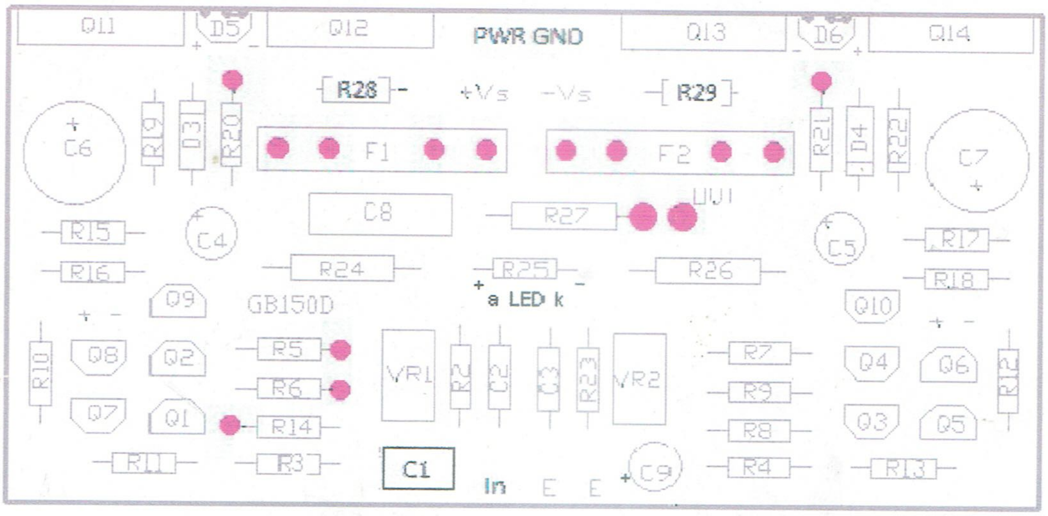
GB150D Assembly Instructions

Stuffing Guide



R1 = off board 100K if used
R2 = 750 (update)
R4 =1K
BC546C
R3, 23 = 27K
R5, 6, 7, 8 = 22R
R9 = 100K
R10, 12 = 1K1 or 1K (with BDs)
R11, 13 = 100K
R14 =10R
R15, 16, 17, 18 = 3K
R19, 20, 21, 22 = 47R
R24, 26 = 4K7 1W
R25 = wire link or red LED
R27 = 2R7 1W
R28, R29 = 0R15 2W 5% metal oxide

C1 = link if no DC, or use 1-5uF if DC on the input from a source.
C2 =330pF polystyrene
C3 = 6pF silvered mica
C4, 5 = 10uF 50V electro
C6, 7 = 330uF 63V 105C
C8 = 100nF 250V PP
C9 = 47uF 25V low ESR
C10, 11 = 10uF 50V electro

VR1= 50K trim-pot
VR2 = 50K trim-pot
F1, 2 = 3.15A delay

Q1, 2 = 5% Hfe matched
Q3, 4 = 5% Hfe matched BC556C
Q5, 6, 9 = BC546B
Q7, 8, 10 = BC556B
Q11, 12 = Vgs matched IRFP9240

Q13, 14 = Vgs matched IRFP240
D1, 2 = nolonger used
D3 = 6V2 400mW zener * see text
D4 = 5V6 400mW zener
D5 = BC556B or BD140
D6 = BC546B or BD139

Additional Parts for Options

BD140 to replace D5 BC556B sensor for piggyback to MOSFET or bolt down to heatsink mounting.
BD139 to replace D6 BC546B sensor for piggyback to MOSFET or bolt down to heatsink mounting.

Assembly

1. Identify resistors using a multimeter and insert in the positions shown above, bending the legs outwards to retain the component. R24 and R26 should be mounted on extended leads about 3-5mm above the board surface. Clip and solder all.

2. Insert Q1- Q10 and D5, 6 in the orientation shown and solder the board connections quickly. Solder

ONE PAD ON EACH, then a second pad on each, to minimise heat soak. D5 and D6 are best soldered in at full lead height,
then leads bent later to suit positioning in the heatsink. If piggyback BD139/140s are used pins 2 and 3 need to be joined as they would if inserted in the PCB.

3. Insert R28 and R29 on slightly extended leads just off the board. These could get hot.

4. Insert capacitors taking care of polarity for C4, 5, 6, 7, 10, 11. C10 and C11 are shown on the drawing below. Allow for tilting of C6, 7 (for easier MOSFET mounting to heatsink) by raising each up off the PCB slightly. Insert an off-cut wire as link across C1 holes if it is not being used, so the input signal can enter the amp.

5. VR1, 2 have staggered pins so will only go in one way (extra in line centre holes are provided for alternative industry standard 25 turn trim-pots if desired for easier setting accuracy). If horizontal type have been supplied, carefully bend outer pin inwards and align with holes before soldering. If using substitute multiturn trimpots ensure VR1 is set at about 5K from top and VR2 is centred for startup. These can be measured in circuit.

6. Fuse clips should be inserted carefully the correct way around so the end fuse-retaining flange is to the outer side. Fuse pins should be soldered well to the pads on top, and the underside for strength.

7. Flow some solder on the rear outer fold of the fuse clips to attach the test resistors. Bend leads on the two 10R test resistors and solder to the rear of the fuse clips so they angle forward and up for easy multimeter access. Fuses should be removed initially.

8. Check all purple pads shown above are soldered on the top level while still accessible, particularly R5, 6, 20, 21.

9. Q11-14 need to be handled with care to ensure the gates (left pin facing the plastic side) receive no static charge. Ensure your soldering iron is grounded and that you are not carrying any static charge. Depending on your heatsinking and mechanical layout the MOSFETs may be mounted from the top or bottom of the PCB. Insert with metal side away from the board. Insert the 9240's at locations Q11,12 on the left and the 240's at Q13,14 on the right. Solder one leg on each at the minimum height, level, then solder all remaining terminals on underside of board, one pin on each to minimise chip heating, then clip leads short.. This procedure can be easily done by inserting all MOSFETs, then turning the PCB over on a flat surface and aligning them, then soldering ONE TERMINAL ON EACH for all, then the second and third. There is no need to solder the top side pads as they are through hole plated. They can be mounted from the top or underside. Solder pads are provided both sides on the newest boards..

10. Drill and tap the heatsink using the centre MOSFET pads on the PCB or actual MOSFET holes for hole positioning. Holes should be spaced inline at –

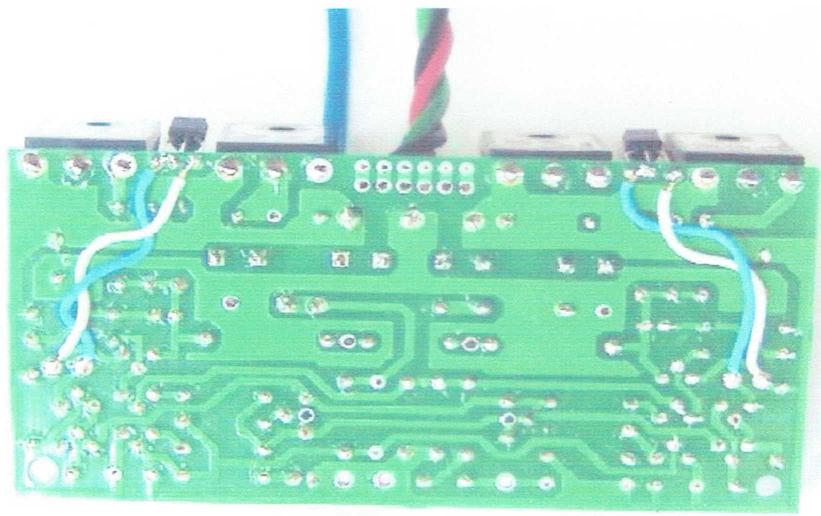
* 24mm * 34.5mm * 24mm * OR * 15/16" * 1 3/8" * 15/16" *

You can use 3mm or 1/8" screws and nuts if the heatsink allows or you can use a 'tap' tool to insert a thread. This is a small bit of similar size to a drill bit and can be bought at a good hardware shop for < $5. Make sure to buy the size for the screws you are using. The tool, in this small size, should be used by hand and carefully to work it back and forward so the thread does not clog - they can easily snap and become permanently stuck in the heatsink, so avoid excess force. Use a little oil when using the tap, and an old toothbrush is ideal for clearing the threads. The hole should be predrilled just undersize for the thread. I use a 2.7mm drill for 3mm screws. Make sure the screws will tighten down the MOSFETs properly, that you have drilled and tapped the holes deep enough first.

If using the original BC temp sensors mount the MOSFETs and trim back the silpads so the sensing D5, 6 can press against the heatsink with no silpad in the way, if mounting them to the heatsink. Bend D5, 6 forward to establish pressure contact when MOSFETs are screwed up hard. Alternatively, for better bias stability a 4.5mm tight fit blind hole can be drilled approx 5mm deep at half height of D5, 6 so they can be bent down and their heads will insert into the blind hole in the heatsink. File any mold seams so it goes in tightly and makes great contact all over.

11. An alternative simple mounting method can be used on heatsinks with a T flange. The MOSFETs can be lined up against the flat flange (6mm max) and attached using foldback spring clips over 2 MOSFETs, so 2 clips are used. The silpads need to be positioned first and the clips applied. The wire handles can be removed for a clean appearance.

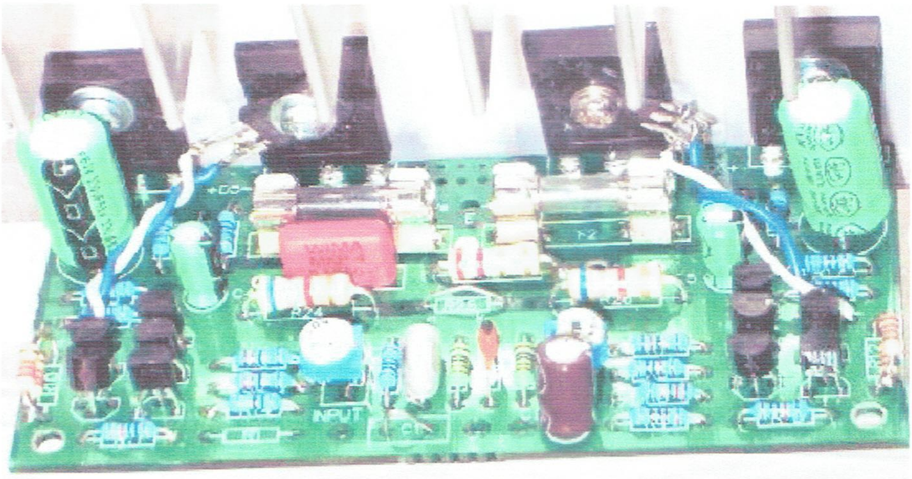
Thermal Sensing D5, 6 are diode connected transistors used as output stage thermal sensing to regulate input stage constant current source currents according to the output stage/heatsink temperature. Suitable mounting locations have been shown on the stuffing guide for these, between the output MOSFET pairs. They require hard-wiring back to the holes similarly marked + - near Q8 (for D5) and Q6 (for D6) with correct polarity. Twist the blue and white wires together and join to the mounting holes in front of Q6 and Q8 UNDER THE BOARD ensuring correct polarity and connect to the solder pads of D5, 6, + to + and - to - as shown on the stuffing guide. The white wire on each connects the single (emitter) pin (RH pin on underside) to the supply rail (outer) track connection near Q6 or Q8. CHECK! **Incorrect wiring here WILL blow transistors at switch on.** The efficacy of these will be tested on set-up as DC drift and bias stability with temperature depend on these.



Ensure D5, 6 are in intimate contact with the heat-sink when the MOSFETs are screwed against it.

Alternative BD139/BD140 sensors These are now provided in the kit as many DIYers prefer a bolt down arrangement and these transistors have a mounting hole so they can be bolted to the heatsink in the standard location (with insulators) or mounted 'piggyback' on the inner pair of MOSFET bolts and wired with the blue and white wires to the topside holes. Note that pins 2 and 3 of the BDs are soldered together (blue wire connection shown below).
Use BD140 for the BC556B and BD139 for the BC546B.

This pic is the BD mod applied to an earlier version so don't be concerned that some component differences exist, it is only to illustrate the BD fitting. The blue and white wires are now on top.



Connections External connections require appropriate gauge and type of cabling for best performance. V+,V- wires represent supply impedance and should be minimum 1.5mm multi-strand insulated copper wire of minimal length, preferably less than 15cm. They are located together with the PG (or E) power supply ground wire closely at the middle of the output section of the board for a reason - the 3 should be twisted or plaid together and brought out perpendicular to the board to minimise EM radiation of the Class AB commutation currents running in these wires intruding into sensitive input and high impedance stages. Input wiring should be quality shielded cable. In E must have a wire connection to supply E. The output cable can be OFC of 1.5mm multi-strand or other to preference. In the absence of an output R//L this 'medium' gauge provides some of the same function in 10cm minimum length at least as far as the chassis output terminal.

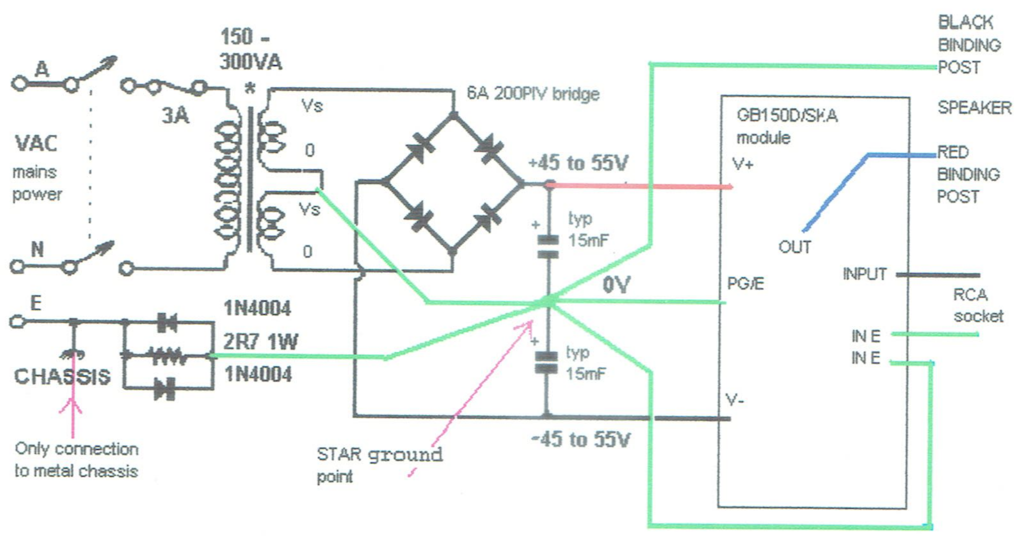
Design Limitations/Hurdles This unique and simple design, exploits the strengths of modern 'vertical' MOSFETs and is optimised for these. It achieves low distortion into nominal 4 -8 ohm loads but is intolerant of very low impedance loads. Output current is limited to just over 10A so supply voltages must be reduced so this is never reached. However, within it's load range, it delivers a very creditable < 0.025% of (mostly 2hD and 3hD) THD (typically < 0.01% 2HD at moderate levels) throughout the audio band, while being quite insensitive to power supply commutation artefacts which cause harshness, loss of detail and blurring of the image. The IRF 'complementary' MOSFETs used have slightly differing tempcos and, as a consequence, there may be some small DC drift of output point with DC coupling, so this is not recommended where heat-sinking is poor and/or extreme levels are routine.

For highest current demand (demanding 4 ohm use) the Pchannel MOSFETs are best to be IRFP9140 (optional) and lowered supplies of +/-45V are needed. D3 should be 5V6 rather than 6V2 to limit the maximum current in overload.

Requirements The amplifier module requires a split power supply and adequate heatsinking to function reliably and at its best. A power supply of 300VA and +/-45V (4 ohm load) to +/- 55V (8 ohm load) with typically 15,000F on each rail is reasonable for 150W into 4 or 8 ohms for two channel. The GB150S power supply kit/module is available with star ground plane and ripple isolation (see other products below).The heatsink should be a minimum 0.8degC/W for each, larger for heavy duty. For heavy 4 ohm load use +/- 45V and 0.65degC/W min. For economy, a single 0.4degC/W (e.g. Conrad MF30-75) heatsink can be used for both modules.

NOTE: Really oversize heatsinks will perform better at drawing heat away from the MOSFETs for improved reliability with difficult loads and higher supplies, but they also reduce the thermal feedback to the sensor diodes so the bias may not be as well controlled. In this case it is preferable to use the flatpack BD139 and BD140 instead for D5, D6 and mounting them directly on the MOSFET to reduce the thermal lag.

MONOBLOCK or Single Channel connectivity for GB150D



Earthing/Grounding On the drawing above, the transformer secondary centre tap to supply ground wire should be brought in to one end of the PS ground and the signal connections at the other, so charging spike imbalance currents are not running through the ground track between the PS capacitor grounds and the input, ground plane and spkr grounds. These are carefully separated on the GB150S supply module which is designed to minimise charging spike interaction with quiet grounding. In a dual mono chassis the ground wiring is simply duplicated. The 2R7 resistors to chassis/mains ground can be increased up to 10R or more if hum is present. There may be a ground loop formed when an external connection to both input RCA's is made. In this case join the two RCA input grounds together at the sockets and run one 10R with antiparallel diodes to chassis instead of two seperate ones. If there is still hum/buzz, try shorting across the R//diode connection to chassis.
For 2 channels running from a single power supply the arrangement pictured should work well, with duplicated module wiring. A short heavy wire strap between L and R star grounds may be needed if a ground loop causes hum.

*****_The incoming mains ground should always be connected directly to the chassis for safety!!_ ******

The transformer to bridge to capacitor bank wiring should be twisted together to reduce radiated fields and kept well away from input wiring and the modules or hum/buzz may be impossible to eliminate. The red, green and black supply to module DC power wires should be twisted together as shown on the module picture above and kept well clear of the input area of the PCB and input to RCA wires. Layout is very important for best results.

AC/DC feedback coupling The standard operation of the amplifier is with C9 in place providing unity DC gain. Any small DC on the output can be nulled by adjusting VR2. For DC coupling of the feedback loop (so DC gain = AC gain) C9 is simply linked across its pins. The amplifier is first turned on with no speaker connected and VR2 adjusted for < 5mV DC on the output. The unit then needs to be run at usual power on music and the DC offset measured and adjusted to zero. In normal operation C1 is linked out, but a 1-5uF must be fitted if there is known to be DC on the input signal. The DC can be checked across the output speaker terminals at any time and reset to zero. IF IN ANY DOUBT

WHETHER THERE IS INCOMING DC, INSERT A C1 so incoming DC is blocked. An R1 will refer this input side to ground.

Layout Amplifier modules, power supplies, transformer(s) need to be positioned in the chassis so that sensitive input stages and wiring are a comfortable (~50mm or 2") away from all power supply wires, transformers and their wiring. This is very important to ensure low background noise and best performance from the build. Wires for the inputs and the input RCA's themselves need to be well away (50mm) from the output wires and this needs to also be away from supply wires. It is often best to twist and run all transformer to power supply wires down the middle and signal ground and input wires down on the metal chassis with the twisted supply to amp module wiring up high. A metal base and lid are recommended, with the rest being metal or wood or other depending on your capabilities. A MDF or other insulating back panel can facilitate insulating the RCA and binding post connectors from the chassis.

Fine tuning When a commercial amplifier is designed, great attention is paid to optimising the layout and then this is repeated exactly to ensure repeatable performance. We can't do that in DIY as every build is different so we need to employ a few strategies to get the best from our build after it's up and running.

First is to leave the wires full length on the transformers. Often the fields radiated from these can vary in intensity and their ability to cause hum is well known. Most toroids spin on a single bolt mounting, so it's good practice if you find a hum to experiment with turning the transformer(s) on their axis for a minimum hum. Then cut and twist and route the wires.

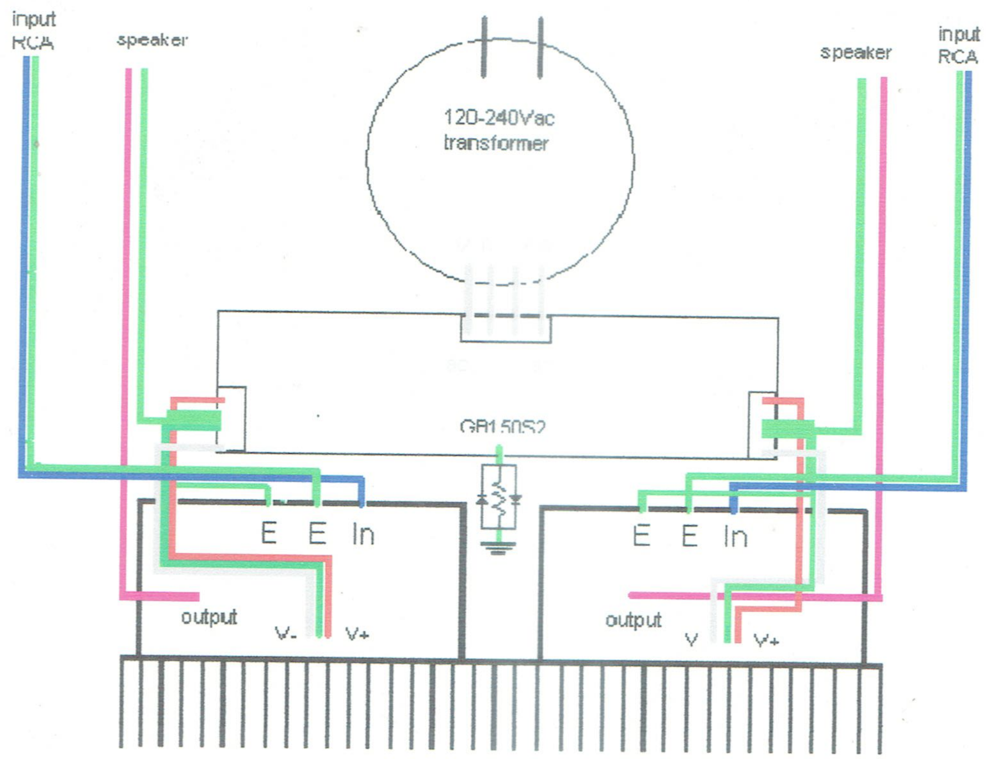
Also helpful to have is a few 1 ohm and up to 10 ohm 1W resistors to insert in various ground wires to 'break' possible hum loops.

Another, more exotic material for reducing stray fields is Mu-metal which is available in sheets about 30cm on a side.
This is a high permeability flexible metal alloy material that can be wrapped in sensitive areas to block EM fields, most effective at the weak pickup point rather than the source as the thin workable sheets saturate easily and become effectively transparent.

There is still a fair bit of experimentation to get any hum minimised but you will have the goodies. It can be frustrating, believe me, and no one solution is universal!

A simple Chassis/Layout

This is similar to the GB150D complete kit.



Testing

1. Once the module is completed and connected to the power supplies, set VR1 to just clockwise of centre, no speaker connected, the 10R 1W resistors across the fuse-holders (fuses out of holders) and a multimeter monitoring the voltage across them (20V DC range).

2. Check all connections before applying power. The power supply should have already been tested.

3. Switch on and check nothing is burning and there is less than 1V across each 10R test resistor and close to 0V between the output and E.

4. Slowly increase the trim-pot VR1 (rotate anti-clockwise) until 0.7 - 1V DC is across each 10R resistor and holding stable, backed off as heatsink temp rises and a thermal loop is established.

5. Check and null DC on the output using VR2 on a DC Volts 200mV range.

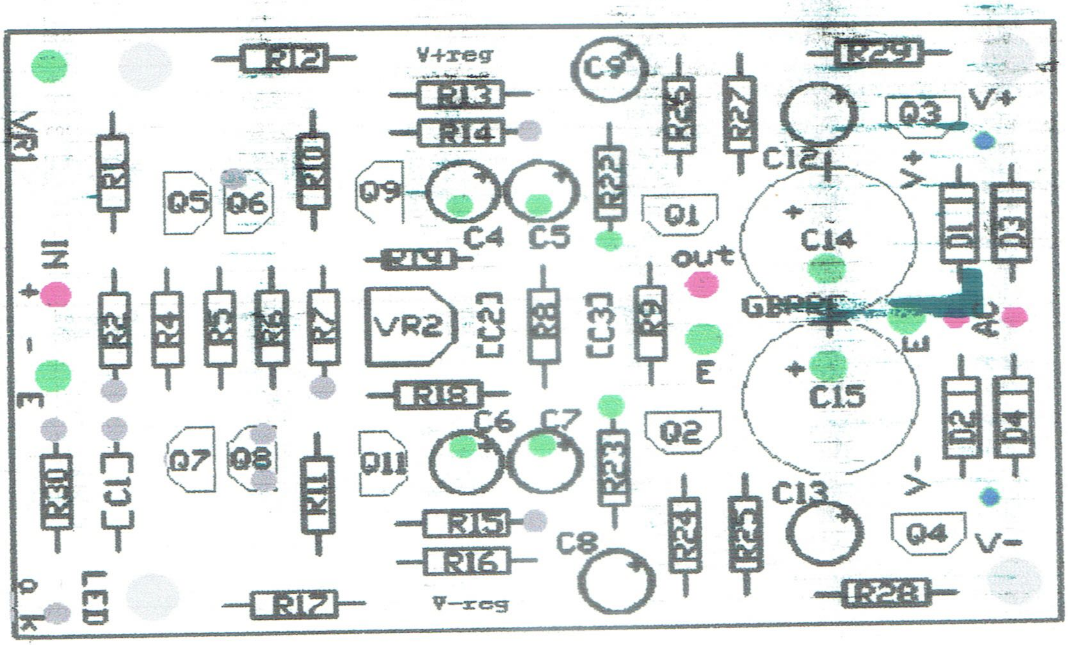
6. Connect your usual input and turn the volume control up with no signal. Then re-check the DC offset (200mV DC scale) on the output is zero. If more that 50mV of DC is present insert a C1. SWITCH OFF AND DISCHARGE.

7. Connect inputs, earths, and outputs according to the drawing below, and switch on. Play music at low-medium level and, after 5 to 10minutes, pause the music and reset bias to 1V. If 1V cannot be reached you may need to parallel a 47K across both R10,12 under the PCB. This is more likely with lower supply voltages.

8. Once the amplifier is biased up and functional, switch off and insert the fuses. It can run for a few days playing music at moderate level. The 10R resistors may be removed or neatly left across the fuses as an

Assembly and Connectivity for SKpre

Stuffing Guide



Green pads are showing ground plane connections FYI; grey are mtg holes

R1,19	100K	D1,2,3,4	1N4004
R2,13,16	1K	LED	3mm grn
R4,5,6,7,9	47		
R8	3K	Q1,3,11	BC546B
R10,11,24,26,30	27K	Q2,4,9	BC556B
R12,17	6K2	Q5,6	BC546C
R14,15	150	Q7,8	BC556C
R18	wire link std		
R22,23	1K2		
R25, 27, 28, 29	3k	VR1	10K vol taper
C1	100pF polystyrene	VR2	50K trimpot
C2	6 pF silver mica		
C3	not used std		
C4,5,6,7,8,9	10uF 50V electro		
C12,13	100uF 25V electro		
C14,15	2200uF 25V electro		

Assembly

1. Identify all resistors using a digital multimeter and insert as shown in the stuffing guide above. Clip and solder wires.

2. Insert the trimpot VR2 in the centre of board position and solder 3 pins under.

Testing

1. After assembly and wiring all components should be double checked for location and orientation. Check transistor positioning against the PCB outlines and check capacitor direction against the stuffing guide.

2. Connect the ac power and switch on. The LED should glow immediately.

3. Using the multimeter, measure firstly the +V (to E) and -V (to E) unregulated supplies on the outer end of the diodes. This should be close to 20V and not over 25V, the large capacitor rating.

4. Then measure the DC voltage at the end of R12 and R17 closest to the Vreg writing. This should be +15V and -15V (for +/-20V unregulated) or close to it.

5. Using a 200mV DC multimeter range, check the DC at the OUT wire (to E). This should be < 5mV. Turn VR2 to adjust this to zero. This setting will depend on having the volume control fitted or an external input connection. Re-check before connection to a power amplifier, at the output RCA, and trim to zero.

6. Connect to a signal source and power amplifier with the volume control turned down, check for minimal DC offset at the speaker output, then play some music through the module.

Alternative off-board Volume Control

If using an Alps blue DG 10K volume pot (or other) which has to be wired to both modules to control stereo volume with one control, the 3 holes at the front edge of the PCB marked VR1 are the connections. They are, from front left - GND, Wiper, Input. The Alps pot will have the same connections.

Simplest is to mount the Alps pot to the front panel and locate the Left and Right modules just behind and either side. Attach 3 colour coded wires to the holes and mount the modules in the chassis. Spin the Alps around so it's pins are facing up and solder the wires in reverse order to the 3 pins in each row, one row for Left and one row for Right module connections. Then turn the Alps body back around 180 degrees so the pins are facing down for neatness and tighten the locknut. A small hole can be drilled in the panel to take the locating pin, or it can be filed off.

If you have a piece of copper sheet for a shield, this should be ~ 75mm (3") by 50mm (2"). A cut halfway down each side and in 25mm (1") will allow it to be folded into a box to cover the top and back and double side thickness. Fold the box and solder a wire on. Fit it on the Alps and ground it to the case. This will provide some shielding and crosstalk reduction. This will have a greater effect

3. Identify the different types of small TO92 transistors and insert them with the correct orientation and locations and solder in position quickly. Be careful not to overheat them - by soldering quickly, one pin on each group then the second...

4. Insert diodes D1, 2, 3, and 4 with the correct polarity (line marked end).

5.Insert C1, then C4 through C15, taking care of polarity - THESE COULD EXPLODE if inserted the wrong way. There is no need to make solder connections to top side pads as the holes are 'through hole plated' and the pad on the bottom of the PCB is all that needs solder.

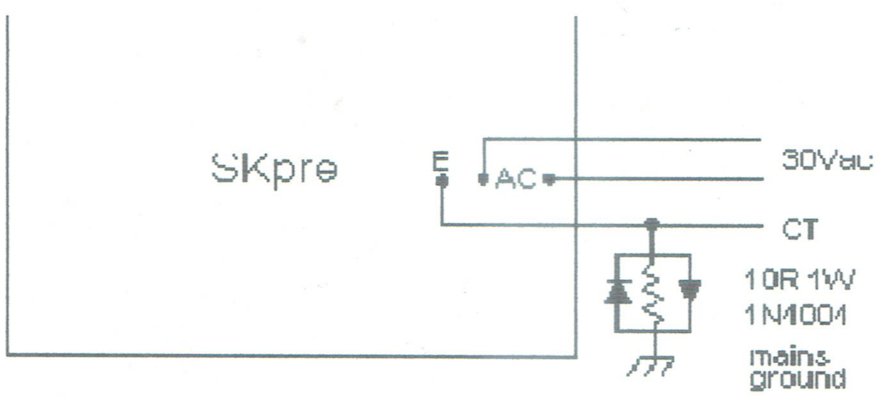
6. Insert the green LED with the longer lead in the hole marked "a". Long or short pins or flying leads for remote mounting can be used according to your needs. The circuit can power a blue LED instead if required.

7. Insert the 10K log volume pot in the 3 x 5mm spaced front edge holes labelled VR1 or wire to a remote dual gang pot or stepped attenuator. Connections are from left to right GROUND, wiper (volume level), input (full level).

Supplying Power

To power the single channel from either 120Vac or 240Vac, a variety of options are possible -

1. [BEST] A regular 24 - 30Vac CT or 2 x 12V to 2 x 15Vac power transformer of 5VA upwards. The transformer secondary power connections are made to the AC and E as below -



Make sure the transformer regulation is < 15% or use a lower voltage like 24Vac CT if it's greater than 15% regulation, as with smaller ones.

2. A 12V or 15Vac supply like a plugpack/wallwart can be used but with 6dB