Oakley Sound Systems

Eurorack Modular Series

Envelope

PCB Issue 1

Builder's Guide

V1.3

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

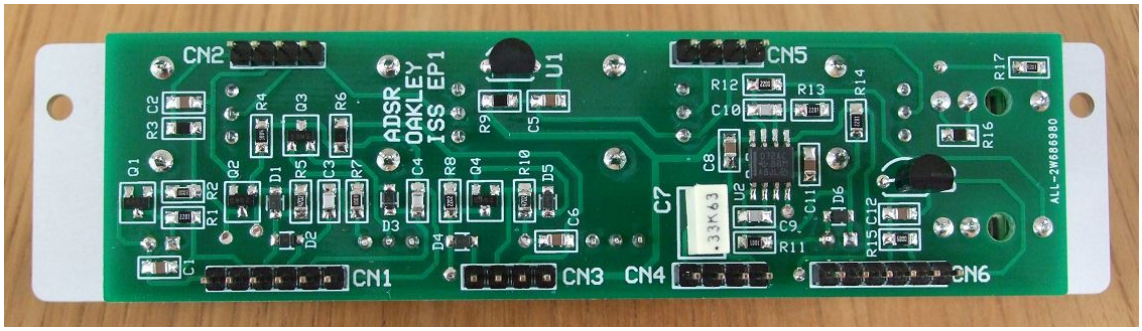
This is the Project Builder's Guide for the issue 1 Envelope Eurorack module from Oakley Sound. This document contains a basic introduction to the board set, a full parts list for the components needed to populate the boards, some basic help on how to purchase parts and to build the project, and a simple test procedure.

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

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

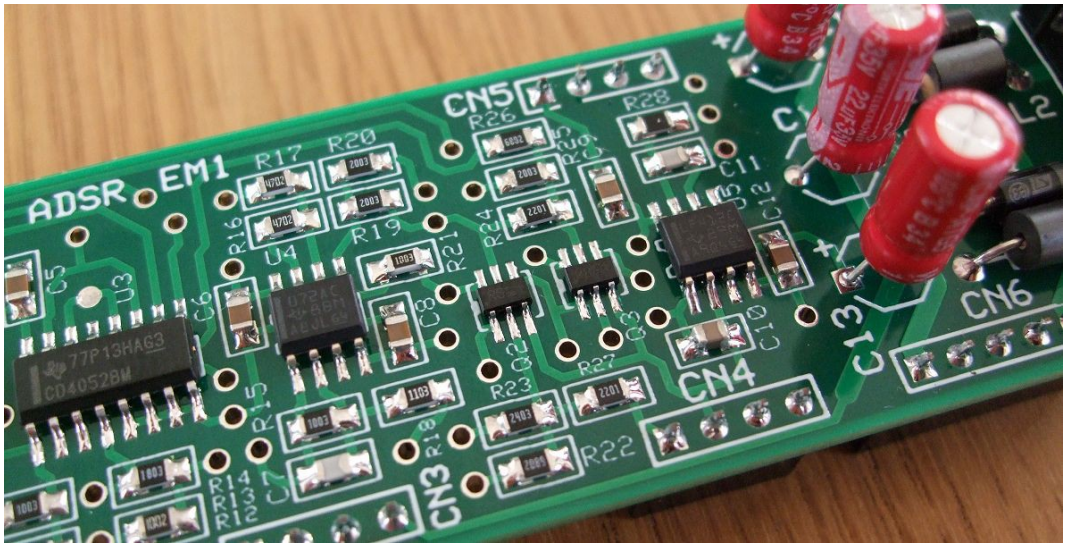
The Envelope PCB Set

The electronics of the Oakley ADSR Envelope is built on two printed circuit boards (PCBs). The Pot board holds the four pots, two switches, two LEDs, input and output sockets, the gate processing circuitry, the two voltage references, the LED drivers and the output buffer. All the front panel components are directly soldered to the pot board. The pot 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, switches 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 ADSR's slew generator core, the control voltage switching, the loop and attack peak comparators, 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.

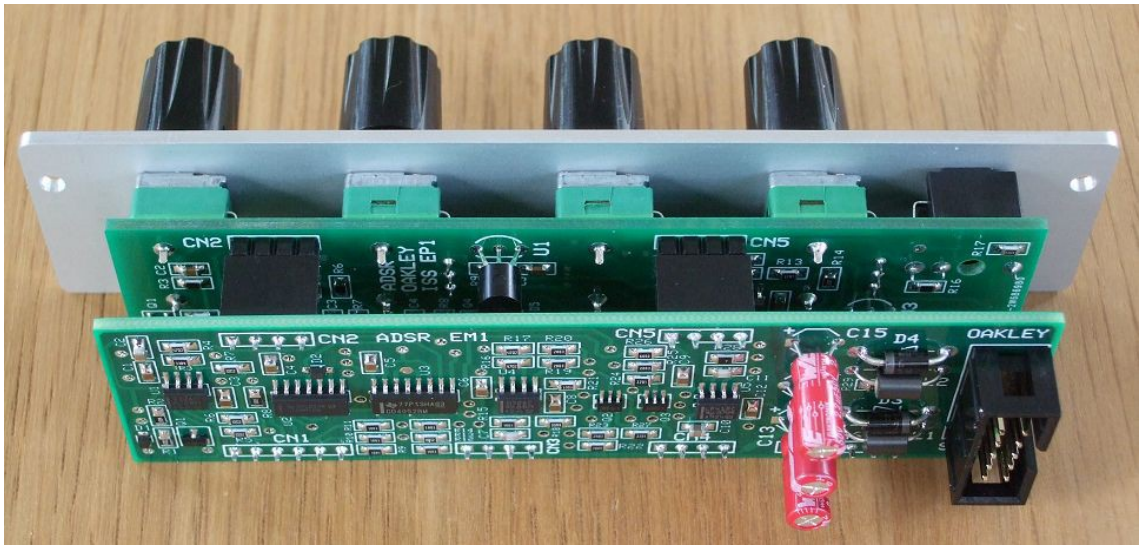


A close up of the main board showing the ADSR's core.

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 mostly 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 dual transistor arrays, Q2 and Q3, are housed in a very small SOT457 package but can soldered easily enough. The larger capacitors, ferrite beads, power diodes, 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 two 6 way and four 4 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, CN5 and CN6 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. The power supply should be adequately regulated. The current consumption is slightly below +35mA and -25mA at +/-12V.

Envelope 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 th 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.

22R	R29
2K2	R24, R27
4K7	R28
10K	R5, R6, R7, R13
47K	R1, R4, R16, R17
68K	R22, R26
100K	R8, R10, R11, R12, R15, R21
110K	R18
180K	R14
200K	R19, R20, R25
240K	R23
1M	R9
3M3	R2, 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	C11
100pF	C7
47nF	C10

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

100nF/50V	C1, C3, C4, C5, C6, C8, C9, C12
1uF/25V	C2

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

2u2/63V electrolytic	C13, C15
22uF/35V electrolytic	C14

Discrete Semiconductors

The following devices are surface mount parts.

1N4148WS signal diode	D1, D2
BC850 NPN transistor	Q1
BCM847DS dual NPN transistor	Q2
BCM857DS dual PNP transistor	Q3

The following devices are standard through hole parts.

1N5819 Schottky diode	D3, D4

Integrated Circuits

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

CD4001BM quad NOR gate		U2
CD4052BM dual 1 to 4 switch		U3
LF412CDR	dual FET op-amp	U5
TL072ACD	dual FET op-amp	U1, U4

Miscellaneous

Axial ferrite bead	L1, L2
2 x 5 0.1" boxed header	STRIPE
4-way SIL 0.1" socket	CN2, CN3, CN4, CN5
6-way SIL 0.1" socket	CN1, CN6

Parts List for ADSR Pot Board issue 1

Resistors

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

220R	R12
680R	R15
1K	R11
2K2	R1, R13, R14
4K7	R3, R6, R9, R16
22K	R8
47K	R5, R10
100K	R2, R7, R17
1M	R4

Capacitors

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

100pF	C9, C10
1nF	C3, C4

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

100nF	C1, C2, C5, C6, C8, C11, C12

The following capacitor is a through hole component.

330nF, 63V polyester	C7

Discrete Semiconductors

The following devices are surface mount parts.

1N4148WS signal diode	D1, D4, D5, D6
BAT42WS Schottky diode	D2, D3

BC850 NPN transistor	Q1, Q2, Q3, Q4

The following devices are standard through hole parts.

3mm green LED	GATE
3mm red LED	OUTPUT

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	U2

The following devices are standard through hole parts.

LM4040-5 5V reference	U1
LM4040-10 10V reference	U3

Potentiometers

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

50K linear	ATTACK, DECAY, SUSTAIN, RELEASE

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

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

Miscellaneous

SPDT on-off-on switch	Mode
SPDT on-on switch	Range
3.5mm socket	Gate, Env Out

The two 3.5mm sockets and two switches are to be fitted to the underside of the board and their pins soldered from the topside. See later for more details.

4-way SIL 0.1” header	CN2, CN3, CN4, CN5
6-way SIL 0.1” header	CN1, CN6

Special care must to be taken to ensure that connectors CN1 to CN6 are mounted perpendicular to the board surface.

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 direct from 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 four 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 toggle switches are a standard sub-miniature type that is designed to fit directly into a PCB. The on-on switches are available from Thonk as their DW1 sub-miniature switch, and the on-off-on are available from Thonk as their DW2 switch. They are made by Dailywell and offer good performance for their price.

The two 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 four way and one six way single in line (SIL) connectors in this project. 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:

6 way socket	2212S-06SG-85
6 way header	2211S-06G
4 way socket	2212S-04SG-85
4 way header	2211S-04G

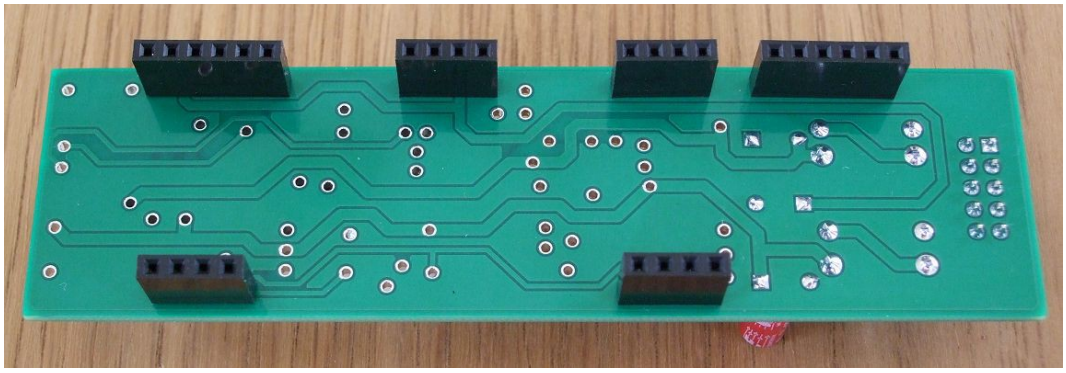
Populating the Circuit Boards

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.

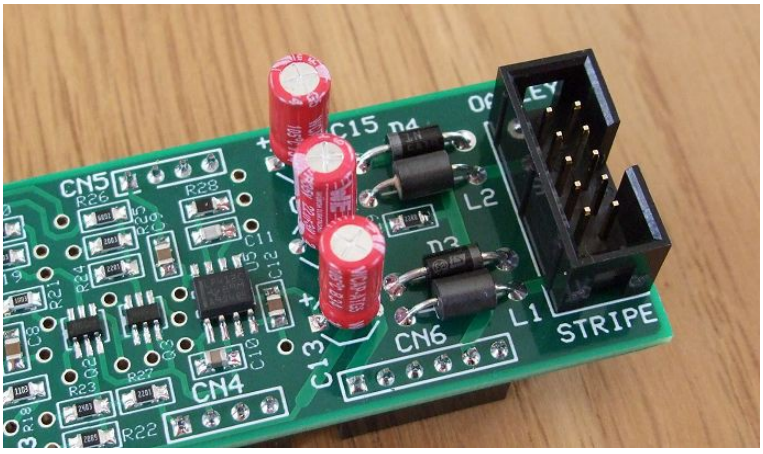
Main Board Construction

All the surface mount components should be soldered first. Take care to treat all transistors, diodes and ICs as static sensitive devices, especially the two 4000 series ICs which are particularly vulnerable. I usually solder the resistors first, then the capacitors, then the discrete semiconductors, 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.



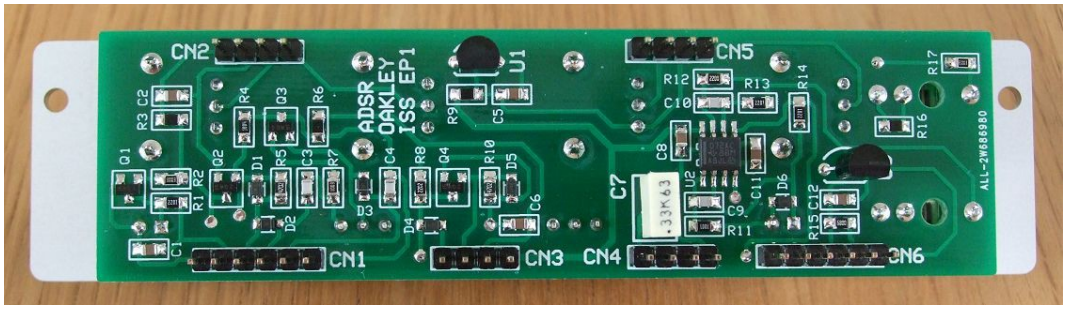
The next items to be soldered will be the through hole components. Remember that diodes and electrolytic capacitors are polarised so they need to be fitted the right way around. You also

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 two sockets and two switches into their places on the board but again do not solder yet. The switches are two different types: Fit the two position on-on switch in the lower of the two locations, and fit the three position on-off-on switch in the upper position. Place a toothed washer that came with the switches onto each switch.

Fit the LEDs loosely into their holes. Remember that LEDs are polarised and will not light up if they are fitted incorrectly. The anode of each LED should be fitted through the square solder pad on the PCB. Once again we are fitting each device into place but are not soldering the leads just yet. The green LED is placed into the top pair of holes (GATE) and the red one into the bottom pair of holes (OUTPUT).

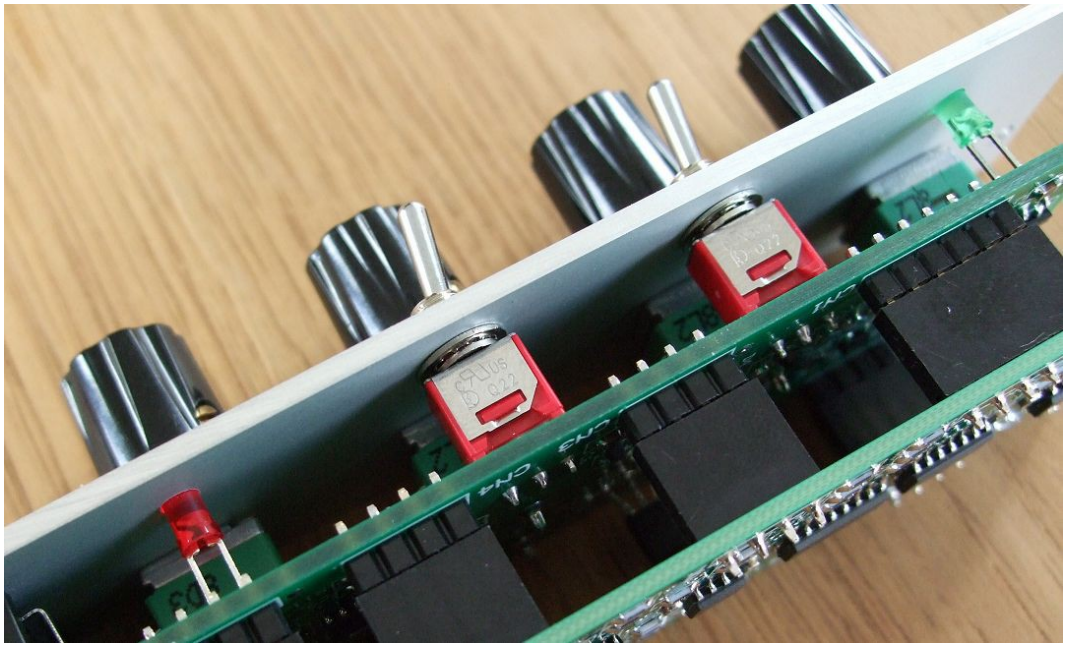
Now ease the panel down onto the pots, switches and sockets making sure that all their threads are sitting snugly in their holes. You can ignore the LEDs as they will just sit below

the panel for the time being. Place a washer and a nut on each of the pots and sockets, but not the switches. 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 LED 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. Gently tighten the nuts 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' leads 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 a standard dome topped LED then simply push it into the hole as far as it will go and solder the leads. 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 their leads.



These are flat topped LEDs. You can see that their base sits significantly below the inside panel surface to allow the flat top to be flush with the front panel's top surface.

Initial 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 -16mA from the negative rail and +16mA from the positive rail. Anything significantly more than this, like 30mA, will indicate a problem. If you can't measure current then check that no devices are getting warm.

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.

Set the three way mode switch to Norm and power up the module and, if you can, check the current draw of the module. It should be around +35mA and -25mA. Anything significantly different to this, say over 50mA, will indicate a problem.

Set the three way mode switch to Loop, the two way range switch to SLOW and all the pots to their minimum. Both LEDs should now be lit with the green Gate LED a little brighter. Advance the Decay time and you soon start to notice the Red LED flashing, the frequency of the flashes will decrease as you further increase the decay time.

Check that the flashing stops if you turn up the Sustain control past about 20% of full scale. As you further increase the sustain level the brightness of the red LED should increase.

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