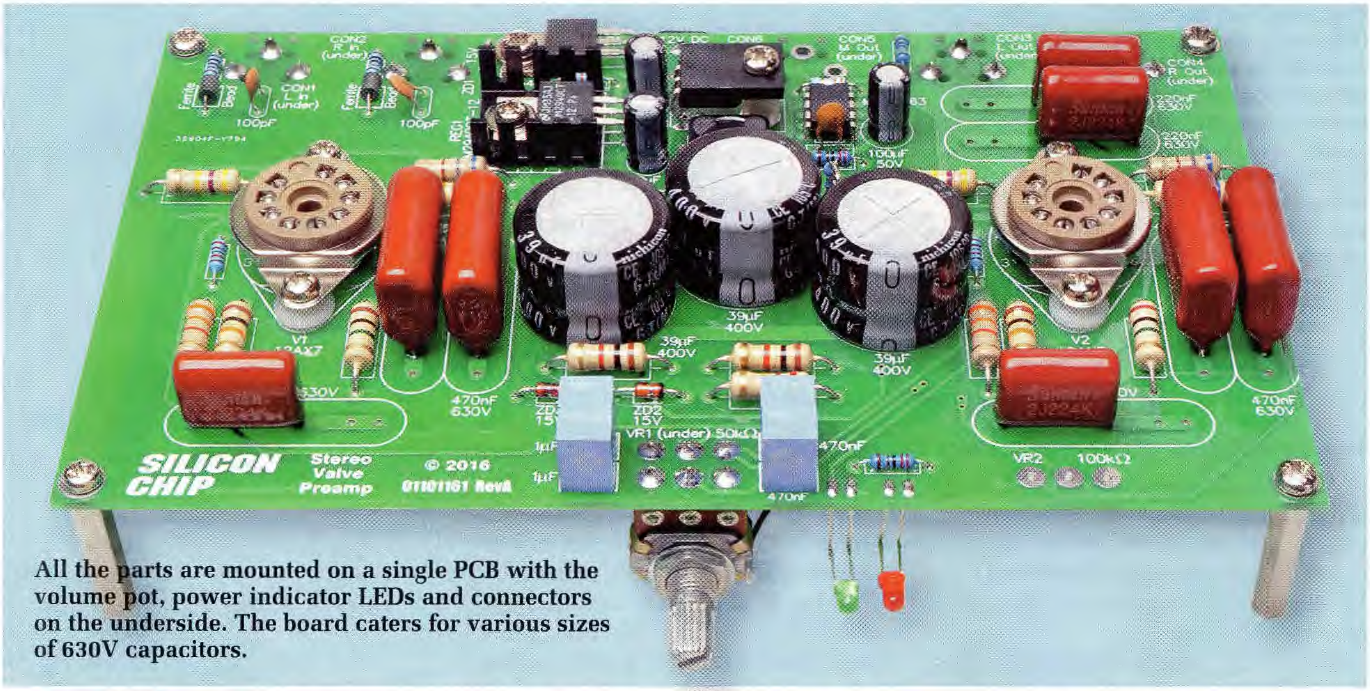
Fig.7: mounting details for Q1 (top), Q2 (middle) and REG1 (bottom). Note that a longer machine screw is used for REG1 and that Q1 is in a fully insulated package with its centre lead bent over and soldered closer to the body than the other two.

be radiated and depending on how many pulses are skipped, these could be in the audio band (ie, below 20kHz) and could couple into the preamplifier, reducing its signal-to-noise ratio.

This means that the HT voltage is not actually regulated but that isn't much of an issue as the 12AX7s will run happily off quite a wide range of voltages; anywhere in the range of 250-300V will do. The feedback divider really only exists to prevent damage in case one or both valves are removed, fails or becomes disconnected during operation. In this case, it will limit the HT rail to around 285V DC.

The actual operating HT voltage will depend on a few factors but mainly on the exact value of L1, the 0.33Ω shunt, REG2's current limit voltage sense threshold and the 150pF capacitor. These all affect how much energy L1 can store for each cycle, or in the case of the 150pF capacitor value, the maximum number of charge cycles per second.

We've set the circuit up so that in most cases, the actual HT voltage produced should be high enough for



All the parts are mounted on a single PCB with the volume pot, power indicator LEDs and connectors on the underside. The board caters for various sizes of 630V capacitors.

correct operation but not so high that pulse skipping is employed (ie, below the ~285V regulation target). In our prototype, it reaches 280V after about 30 seconds and eventually drops to about 265V once the valves have fully warmed up.

Construction

All parts are carried on the main PCB and assembly is quite straightforward. It should only take a couple of hours for experienced constructors. The board itself is coded 01101161 and measures 170 x 102mm. Referring to the PCB top side overlay diagram Fig.6, begin by fitting all the smaller resistors.

It's best to check all the resistor values with a DMM before fitting them. Don't forget that the 68Ω and 100Ω resistors must be 0.5W types and that two of the other 100Ω resistors have ferrite beads slipped over their leads before they are soldered in place. The 0.33Ω resistor should also be fitted now, whether it's a through-hole or SMD type.

Follow with diodes D1 and D2 and zener diodes ZD1-ZD3. Don't get the three different types mixed up and pay careful attention to polarity. This is indicated by the cathode stripes shown on Fig.6 and the PCB silkscreen.

Having done that, solder inductor L1 in place. This is most easily done by first applying a little flux paste to the pads, then adding some solder to one of the pads – the right end if you are right-handed or left if you are left-handed. Then place the inductor next to its final position, heat the solder on that pad and slide the component into place.

You will find that once it contacts the solder, it will take a few seconds to heat the component up to the point where it will adhere and you can then move it into its final location. It's then just a matter of adding solder to the opposite pad and continuing to heat it until it adheres to both the pad and component lead. Finally, go back to the other side, add some fresh solder and heat it further, again making sure it forms a good fillet.

Next, solder REG2 to the board. Don't use a socket and make sure its pin 1 dot is at upper-left as shown on Fig.6. Press it down flat on the PCB before soldering the pins. Follow with the larger (1W) resistors, using a similar procedure as before.

Now bend the leads of Q1 and Q2 down through 90° about 5mm from the body of each component and attach them to the board using M3 x 6mm machine screws and nuts, with a shakeproof washer under each nut and a flat washer under the head. Don't get these two components mixed up – Q2 should be encapsulated in black plastic while Q1 may have a metal tab (if you are using an IRF540) – see Fig.7 for details.

Once the screws are done up tightly and the parts checked for proper alignment, solder and trim the leads.

Having done that, solder the ceramic and MKT capacitors in place. These can all go in either way around, as they are non-polarised. Refer to Fig.6 to see which value goes where.

Now fit regulator REG1. The procedure is the same as for Q1 and Q2 except that a flag heatsink is positioned under the regulator's tab and an M3 x 10mm machine screw is used to secure it instead of an M3 x 6mm screw. Make sure that the regulator's body and heatsink are square and that the screw is done up tightly before soldering the leads – see Fig.7.

Fitting the valve sockets

The valve sockets are retained mechanically, to avoid placing stress on the solder joints when inserting and removing the valves. Each is held in place with two M3 x 10mm machine screws, with a Nylon nut and two Nylon washers used to form a spacer. Fit a shakeproof washer under the nuts (see the photos for details).

Basically, it's just a matter of inserting an M3 x 10mm machine screw through the top of the two mounting holes on the valve socket and screwing a Nylon nut onto each thread. Do the nuts up tight, then slip pairs of Nylon washers over each screw shaft and feed these through the mounting holes on the PCB. You'll need to coax the nine solder tabs into the slots on the PCB, then the whole thing should drop into place.

Use the shakeproof washers and

nuts to fasten it in place, make sure the nuts are done up tightly, then solder and trim the nine tabs on each socket.

You can now solder the three small and three large electrolytic capacitors in place (see Fig.6). In each case, make sure that the longer lead goes through the hole nearest the + symbol.

Underside components

Now it's time to fit the components on the other side of the board – see Fig.6. The RCA connectors fitted are CON1-CON4 (for a stereo preamplifier) or CON1, CON2 and CON5 (mixed mono preamplifier for instruments). CON1 and CON3 are white, CON2 and CON4 are red and CON5 can be black.

Unfortunately, white RCA sockets aren't that easy to come by. We sell a set of four on our Online Shop, including red, white, black and yellow. These have a slightly different footprint to the types available from Jaycar and Altronics but as you can see from our prototype, the leads can be bent so that they fit. In fact, they are a little easier to fit than the other type and as a bonus, have a consistent mounting height, unlike some types which can vary between different colours.

Whichever sockets you are fitting, make sure they are pressed down fully onto the PCB and are perpendicular to the board edge before soldering the three pins. You can also fit DC socket CON6 now, on the same side of the board, again making sure it's nice and square before soldering.

Before fitting the pot(s), you will need to use a file to scrape off a small area of the passivation on top of the body so that you can solder an earth wire in place. Basically, it's just a matter of holding the body in a vice using a couple of scrap pieces of timber to prevent damage and then a few passes with a file should reveal a shiny surface. Don't breathe in the dust produced; it may be toxic.

If your pot(s) have long shafts, you will also want to cut them short now. Use a hacksaw and file to cut it/them to no more than 15mm. Then, referring to Fig.6, solder the pot or pots in place on the underside of the board. Solder some tinned copper wire between the provided GND pads, across the top of the pot body(s), then solder the wire to the pot(s) to "earth" them.

Now fit power switch S1 in place, making sure it's first pushed down fully onto the PCB. Finally, install LED1 and LED2. Check Fig.6 to determine the required orientation, then bend the LED leads through 90° 6mm from the base of the lenses. Solder the LEDs in place on the underside of the board, with the horizontal portion of the leads 13mm from the bottom of PCB. This may be easier to do if you cut a 13mm cardboard spacer first.

Testing

The first step is to check that the HT power supply is working but before doing this, temporarily attach the four tapped spacers in each corner using an M3 machine screw.

Test points are provided to monitor the HT voltage, near the centre of the PCB, however it's easier and safer to use DMM alligator clip leads to connect to the anode of ZD3 (negative lead) and the right-hand end of the 220kΩ 1W resistor (positive lead) – see the 0V and ~265V markings on Fig.6. Set your DMM to a range which will read 300V DC and plug the power supply into the PCB but not the mains.

. . . continued on page 96

Parts List

1 double-sided PCB*, code 01101161, 170 x 102mm
1 set of clear acrylic laser-cut case pieces*
1 small tube acrylic adhesive
4 rubber feet
1 15V 1A plugpack
2 12AX7 dual triode valves
2 9-pin valve sockets (Jaycar PS2082)
1 100μH 12x12mm SMD inductor* (L1) (Murata 48101SC; element14 2112367)
1 50kΩ 16mm dual log pot (VR1)
1 100kΩ 16mm linear pot (VR2; optional, see text)
2 knobs, to suit VR1 & VR2
1 mini TO-220 flag heatsink, 6073B type
2 ferrite beads (L2,L3)
2 white switched RCA sockets (CON1,CON3)*
2 red switched RCA sockets (CON2,CON4)*
1 black switched RCA socket (CON5; optional, see text)*
1 PCB-mount DC socket to suit plugpack (CON6)
1 PCB-mount right-angle mini SPDT toggle switch (S1) (Altronics S1320)
2 M3 x 6mm machine screws
5 M3 x 10mm machine screws
4 M3 x 32mm machine screws
7 M3 shakeproof washers
3 flat washers, 3mm I.D.
7 M3 nuts
4 M3 Nylon nuts
8 Nylon washers, 3mm I.D.
4 M3 x 12mm Nylon machine screws
4 6.3mm M3 tapped Nylon spacers
4 12mm M3 tapped Nylon spacers
4 25mm M3 tapped metal spacers
1 200mm length 0.7mm diameter tinned copper wire

Semiconductors
1 LM2940CT-12 12V 1A low-dropout regulator (REG1)
1 MC34063 switchmode regulator (REG2)
1 IRF540 or IPA60R520E6* N-channel Mosfet (Q1)
1 IPA60R520E6* 600V N-channel Mosfet or equivalent (Q2)
1 green 3mm LED (LED1)
1 red 3mm LED (LED2)
3 15V 1W zener diodes (ZD1-ZD3)
1 UF4004 ultrafast diode or equivalent (D1)
1 1N4004 1A diode (D2)

Capacitors
3 100μF 25V low-ESR electrolytic
3 39μF 400V low-profile snap-in electrolytic (Nichicon LGJ2G390MELZ15* from Mouser)
4 470nF 63V MKT
4 470nF 630V metallised polyester
4 220nF 630V metallised polyester
1 150pF disc ceramic
2 100pF C0G/NP0 disc ceramic

Resistors (1W, 5%)
2 1MΩ
2 270kΩ
1 220kΩ
2 68kΩ
2 10kΩ
2 3.3kΩ
2 680Ω

Resistors (0.25W, 1%)
4 1MΩ
1 270kΩ
1 220kΩ
1 100kΩ
2 22kΩ
2 2.2kΩ
1 100Ω 0.5W
2 100Ω
1 68Ω 0.5W
1 0.33Ω through-hole or SMD 1206 resistor*

* Available from the Silicon Chip Online Shop; details in next month's issue.

Valve Preamplifier

... continued from page 35

Make sure nothing conductive is near the PCB and it isn't close to the edge of your bench. Then, keeping clear of the assembly, plug the power supply into mains. Within about one second of power being applied, the HT voltage should reach 285V or thereabouts and stabilise, with the green and red LEDs lit. Either way, switch off power and wait for it to discharge to a safe level (below 40V) before continuing.

If there's a fault, once the HT rail has discharged, check component placement and orientation as well as solder joint integrity.

Assuming all is well, connect regular probes to your DMM but leave it on the 300V (or higher) range. Power the board back up and measure the voltage between pins 4 and 5 on both valve sockets (see Fig.6). You should get a reading close to 12.6V. Now check the voltages at the other pins relative to GND. You should get ~285V for pins 1 and 6 and close to 0V for pins 2, 3, 7 and 8. Pin 9 is not connected to anything.

You can now switch the power off and push the two valves into their sockets. They will be stiff, especially if this is the first time the sockets have been used. You may find it easier to gently rock them in. While you can in theory install the valves with HT voltage present, it's much safer to wait for it to decay first.

With the valves in place, power back up and check the HT voltage, using the test pads in the centre of the board. It should rise to around 270V at first and then slowly decay to around 250-260V as the valves warm up and their operating current builds.

In the unlikely event that the HT supply remains above 280V and there are no board or valve faults, this may be because component variations are causing the supply to deliver more current than it's designed to. The simple solution is to reduce the value of the 150pF capacitor to 120pF. This will increase the switchmode frequency and reduce the duty cycle and should bring the HT back in line. **If you need to do this, don't forget to wait for LED2 to go out before working on the board.**

Finally, perform a live signal test. Switch off, wait for LED2 to go out and connect a signal source to CON1/CON2 and an amplifier to CON3/CON4. Next, turn the volume right down, power on and wait 30 seconds or so for voltages to stabilise. Then press play on the signal source and slowly advance the volume until you hear clean, undistorted sound.

If the sound is distorted or missing, switch off and carefully check the component values around each valve socket as well as the solder joints.

Putting it in the case

That's all for this month. In the second and final article next month, we'll go over the details of how to put together the custom laser-cut case and fit the PCB inside it. **SC**

Pt.2 By Nicholas Vinen

High-performance stereo valve preamplifier

Having described how our new stereo valve preamp works and how to put the PCB together, now it's time to build the case. This houses the PCB so that you can still see all the components but can't touch the high-voltage sections. It's custom designed and made from clear acrylic, glued and screwed together.

THE STEREO Valve Preamplifier PCB was originally designed to fit inside a UB2 "jiffy" box. However, because it has connectors on the back and controls on the front (including a volume knob), it's pretty much impossible to actually get it inside such a box even after drilling the required holes.

Rather than compromise the practicality of the device to allow it to fit in that box, by doing something like putting the inputs on the front or side, we have designed a custom case instead. This is made in three sections which screw together so you can get it apart if necessary. We think the result is much more attractive – hopefully you'll agree after seeing the photos of the prototype.

We're using clear 3mm acrylic to show off the components and the cutting pattern is shown in Fig.8. It can be downloaded from the SILICON CHIP website (free for subscribers) in a variety of formats to suit typical laser cutter software, including AutoCad DXF.

We used a 60W CO_2 laser operating at full power and at a speed of 8mm/s (10mm/s also works but we're being a little conservative to ensure it cuts reliably).

The parts are cut from a sheet of acrylic measuring at least 230 x 315mm. The cuts shown in red are done first, in case the sheet isn't perfectly flat. In this case, if the outer sections were cut first they could shift slightly and then the inner cuts would be less accurately placed. We have optimised the cutting path to eliminate the duplication of overlapping cuts, although typical laser cutters have an extremely high degree of repeatability so this will generally not affect the quality of the end result.

The case pieces incorporate holes in the top for the valves and some cooling slots over the power supply. The front panel has holes for the power switch, volume pot, its anti-rotation tab and the two power LEDs, while the rear panel has holes for the input and output RCA sockets and the DC power connector. There are also four screw holes in the top and bottom panels corresponding to holes on the PCB, to hold the whole thing together.

If you've building the musical instrument version of the PCB with the extra panpot (VR2) and mixed output (CON5), we'll supply a modified version of the cutting diagram with these extra holes. Their locations are shown in Fig.9, relative to the pre-cut holes.

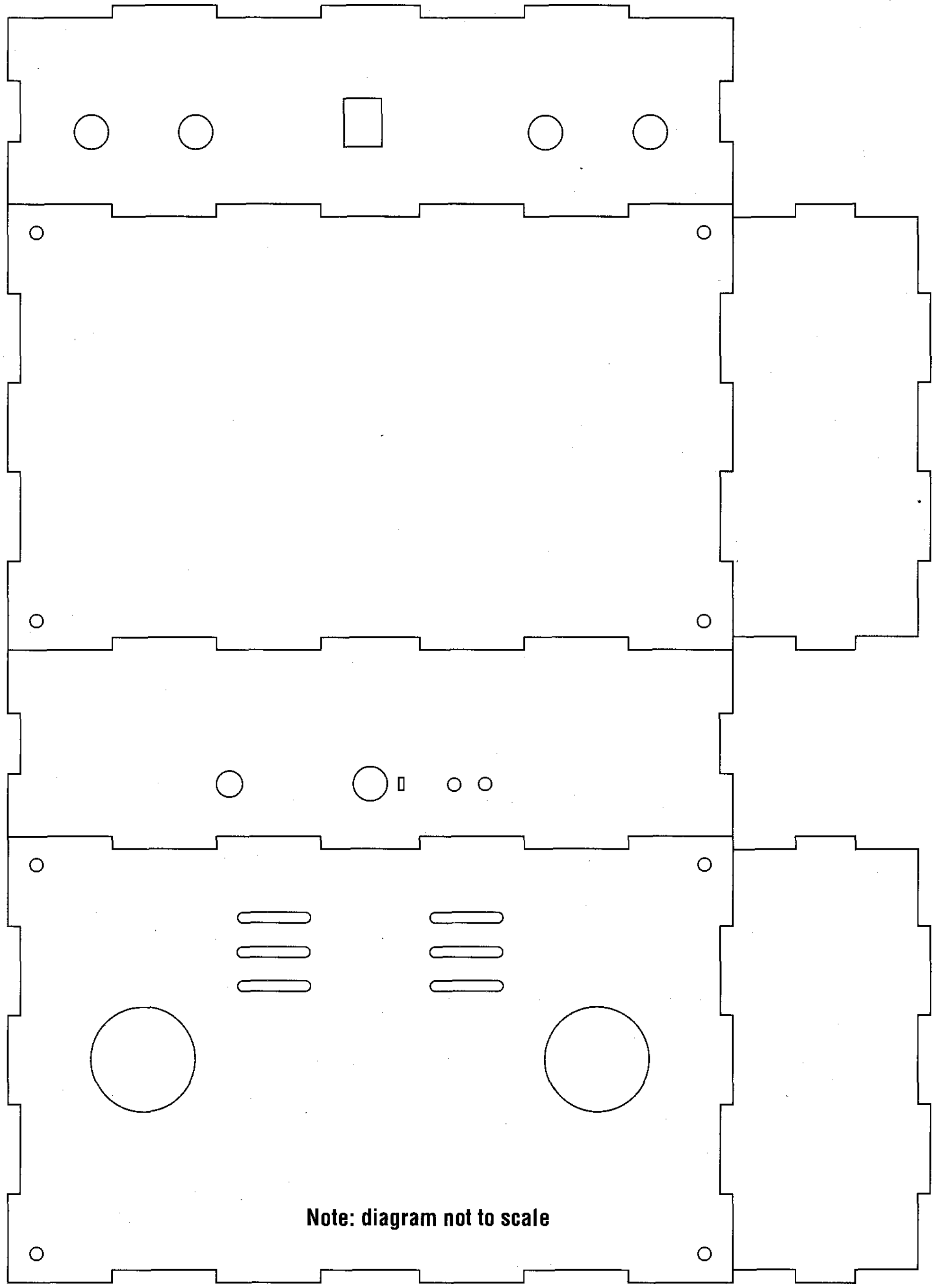


Fig.8: the laser cutting pattern used to produce the six pieces for the Stereo Valve Preamplifier case, from a 230 x 315mm piece of 3mm-thick clear acrylic (Perspex). The sections shown in red are cut first to maximise precision of the hole placement. Note the cooling slots in the top cover, which go over the power supply circuitry. The prototype lacked an onboard power switch so a hole has been added to the front panel for the final version (note: diagram not to scale).

Putting it together

You will now have the completed PCB, six case pieces (with protective film on both sides) and an assortment of machine screws, tapped spacers and feet (as specified in he parts list last month). Start by removing the two 12AX7 valves (or "vacuum tubes", if you prefer) and setting these aside so they won't get damaged. Don't put them on a surface they might roll off!

The bottom, rear and side panels of the case are glued together into one assembly which screws to the PCB. The top panel is also screwed to the PCB while the front panel is held on by the potentiometer nut(s). The PCB is sandwiched between the top panel and the bottom panel assembly and held in place with screws and spacers.

It's a good idea to check that everything will fit before gluing. Leave the protective film on the case pieces for now, so they don't get scuffed or dirty – clear acrylic shows fingerprints quite well unfortunately, so you want to avoid getting these on the inside of the case if possible, where they're hard to clean off.

Using Fig.10 as a guide, pass an M3 x 32mm machine screw through a hole in the top panel and screw on one of the shorter spacers. Do it up tight against the underside of the lid, then thread the other spacer on and repeat the procedure for the remaining corners.

Next, orientate the lid so that the valve socket holes are above the sockets, then push the protruding screw threads through the PCB mounting holes and attach the M3 x 25mm tapped

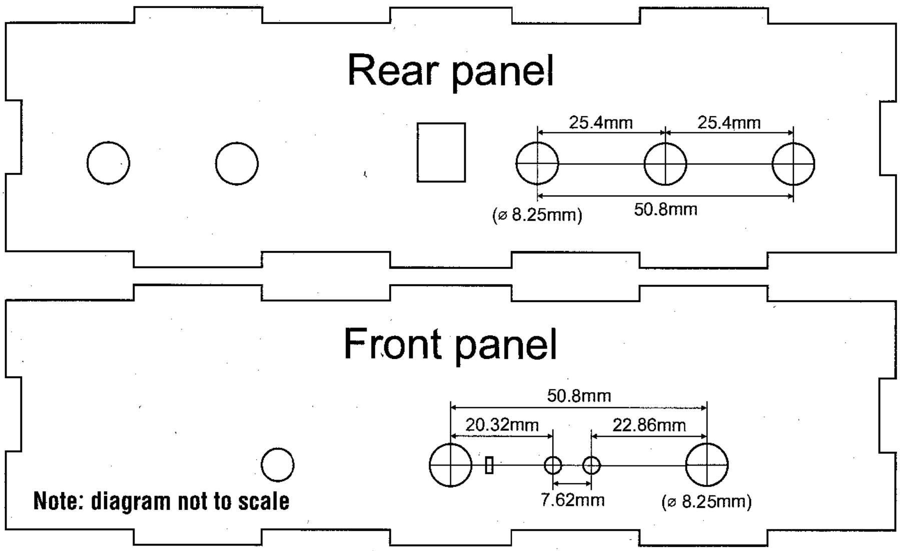


Fig.9: here's where to drill the extra holes in the rear panel (for the mixed output RCA socket, CON5) and in the front panel (for panpot VR2) if required

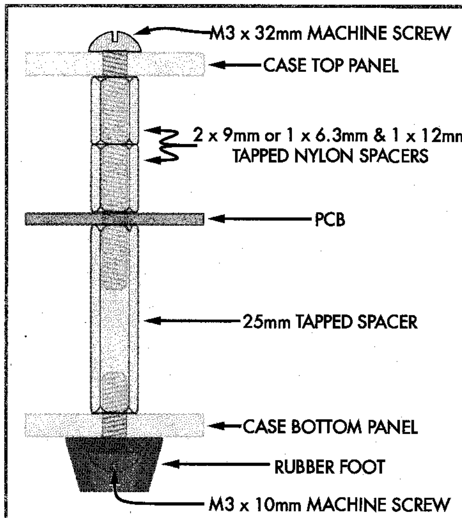


Fig.10: the top panel, bottom panel and PCB are sandwiched together using four sets of machine screws and tapped spacers. This spaces the top and bottom panels the correct distance apart for the front, back and side panels to fit. If they're too close together, add 3mm inner diameter washers in each stack as shims.

spacers to hold the PCB in place.

You can then screw the bottom panel on using the remaining 10mm machine screws (no need to fit feet just yet).

With the top and bottom panels held rigidly in place, check that the rear panel fits. The RCA sockets will be a relatively tight fit through the holes but assuming they have been soldered evenly, it should slide into place. Otherwise, you may need to use a tapered reamer to open the holes up slightly.

RCA sockets

Note that, on our prototype, we used the RCA sockets which are available from our online shop, because it's hard to find white sockets of this type at the usual retailers. They have a slightly different footprint but will fit on the board with some slight bending of the pins.

We mention this because this is likely to affect the height that the sockets sit at, so if you use a different type, they may not line up with the holes in the case. In that case, you would either need to adjust the RCA socket height by melting the solder joints (tricky) or simply enlarge the panel holes until they fit through.

You may find, depending on the exact height of the spacers you have used, that the rear panel will be slightly too tall to fit between the top and bottom panels. In our prototype, it was an almost an exact fit but spacer lengths can vary slightly. In this case, you may need to add some sort of a shim (eg, a washer or two) somewhere in the spacer stack to increase the gap enough for the panel to fit correctly.

Assuming it fits OK, remove the nut from the pot(s) and fit the front panel. This can be held in place temporarily with the potentiometer nut(s). If necessary, reach behind the front panel using small pliers to push the LEDs through their respective holes.

With the front and rear panels in place, it should be possible to slide the side panels into place.

Gluing the case

Once you've confirmed that everything fits, disassemble it and peel the protective film off the pieces. Reassemble the top and bottom halves and the PCB as before, using the screws and tapped spacers, to form a rigid assembly.

The parts are glued together using a specialised, solvent-type plastic adhesive formula. We used SciGrip "Weld On" 16 fast set clear, medium-bodied solvent cement. This is available from Plastix [Sydney (02) 9599 2499, Northern Beaches (02) 9939 0555].

With a clean cloth at hand to wipe up any excess, the next step is to glue two pieces of the case together (see below) by applying a moderate amount of the adhesive to all the mating surfaces and then pressing and holding them together. Try to avoid getting any of the adhesive on the faces, especially via your fingers.

Start by gluing the rear panel to the base. Make sure it's pressed in fully until the adhesive sets (this takes a few minutes). You may find that friction holds it in for you, otherwise you

Changing The Preamp's Gain

The circuit as presented last month has a maximum gain of four times (12dB). While the output swing is limited by the valve operating conditions, if you have a situation with low-level input signals, you may wish to increase this. This can be achieved by increasing the value of the 10kΩ 1W resistors to the lower left of each valve socket.

For example, using a 15kΩ resistor will give a gain of 15kΩ ÷ 3.3kΩ + 1 = 5.5 times or 15dB. A 22kΩ resistor will give 22kΩ ÷ 3.3kΩ + 1 = 7.5 times or 17.5dB. And a 30kΩ resistor gives a gain of 10 times or 20dB.

Note though that increasing the gain will slightly prejudice the performance by increasing the distortion and reducing the signal-to-noise ratio. Having said that, the performance as specified is sufficiently good that you probably won't notice the difference.

The rear panel of the case has holes for the RCA stereo input and output sockets and a square cut-out to provide access to the DC power socket. Note the ventilation slots in the top cover above regulator REG1 and Mosfet Q2.

Warning

Voltages of up to 285V DC are present on the PCB when power is applied and while ever the red LED is lit. Do not operate this unit without the top cover in place.

may have to hold it. Once it's nice and rigid, carefully unscrew the base and remove the two panels which are now joined. You can then glue the two side panels on, again making sure they are pressed in fully before it sets.

Don't use great dollops of glue but don't be too stingy either. If you're quick, you can wipe off any excess from the outside with a cloth.

Leave this assembly aside for some time (ideally, overnight) before re-fitting it using the four screws. If you have stick-on rubber feet, stick them on now, otherwise attach screw-on feet using the four mounting screws. That's what we did on the prototype (see photos).

Fitting the front panel

All that's left is to push the front panel in place, ensuring the LEDs pass through the two holes, attach the potentiometer nut and fit the knob. If you find the front panel won't sit flat, it may be that the LEDs are protruding too far and pushing on it. Pressing them carefully back into the case should fix this. You will need to do this to both LEDs or the result could look strange.

It's now ready to use. Remember that it takes 10-15 seconds each time you power it up before the HT rail rises to its normal level and all the bias voltages stabilise. Until then you aren't likely to get much output. Ideally, wait 30 seconds or so after powering up for it to achieve a reasonable level of performance. The valves will continue to warm up for some minutes and this may affect performance slightly.

Note: Altronics have announced that they are in the process of preparing a kit for this project, Cat. K5192.
The plan is for it to be a complete kit, including the parts required to make the case. However, the case may not be identical to the one described here. Instructions regarding the case assembly will be supplied with the kit. **SC**

Improving The Bass Response

A reader has brought to our attention the fact that the circuit as presented last month could have a significant amount of bass attenuation when driving a fairly typical power amplifier load impedance of around 10kΩ.

This is because the 220nF output coupling capacitor is not quite large enough. Our Audio Precision test equipment has a 100kΩ input impedance and in combination with the 220nF coupling capacitor and 1MΩ onboard bias resistor, this results in a -3dB point of around 8Hz.

However, with a 10kΩ load impedance, the -3dB frequency increases by nearly a factor of 10, to 72Hz. We've confirmed this by simulating the full preamp circuit (see the graph in Ask SILICON CHIP). This is not an unrealistic load impedance; for example, our Ultra-LD Mk.2, Mk.3 & Mk.4 power amplifiers all have an input impedance of around 11.8kΩ.

The solution is simple: increase the coupling capacitor value. At the very least, use 470nF 630V capacitors (one in each channel) for a -3dB point of 34Hz for a 10kΩ load. Ideally, though, they should be at least 1µF. The simplest way to achieve this is to use pairs of parallel 470nF capacitors, one on either side of the PCB for each channel. This will yield a -3dB point below 20Hz.

ASK SILICON CHIP

Got a technical problem? Can't understand a piece of jargon or some technical principle? Drop us a line and we'll answer your question. Send your email to silicon@siliconchip.com.au

Improving the bass of the valve preamplifier

Your valve preamplifier in the January 2016 issue is a great project and I'm interested in building one to try the "valve sound" for myself. I've tried many op amp and discrete semiconductor preamp designs over the years and have tended to prefer the sound of simpler designs based around a complementary-feedback pair of bipolar transistors or discrete op amp types over the ultra-low THD op amp versions, so I expect to like the valves.

There is a key specification missing in the article though: the preamplifier's output impedance. Is it low enough to drive a solid-state power amplifier's input?

Certainly, the output coupling capacitor is far too small. With a power amplifier input impedance, of say, 10kΩ, the -3dB point is about 72Hz with the 0.22µF value used. How many builders are going to try it in their system and conclude that valves are no good for bass? Surely at least 1µF and preferably 2µF would be more like it?

While I admire your clever design of efficient power supplies, I'm not keen on the use of a switching supply so close to the signal stages; especially one running only just above the audio band at 100kHz. There are small isolation transformers available with 230VAC output that could provide a simpler linear solution for the HT, eg, Hammond 185d230, available from E14 or Mouser for about $27.

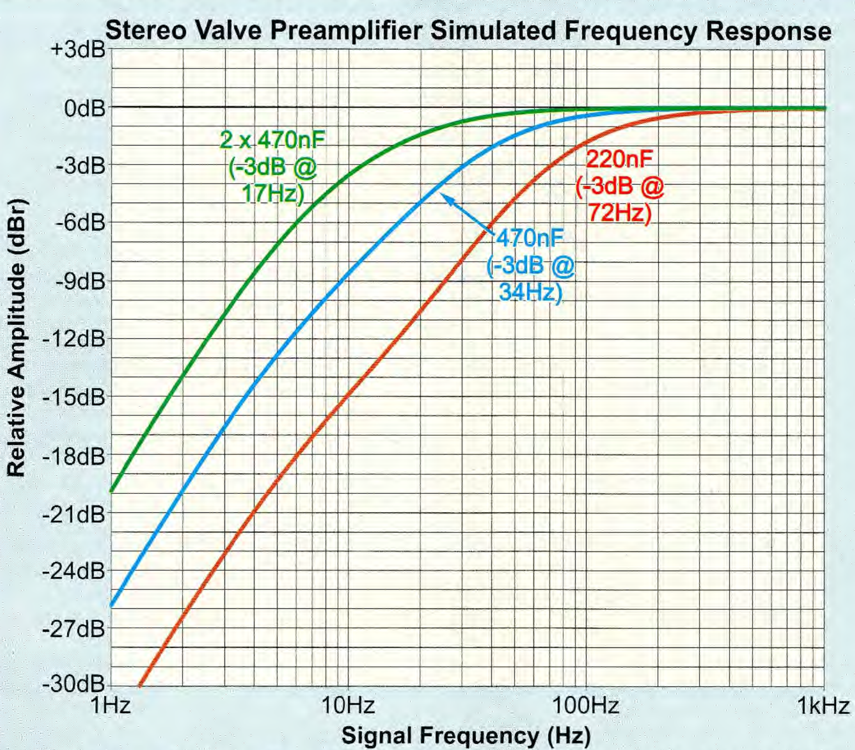


Fig.1: this graph shows the expected frequency response of the Stereo Valve Preamplifier with different output coupling capacitors.

I also have to say your PCB looks like a lot of it was done by an autorouter. Hence I'm thinking to build a hard-wired version on tag strip with my own simpler power supplies and internal power transformers – the design is simple enough for this to work well I think.

Another thing: the low dropout regulator on the heater supply is quite expensive and hard to get. Would not a 7812 work just as well, providing the input voltage is always above 15V DC?

Keep up the great work and thanks as always for a fantastic magazine. (I. B., via email).

● That's a very good point about the size of the output coupling capacitors and it's a bit of an oversight on our part. However, it is easily fixed, as you can substitute two 470nF/630V capacitors for the 220nF capacitors in each channel. You can install one above and one below the PCB.

We ran a circuit simulation to check the likely performance with one or two 470nF output coupling capacitors feeding a load of 10kΩ and as you can see in the diagram of Fig.1, a 470nF capacitor results in a -3dB point of 34Hz while two 470nF capacitors result in a -3dB point of 17Hz (as you suggested).

The output impedance of the preamplifier is lower than you might expect, at less than 200Ω. That is a beneficial side effect of the relatively high negative feedback in the circuit and it means that long output cables from the preamplifier will cause negligible high frequency loss. It certainly is more than low enough to drive the input of any amplifier that you are likely to consider.

As far as the power supply is concerned, do not worry about the fact that it is a switching design. The key figure to watch is the signal-to-noise ratio: -105dB is exceptional for a valve preamplifier and pretty good for all but the best solid-state preamps. We would suggest that you build the preamp as described initially. If you don't like the result, you can always substitute an analog supply later.

The PCB layout was most definitely not done by an auto-router. It was a human router (one Nicholas Vinen, in fact) and the decisions about how to run signal lines and general layout were quite painstaking, all with high performance in mind. There is simply no way in which you would get the same performance from a hand-wired layout using tagstrip or tag-board. Consider: this is 2016, not 1956!

As far as the regulator is concerned, a 7812 would give the same performance as the LM2940, provided the input supply was above 15V DC. However, we wanted to make the preamp compatible, as far as possible, with 12V plugpack supplies and that would not have been workable with the 7812.

In any case, compared with the likely cost of the valves, the bigger capacitors, PCB and case, the cost the specified regulator is not really significant.

By the way, Altronics will have this preamp available as a kit.