Oakley Sound Systems

Eurorack Modular Series

Multimix

PCB Issue 1

Builder's Guide

V1.3

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the issue 1 Multimix Eurorack module from Oakley Sound. This document contains a basic introduction to the board set, a full parts list for the components needed to populate the boards and some hints on how to build the unit.

For general information regarding where to get parts and suggested part numbers please see my useful Parts Guide at the project webpage or http://www.oakleysound.com/parts.pdf.

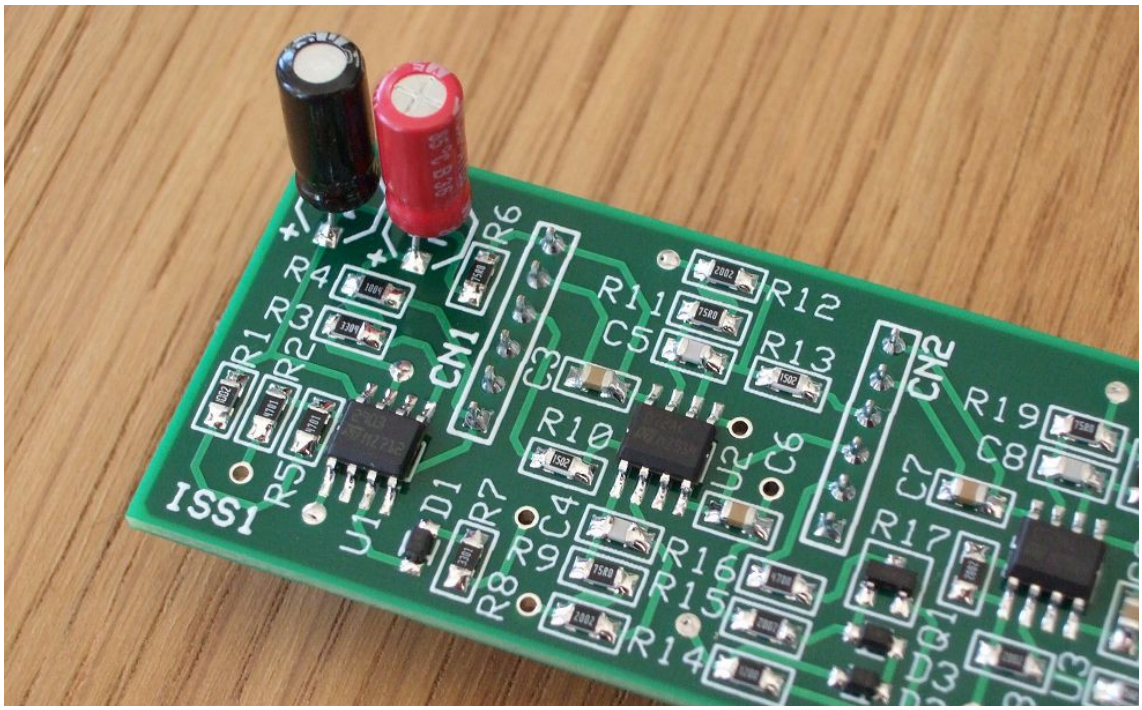
For general information on how to build my modules, including circuit board population, mounting front panel components and making up board interconnects please see my generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

The Multimix PCB set

The electronics of the Oakley Multimix is built on two printed circuit boards (PCBs). The pot board holds the three pots, three switches, LEDs, input and output sockets, and the audio input and output circuitry. All the front panel components are directly soldered to the pot board. The board is a double sided design meaning that there are electrically conductive copper tracks on the top and bottom surfaces of the board. The surface mount components are all soldered to the top side of the board while the pots and sockets are soldered to the underside which faces the inside surface of the front panel.

The top side of the pot board. The board is already fitted to the front panel. Note that all the surface mount components are fitted to this side. The pots and sockets cannot be seen as they are fitted to the underside of the board.

The second board is called the main board. This holds the output driver circuitry, the level monitoring circuit, the precision 5V reference, the power supply conditioning, and the power inlet. The main board is a four layer design. This means that the board has layers of copper on top and bottom sides, as well as two internal copper layers. The top internal layer is solely reserved for the 0V connections – sometimes called module ground – used for the main and top board's electronics.

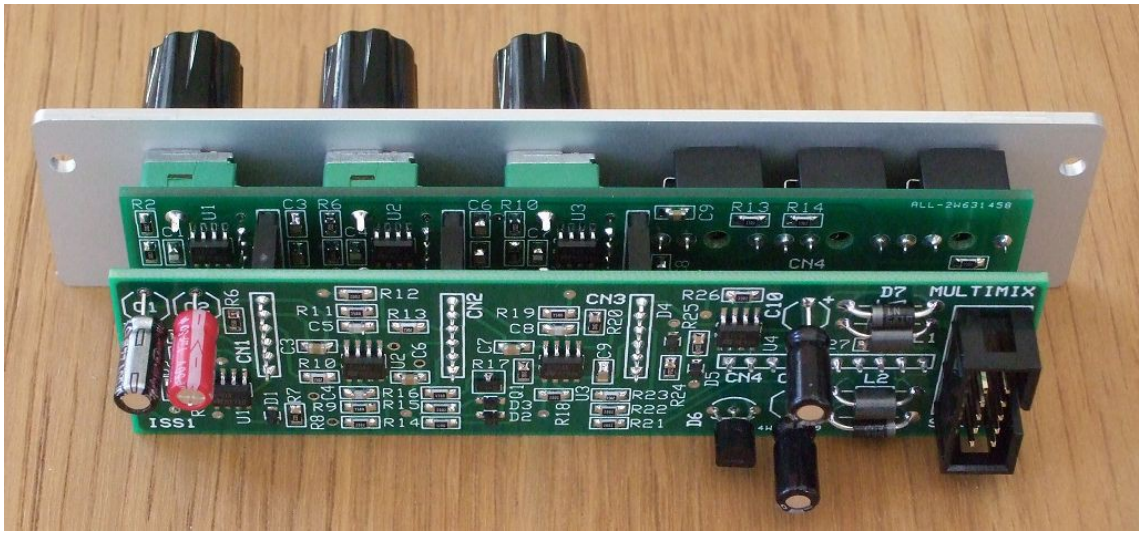


A close up of the main board showing the level detector and first two output channels.

The components, again a mixture of through hole and surface mount devices, are soldered to the top of the board. The main board's components are all accessible from the rear of the module. Although this means that you need to be a little careful when handling the module, it does mean that fault finding is much easier than it would be if the main board was facing the other way and none of the components would be visible.

The surface mount components are of relatively large geometries to make the build suitable even for beginners in surface mount soldering. The resistors and capacitors are all 0805 and the ICs are narrow body SOIC. The larger capacitors, ferrite beads, power diodes, reference diode, and all the interconnects are through hole parts.

Both circuit boards are the same size, that is, 29 mm (wide) x 107 mm (high). The boards are designed to go behind a panel that is 6HP wide.



The module comprises of two circuit boards connected together with three six way and one eight way 0.1" SIL headers and sockets.

The grip of the multiple SIL interconnects is strong enough to hold the boards together for most uses. Should you require more rigidity then a **small** blob of silicone sealant or heat melt glue across the join between the plastic housings of CN1, CN2 and CN3 would hold everything together tightly. However, it is best not to do this until your unit has been thoroughly tested.

The design requires plus and minus 12V supplies, although it will work on +/-15V too with no modifications. The power supply should be adequately regulated. The current consumption is around +31mA and -26mA at +/-12V.

Multimix Parts Lists

The components are grouped into values, the order of the component names is of no particular consequence.

Many of the parts for this circuit board are surface mount devices but not all of them. Take special care when ordering your parts that you order the correct type of part. This parts list shows the type of part needed whereas the circuit diagram does not.

A quick note on European part descriptions. R is shorthand for ohm. K is shorthand for kilo-ohm. R is shorthand for ohm. So 22R is 22 ohm, 1K5 is 1,500 ohms or 1.5 kilohms. For capacitors: 1uF = one microfarad = 1000nF = one thousand nanofarad. For electrolytic capacitors the maximum working voltage is normally given with the value, eg. 1uF/63V is a one microfarad capacitor with a working DC voltage of 63V.

To prevent loss of the small '.' as the decimal point, a convention of inserting the unit in its place is used. eg. 4R7 is a 4.7 ohm, 4K7 is a 4700 ohm resistor, 6n8 is a 6.8 nF capacitor.

Main Board issue 1 Parts List

Resistors

All resistors are surface mount, size 0805 (or metric 2012) 1% 125mW metal film.

75R	R6, R9, R11, R19
120R	R14
470R	R16
3K3	R7
4K7	R2, R5
6K8	R27
10K	R1
15K	R10, R13, R23
20K	R20, R12, R22, R8, R21, R15, R17, R18
33K	R26
47K	R24, R25
1M	R4
3M3	R3

Capacitors

The following capacitors are surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric C0G (or NP0), working voltage 50V, tolerance +/-5%.

33pF	C4, C5, C8

The following capacitors are surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric X7R, working voltage 50V, tolerance +/-5%.

100nF	C3, C6, C7, C9

The following capacitors are standard through hole electrolytic capacitors with 0.1" (2.5mm) or 0.2" (5mm) radial leads.

1uF, 63V electrolytic	C2
2u2, 63V electrolytic	C10, C11
22uF, 35V electrolytic	C1

Discrete Semiconductors

The following devices are surface mount parts.

1N4148WS signal diode	D1, D2, D3, D4, D5
BC860 PNP transistor	Q1

The following devices are standard through hole parts.

1N5819 Schottky diode	D7, D8

Integrated Circuits

The following parts are all surface mount devices. All are small outline (SOIC) narrow body packages.

LM2903	U1
TL072ACD dual op-amp	U2, U3, U4

The following device is a standard through hole part.

LM4040-5.0V reference	D6

Miscellaneous

Axial leaded ferrite bead	L1, L2
6-way SIL socket	CN1, CN2, CN3
10-way SIL socket	CN4
2 x 5 0.1" boxed header	STRIPE

CN1 to CN4 are mounted on the underside of the board, facing down, and soldered from the top. Special care must to be taken to ensure that these connectors are mounted perpendicular to the board surface.

Pot Board issue 1 Parts List

Resistors

All resistors are surface mount, size 0805 (or metric 2012) 1% 125mW metal film.

13K	R13, R14, R15
20K	R2, R3, R4, R6, R7, R8, R10, R11, R12
100K	R1, R5, R9

Capacitors

The following capacitors are surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric C0G (or NP0), working voltage 50V, tolerance +/-5%.

100pF	C1, C4, C7

The following capacitors are surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric X7R, working voltage 50V, tolerance +/-5%.

100nF	C2, C3, C5, C6, C8, C9

Discrete Semiconductors

The following devices are standard through hole parts.

3mm green LED	D2
3mm red LED	D1

See later in this document for details regarding the fitting of the LEDs.

Integrated Circuits

The following parts are surface mount devices. All are small outline (SOIC) narrow body packages.

TL072ACD dual op-amp	U1, U2, U3

Potentiometers

All pots are Alpha 9mm vertical pots with 6.35mm round shafts.

50K linear	LEVEL1, LEVEL2, LEVEL3

The pots are to be fitted to the underside of the board and their pins soldered from the topside.

Miscellaneous

SPDT 'on-on' toggle switch	SW1, SW2, SW3
6 way 0.1” SIL header	CN1, CN2, CN3
10 way 0.1” SIL header	CN4

Special care must to be taken to ensure that connectors CN1 to CN4 are mounted perpendicular to the board surface.

Thonkiconn 3.5mm socket	IN1, OP1, IN2, OP2, IN3, OP3

The six 3.5mm sockets are to be fitted to the underside of the board and their pins soldered from the topside.

3 knobs to suit. Davies 1900H or clones thereof are to be recommended.

Part Sourcing

For general information regarding where to get parts and suggested part numbers for Oakley projects please see my Parts Guide at the project webpage or www.oakleysound.com/parts.pdf.

The front panel is obtained either from Schaeffer in Germany, or Front Panel Express (FPE) in the US. The database for the panel is provided on the project webpage and this file can be opened, edited and ordered using the Frontplatten Designer program available free from Schaeffer or FPE. The cost of the 6HP wide panel was around 27 Euros at the time of writing. The panel is 2.5mm thick and has a natural silvery finish. The black text is printed, although it is possible to edit the database to have the text engraved.

The three pots are Alpha 9mm vertical pots as sold by Thonk and others. I use Thonk's standard ones with 6.35mm (0.25”) round shafts. Other shaft types are available, like splined or D-type, and you should purchase those that are compatible with your choice of control knob. I prefer the round shafts as they require control knobs that have a fixing screw. Although these knobs take more time to place and secure, the height at which the knob sits on the pot shaft is under your control. Push fit knobs can sometimes not sit at the right height which is unsightly if they are too high, or graunch against the pot's securing nut and washer if they are too low.

For control knobs I use Davies 1900H clones available from Thonk and others.

The three toggle switches are a standard sub-miniature type that are designed to fit directly into a PCB. These on-on switches are available from Thonk as their DW1 sub-miniature switch. They are made by Dailywell and offer good performance for their price.

The six 3.5mm sockets are the same type and are available from Thonk known either by their Thonkiconn moniker or as part number PJ398SM.

The fixed 2.54mm (0.1”) interconnects are in two parts, the male header, and the female socket or receptacle. I use three six way and one ten way single in line (SIL) connectors in the Multimix. The ones I use are made by Multicomp, although more expensive types are available which may offer increased longevity. These are the Multicomp part numbers for the parts I used:

10 way socket	2212S-10SG-85
10 way header	2211S-10G
6 way socket	2212S-06SG-85
6 way header	2211S-06G

All the dual op-amps are recommended to be TL072ACD. Note the A in the part number suffix, this is the improved specification version of the standard TL072 which has lower offset voltages. Offset voltage is an unwanted voltage on the inputs of an op-amp which then manifests itself as an error voltage on the output of the op-amp. Other dual op-amps may be used such as the LF412CD and TLE2072CD.

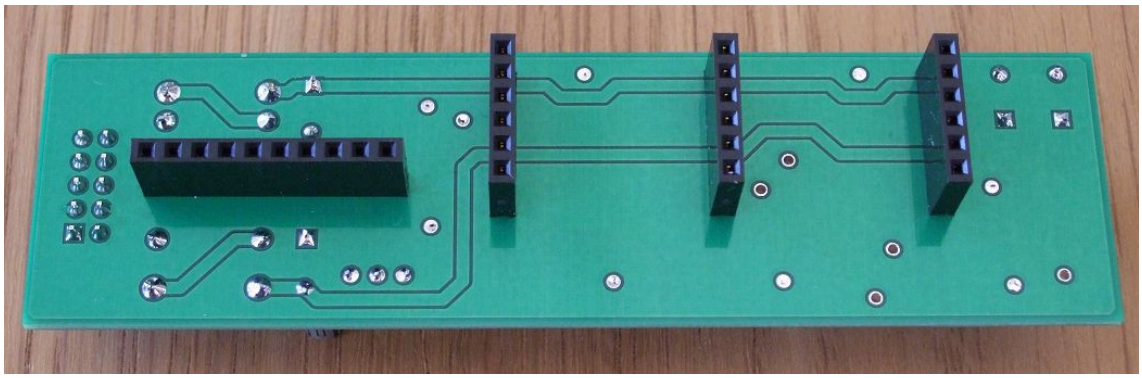
Populating the Circuit Boards

For general information on how to build Oakley modules, including circuit board population, mounting front panel components and making up board interconnects please see my generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

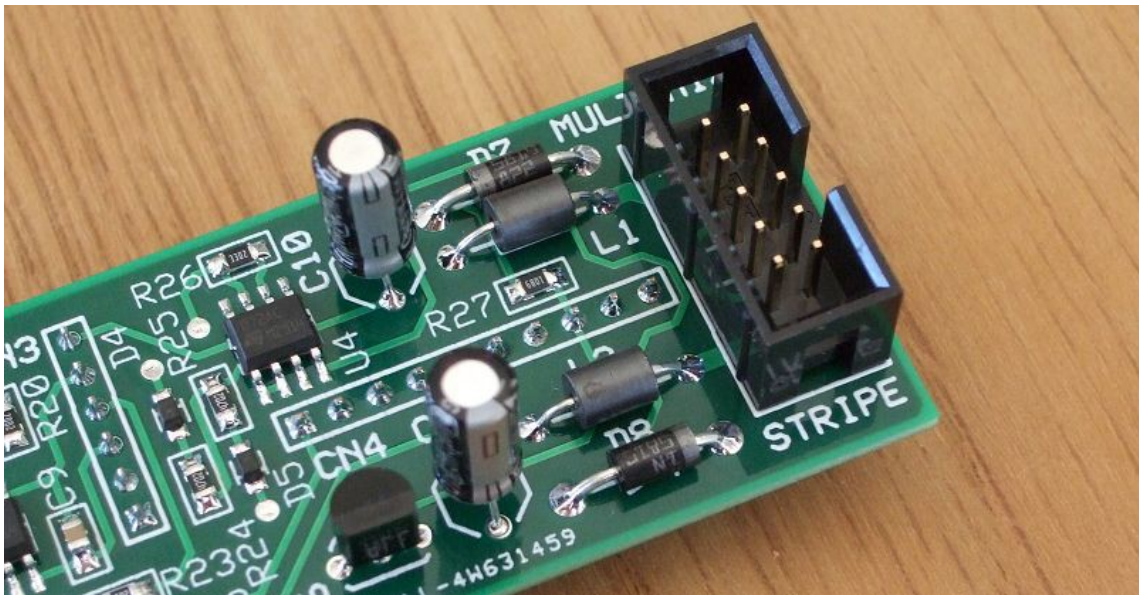
Main Board Construction

All the surface mount components should be soldered first. Take care to treat all ICs as static sensitive devices. I usually solder the resistors first, then the capacitors, and then the ICs.

The next items to be soldered are the single in line (SIL) sockets. These are to be fitted to the underside of the board and soldered from the top side.



Special care should be taken to ensure that each SIL socket sits at right angles to the board surface. One way to do this is to temporarily fit the SIL headers into the SIL sockets and fit the Main board and Pot board together with the headers and sockets in between. Then solder the headers to the main board. Pulling the two boards apart should reveal all the sockets neatly in place and at the correct angle.



The next items to be soldered will be the through hole components. Remember that diodes and electrolytic capacitors are polarised so they need to be fitted the right way around. You also should

especially make sure that the boxed header is correctly orientated. Pin 1 is normally designated with a little arrow shape on the plastic housing of the header and this should align with the square pad on the board. Also, the hole in the housing should correspond to the little box shape on the board's printed legend.

Pot Board Construction

Except for the pots, switches, LEDs and sockets all the parts are to be fitted to the topside of the board. The surface mount parts should again be soldered first.

Now fit the SIL headers into place and solder from the underside of the board. It may be worth temporarily fitting the Pot board to the Main Board so as to hold the headers exactly at right angles while you solder.

The remaining parts are the front panel components and these will be fitted to the underside of the board and soldered from the top of the board. There are no legends on the underside of the board to indicate where the parts will go but it should be clear where they need to be fitted. To ensure the correct alignment of these parts before soldering you should have your front panel ready. The panel will be used as a jig to hold the parts in the correct place while you solder. Failure to use the panel as a jig for these components could mean that the panel will not slide onto the components.

The first parts to place, but not solder, will be the pots. The pots should be fitted so that the three pot pins go into the board first, then ease in the two lugs into their holes and push in firmly so that the pot clicks into place. The pot will sit securely in the board even without soldering.

Now fit the six sockets and three switches into their places on the board but again do not solder yet. Place one of the toothed washers that came with the switches onto each switch.

LEDs do not light if they are not fitted the correct way around so it is imperative that you do this bit right. The cathode of the LED goes into the square pad. Check with the LED's datasheet to see which lead is the cathode if you are not sure. Once again we are fitting the devices into place but are not soldering them just yet.

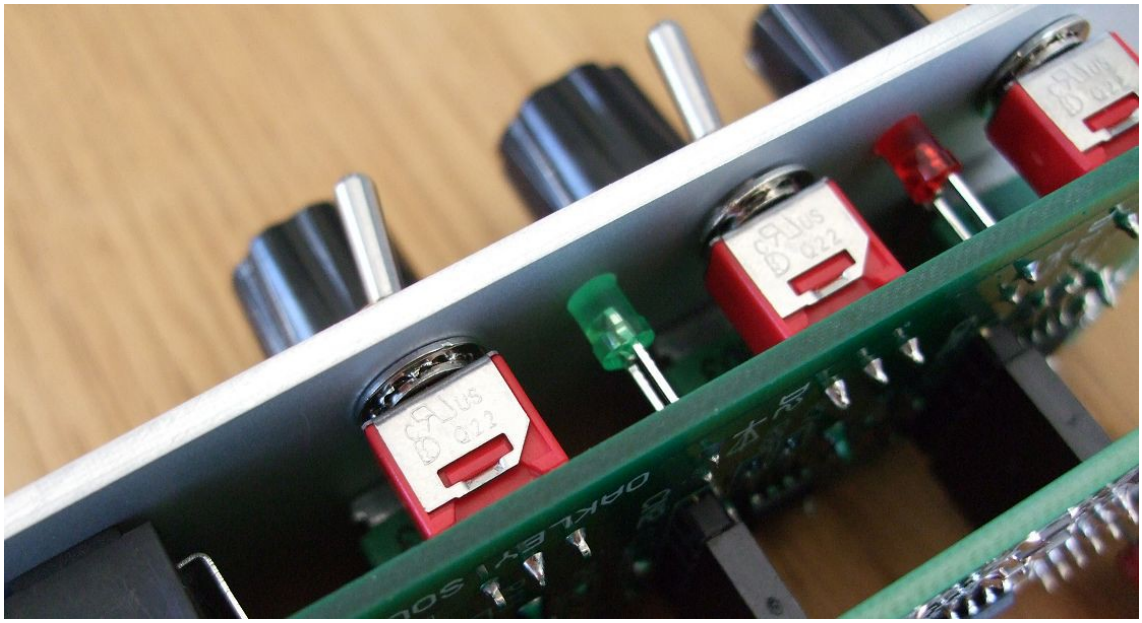
Now ease the panel down onto the pots, switches and sockets making sure that all their threads are sitting snugly in their holes. You can ignore the LEDs as they'll just sit below the panel for the time being. Place a washer and a nut on each of the pots and sockets. Tighten the nuts but not too tightly. Turn the module over to reveal the topside of the circuit board. Now solder all the pots and sockets, but not the switches or LEDs yet.

Flip the module over again and fit a single nut, and if you have one, a flat dress washer, to the exposed thread of each switch. You can ignore the other nut that came with the switch, as well as

any washers that have locating lugs. Tighten each nut gently taking care not to scratch the panel. The switches will be pulled towards the panel and slightly off the board's top surface when you tighten the nuts. The module can be turned over and the switches soldered.

Now tease each LED through its panel hole by using its leads that are sticking out the bottom of the board.

If you are using standard dome topped LEDs then simply push them into the hole as far as they will go and solder each one in turn. If you are using flat topped LEDs then you may wish to have them align with the top surface of the panel. This is not a trivial task but one way is to use a bit of sticky tape over the holes and let the LEDs push up against that before soldering each one in turn.



This one has been built with flat topped cylindrical LEDs that sit flush with the panel.

Testing

It is prudent to test the main board on its own before fitting it to the pot board and panel. If you have a bench power supply that allows you to select the current limit then set both the +12V and -12V supplies to a current limit of 50mA. This should prevent any serious meltdowns if there is a problem with the build.

Power up the main board on its own. If you can measure power supply current then it should be around -15mA from the negative rail and +20mA from the positive rail. Anything significantly more than this, like 50mA, will indicate a problem. If you can't measure current then check that no devices are getting warm.

It may be worthwhile checking the output voltage of each op-amp on the board, that is, U2, U3, and U4. Measure the voltages at pin 1 and at pin 7 with respect to 0V (module ground). 0V may be most easily available from the square solder pad (pin 1) of CN3. Each op-amp output should have a voltage of no more than +/-10mV, that is, very close to 0V. If any of them are not this, like +11V, then that particular circuit has a problem.

Also check that the reference diode is working correctly. Check the voltage of the pad nearest the power connector of R27. This should be 5.0V.

If all is well, then the main board can be powered down. Wait a minute and then attach the main board to the pot board. Be very careful to ensure all the SIL connections are correctly in place.

Power the module up again and, if you can, check the current draw of the module. It should be around +30mA and -25mA. Anything significantly different to this, say over 50mA, will indicate a problem.

Turn all the pots down and put the switches in their upward (ie. non inverting mode) positions. Turn up Level 1 and you should notice that the green LED will come on when the pot gets close to its maximum. Flick channel 1's switch down to turn on the invert mode and the green LED should remain lit. Repeat this for the other two channels – the green LED should light shortly before each level pot gets to its maximum position in either switch position.

Now turn up channel 1 with the invert switch in upward position. Increase Level 2, also with its switch in the upward position. As you reach around 80% of the rotation of Level 2 the red LED should come on too. Turn the Level 2 pot to its maximum and click channel 1's switch to invert. Both LEDs should now go out, the red one first and then the green.

If all these things happen you almost certainly have a working module.

Final Comments

If you have any problems with building the module, an excellent source of support is the Oakley Sound Forum at Muffwiggler.com. I am on this group, as well as many other users and builders of Oakley modules.

If you can't get your project to work and you are in the EU, then Oakley Sound Systems are able to offer a 'get you working' service. If you wish to take up this service please e-mail me, Tony Allgood, at my contact e-mail address found on the website. I can service either fully populated PCBs or whole modules. You will be charged for all postage costs, any parts used and my time at 25GBP per hour. Most faults can be found and fixed within one hour, and I normally return modules within a week. The minimum charge is 25GBP plus return postage costs.

If you have a comment about this builder's guide, or have a found a mistake in it, then please do let me know. But please do not contact me directly with questions about sourcing components or general fault finding. Honestly, I would love to help but I do not have the time to help everyone individually by e-mail.

Last but not least, can I say a big thank you to all of you who helped and inspired me. Thanks especially to all the great people on the Synth-diy and Analogue Heaven mailing lists and those at Muffwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK