Oakley Sound Systems

5U Oakley Modular Series

ADSR/VCA
Looping Envelope Generator

PCB Issue 5

Builder's Guide

V5.5

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

The suggested 1U wide front panel in our traditional MOTM compatible format.

Introduction

This is the Project Builder's Guide for the issue 5 of the ADSR/VCA 5U module from Oakley Sound.

This document contains a basic introduction to the board, a full parts list for the components needed to populate the boards, and a list of the various interconnections.

Issue 5 represents a continued evolution of the voltage controlled design which started in 2004. Issue 5 removes the THAT2181LC which had been used as the control element and replaces it with the same four transistor design as used in the Oakley VRG.

For the User Manual, which contains an overview of the operation of the unit and all the calibration procedures, please visit the main project webpage at:

http://www.oakleysound.com/adsr.htm

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, mounting front panel components and making up board interconnects please see our generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

Nomenclature

ADSR/VCA PCB: The original Oakley Sound System's envelope generator design that used passive components as timing controls. The PCB was superseded by the issue 1 VC-ADSR PCB in January 2004.

VC-ADSR PCB: The PCB, issue 5 of which is described in this document, used in the Oakley ADSR/VCA module since January 2004.

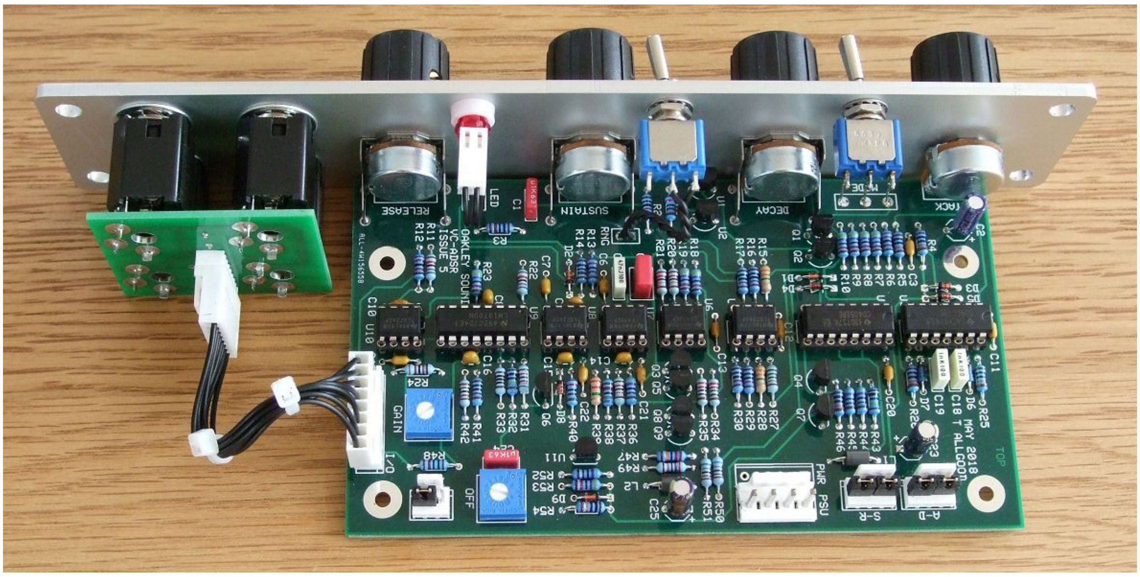
ADSR Upgrade PCB: The PCB that allows the VC-ADSR PCB to be controlled with external control voltages.

ADSR/VCA module: The module that, since January 2004, comprises of the VC-ADSR PCB and, since August 2010, acts as a looping ADSR with a built in VCA on the output.

VC-ADSR module: The 2U fully voltage controllable looping ADSR module with integral VCA.

ADSR Upgrade module: A 1U module that connects to an already constructed ADSR/VCA module to allow for full voltage control of the attack, decay, sustain and release parameters.

The VC-ADSR/VCA PCB



The issue 5 VC-ADSR/VCA board mounted behind a natural finish Schaeffer panel. Note the use of the optional Sock4 board to mount the jack sockets. Note also the two 0.1" KK headers on the lower right. These headers can be attached to the 1U Upgrade module to allow for full voltage control of the modules four parameters.

I have provided space for the four main control pots on the PCB. If you use the specified 16mm Alpha pots and matching brackets, the PCB can be held firmly to the panel without any additional mounting procedures. The pot spacing is 1.625" and is the same as the vertical spacing on the MOTM modular synthesiser.

The design requires plus and minus 15V supplies. The power supply should be adequately regulated. The current consumption is approximately 35mA. Power is routed onto the PCB by a four way 0.156" MTA156 type connector or the special five way Synthesizers.com MTA100 header. You could, of course, wire up the board by soldering on wires directly. The four pins of the four way connector are +15V, module ground (0V), earth/panel ground, -15V. The earth/panel connection allows you to connect the metal front panel to the power supply's ground without it sharing the modules' ground line.

The PCB has four mounting holes for M3 bolts, one near each corner. These are not required if you are using the three 16mm pot brackets. The board size is 143mm (high) x 81mm (deep).

The board has been laid out to accept connection to our Sock4 socket board. This small board speeds up the wiring of the four sockets and reduces the chances of mistakes.

Issue 5 ADSR/VCA Parts List

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.

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

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.

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.

Resistors

All resistors should be 1% 0.25W metal film resistors. The 1M5 and 3M3 values can be 5% metal film or carbon if you can't get 1% devices.

100R	R33
1K	R14, R3, R48
2K2	R13, R40, R1, R35, R34
4K7	R36, R45, R52, R44
6K8	R38
10K	R17, R2, R29, R30, R41, R16
11K	R42
12K	R31, R24
22K	R12, R6
36K	R53
47K	R23, R18, R9, R10, R19, R7, R8
62K	R11, R22
68K	R50, R51
100K	R46, R26, R4, R25, R54, R5
120K	R21, R20, R37
180K	R27
220K	R47
270K	R49
1M	R43, R32
1M5	R39
3M3	R15, R28

The value of R13 sets the maximum brightness of the LED and more efficient LEDs like blue ones will require the value of R13 to be increased to prevent the LED being overly bright. It is not recommended that you take the value of R13 below 1K5 as the increased current may have a detrimental effect on the module's power supply.

Capacitors

100nF multi layer axial ceramic	C3, C17, C11, C20, C22, C6, C21, C13, C12, C7, C10
33pF C0G 2.5mm ceramic	C14, C15
100pF C0G 2.5mm ceramic	C16, C9
470pF C0G 2.5mm ceramic	C8
1nF, 100V polyester	C18, C19
47nF, 63V polyester	C5
100nF, 63V polyester	C1, C24
330nF, 63V polyester	C4
2u2, 50V electrolytic	C25, C2, C23

Discrete Semiconductors

1N4148 signal diode	D3, D9, D5, D1, D8, D2,D4
BAT42 Schottky diode	D6, D7
BC549C NPN transistor	Q4, Q2, Q7, Q1, Q8, Q9
BC559C PNP transistor	Q6, Q5, Q3
5mm green* LED	LED

Green is the suggested colour but you can chose any colour you like.

Integrated Circuits

LM4040-5.0 5V reference	U1
LM4040DIZ-10 10V reference	U2, U11
LM13700 dual OTA	U9
CD4001BE quad NOR gate	U3
CD4052BE dual 4:1 switch	U4
TL072CP dual op-amp	U5, U8, U10
LF412CP dual op-amp	U6, U7

The 4001 and 4052 are more easily damaged with static discharge than the other devices. Take special care when handling these devices. IC sockets are not necessary but I would advise using a 16-pin DIL socket for U4 and U9, and a 14-pin DIL socket for U3.

Trimmers

3/8" horizontal, vertical adjust. For example, Bourns 3386F.

100K trimmer	OFF
20K trimmer	GAIN

Potentiometers

All pots 16mm Alpha or equivalent.

47K or 50K linear	ATTACK, DECAY, SUSTAIN, RELEASE
Alpha pot brackets	3 off

Switches

SPDT, on-on	RNG	- to be connected via twisted pair
SPDT, on-off-on	MODE	- to be connected with single core wire

Miscellaneous

Leaded axial ferrite beads	L1, L2	
MTA156 4 way header	PSU	– Oakley/MOTM power supply
MTA100 6-way header	PWR	– Synthesizers.com power supply
Molex/MTA 0.1” header 3-way	BUS	– for connecting to the Oakley Bus
Molex/MTA 0.1” header 8-way	I/O	– for connecting to sockets
Molex/MTA 0.1” housing 8-way	I/O	– for connecting to sockets
Molex/MTA 0.1” header 4-way	A-D, S-R	– to make the VC-ADSR option.
0.1” jumpers	For fitting to A-D and S-R headers.	
Molex/MTA 0.1” housing 2-way	LED – optional connecting technique for the LED.	
5mm green LED lens	LED	
5mm LED lens securing ring	LED (if lens is not self securing)	

Additional components required if using optional Sock4 board

Molex/MTA 0.1” header 8-way	I/O
Molex/MTA 0.1” housing 8-way	I/O
112APC Switchcraft 1/4” socket	SK1, SK2, SK3, SK4

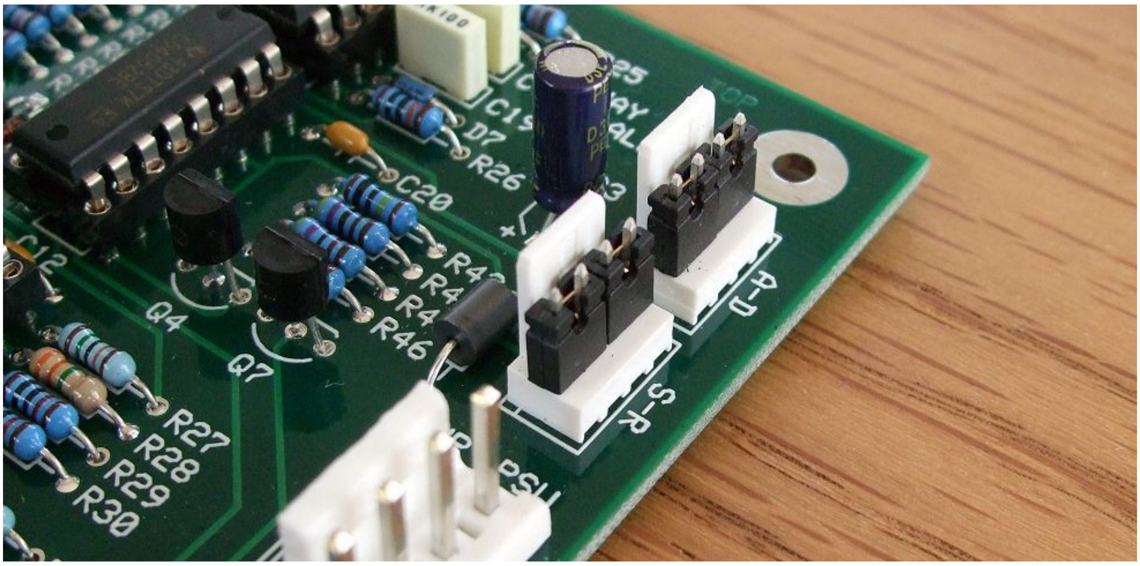
A single wire link is to be fitted to L1 on the Sock4 PCB.

If using Molex KK you'll also need at least 16 crimp terminals.

Suitable lengths of wire to make up the single 100mm interconnect and two cable ties.

Other Notes

For those who have fitted the A-D and S-R 0.1" headers, but are not intending to upgrade to the full VC-ADSR module set immediately, you will need to use those little jumpers. These fit across each pair of pins on the headers, that is pin1/2 and pin3/4.



Four sets of jumpers fitted to two Molex KK headers. This module can now be either expanded to make the VC-ADSR module set, or simply left like this for ordinary ADSR usage.

If you are not intending to ever upgrade your module, then before you can use the module **you must short out the headers**. With four small pieces of uninsulated wire, resistor clippings are fine, you will need to short out the four pairs of pins on both headers. Each header must have the following pins shorted together, pins 1 and 2, and then pins 3 and 4. Pin 1 is depicted by the square pin.

Connections

Power connections – MOTM and Oakley

The PSU power socket is 0.156" Molex/MTA 4-way header. Friction lock types are recommended. This system is compatible with MOTM systems.

Power	*Pin number*
+15V	1
Module ground (0V)	2
Earth or panel ground	3
-15V	4

Pin 1 on the I/O header has been provided to allow the ground tags of the jack sockets to be connected to the power supply ground without using the module's 0V supply. Earth loops cannot occur through patch leads this way, although screening is maintained.

Power connections – Synthesizers.com

The PWR power socket is to be fitted if you are using the module with a Synthesizers.com system. In this case you should not fit the PSU header. The PWR header is a six way 0.1" MTA, but with the pin that is in location 2 removed. In this way location 3 is actually pin 2 on my schematic, location 4 is actually pin 5 and so on.

Power	*Location number*	*Schematic Pin number*
+15V	1	1
Missing Pin	2	
+5V	3	2
Module ground	4	3
-15V	5	4
Not connected	6	5

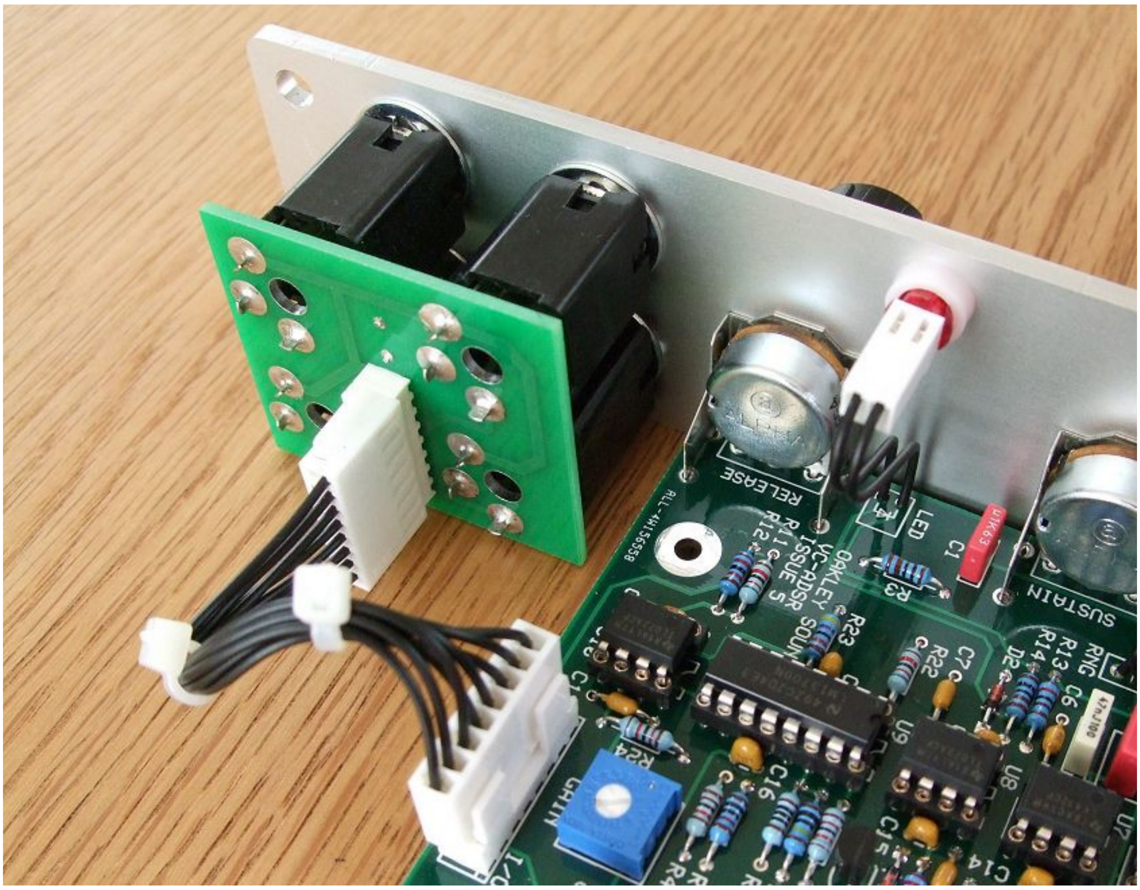
+5V is not used on this module, so location 3 (pin 2) is not actually connected to anything on the PCB.

If fitting the PWR header, you will also need to link out pins 2 and 3 of PSU. This connects the panel ground with the module ground. Simply solder a solid wire hoop made from a resistor lead clipping to join the middle two pads of PSU together.

Building the ADSR module using the Sock4 board

This is the simplest way of connecting all the sockets to the main board. The Sock4 board should be populated in the way described in our construction guide found on the project webpage. There is only one eight way header and it is to be fitted to the bottom side of the board. Don't forget to solder in the wire link L1. Link L2 is left open.

You need to make up only one eight way interconnect. It should be made so that it is 100mm long.



The prototype unit showing the detail of the board to board interconnect. Here I have used the Molex KK 0.1" system to connect the Sock4 to the main PCB.

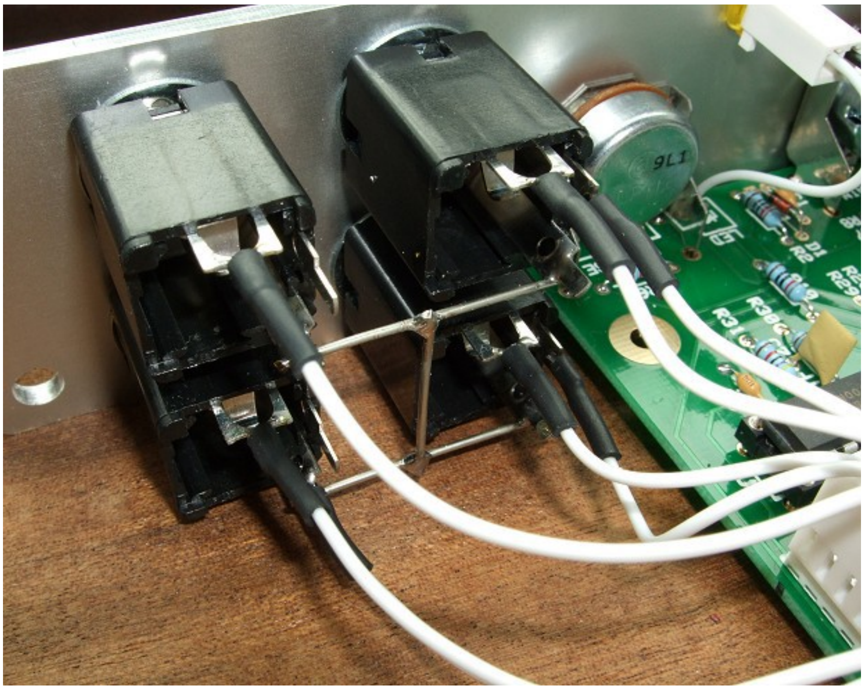
Hand wiring the sockets

If you have bought Switchcraft 112A or 112APCX sockets you will see that they have three connections. One is the earth or ground tag, also called the sleeve connection. One is the signal tag which will be connected to the tip of the jack plug when it is inserted. The third tag is the normalised tag, or NC (normally closed) tag. The NC tag is internally connected to the signal tag when a jack is not connected. This connection is automatically broken when you insert a jack.

Once fitted to the front panel the ground tags of each socket can be all connected together with solid wire. I use 0.91mm diameter tinned copper wire for this job. It is nice and stiff, so retains its shape. A single piece of insulated wire can then be used to connect those connected earth tags to pin 1 of I/O. Pin 1 is the square solder pad.

All the other connections are connected to the signal or NC lugs of the sockets. The table below shows the connections you need to make:

Pin	*Pad name*	*Socket Connection*	*Lug Type*
Pin 1	PANEL_GND	Connect to all sockets	Earth lugs
Pin 2	VCA_OUT	Connect to VCA OUT	Signal lug
Pin 3		No connection	
Pin 4	CV_IN	Connect to VCA IN	Signal lug
Pin 5	CV_NC	Connect to VCA IN	NC lug
Pin 6	ADSR_OUT	Connect to ADSR	Signal lug
Pin 7	GATE_BUSS	Connect to GATE	NC lug
Pin 8	GATE_IN	Connect to GATE	Signal lug

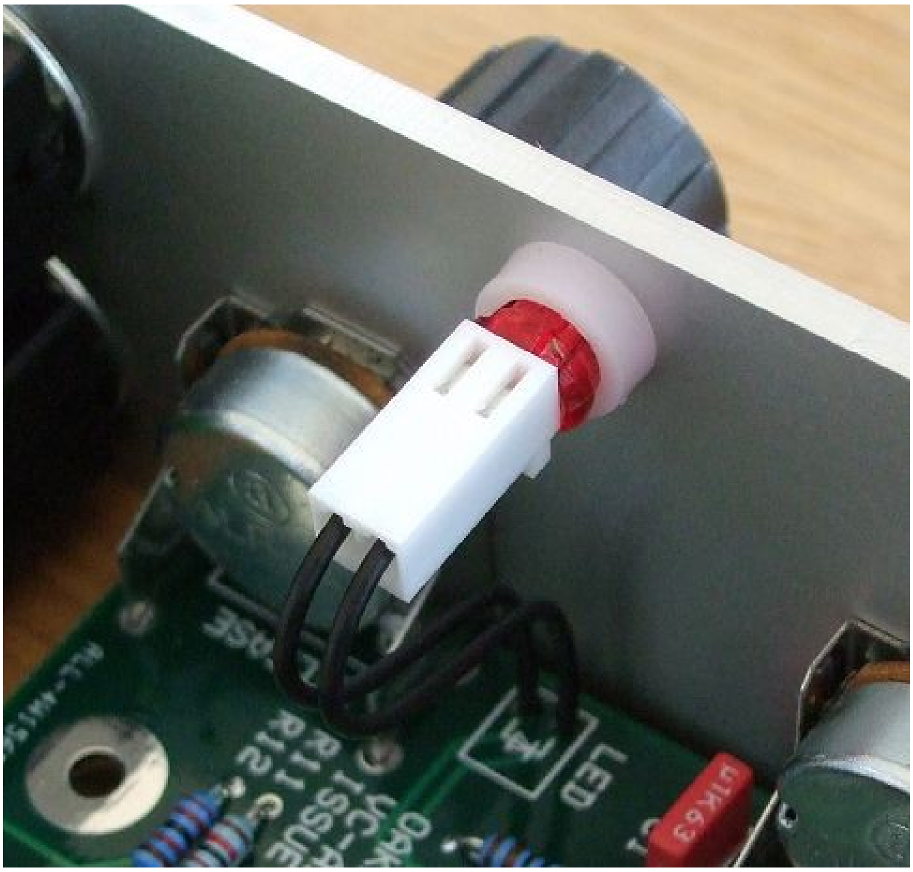


A close up of an older issue 3 build showing the solid core wire frame that joins the earth lugs of the sockets together. A single wire, lower left, goes back to pin 1 on the I/O header on the PCB.

Wiring the LED

Wiring the LED is straightforward. Simply connect the LED to the pads directly below where the LED is mounted onto the front panel. The anode of the LED goes to the square pad and the cathode goes to the round pin.

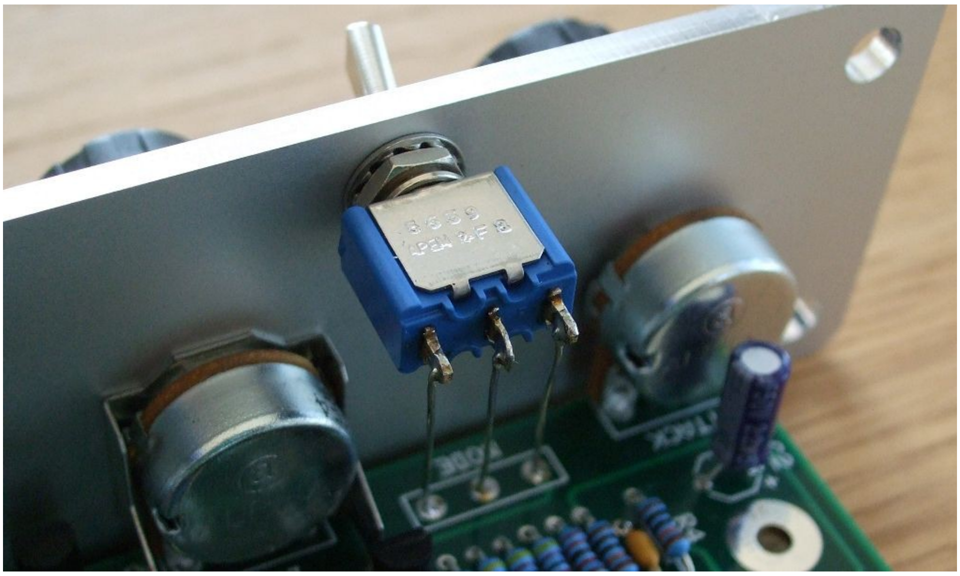
You can either solder your connecting wires to the LED's legs or use a MTA or Molex connector to make the connection.



Using a two way 0.1" Molex KK housing to connect the LED to the circuit board. This red LED is held onto the panel with a red Cliplite lens.

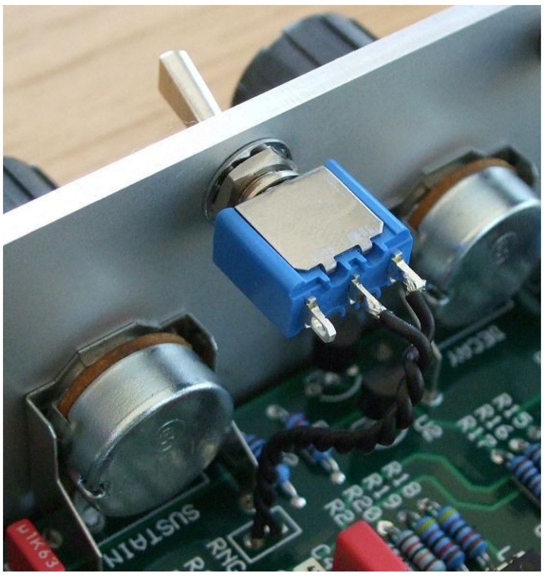
Wiring the switches

The new panel design features two switches. Both are wired in slightly different fashions. The mode switch (on-off-on) is wired with three pieces of straight solid core wire as shown in the picture below:



The solid core wire is looped around each switch lug and soldered in place. The switch needs to be in its place and tightened before soldering the leads.

The range switch is wired with a twisted wire pair. This simply made by twisting two pieces of insulated multistrand wire together to form a simple cable. You can use two different colours but you don't have to. It does not matter which wire connects to which lug on the switch so long as it is the top two lugs that are used. That is the range switch simply shorts out the two connections when set to SLOW.



It is the top two lugs of the switch that connect to the two pads of RNG.

Testing, testing, 1, 2, 3...

Set the mode switch to NORM and the range switch to SLOW. Connect +/-15V to the unit making sure you are applying the power correctly. Make sure that the LED is not lit. If it's on, switch off and check all the parts again thoroughly. If your LED is off, and there is no smoke rising from the board, you should quickly run your finger over all the ICs. Hopefully, none of them are getting too warm to touch. If they are there is a problem.

Click the switch in LOOP and the set the sustain pot to its minimum value. You may see the LED start to flash. Altering the attack value should change the rate at which the LED ramps to full brightness. Altering the decay will affect the time it takes to subside. At fast attack and decay times the LED will simply flicker indicating that the ADSR outputs are now moving very fast indeed. Change the range to SLOW and the LED should pulse at a slower rate. Very slow speeds should be obtainable at the slower settings of attack and decay.

Make sure the looping stops if you turn up the sustain past about 10% of full scale.

If nothing changes when you alter the attack and decay pots do make sure you have the jumpers on, or wire links fitted to, the A-D and S-R headers.

Flip the mode switch back to NORM and use a gate signal from a midi-CV convertor or an LFO's square wave output. Turn all the pots to their minimum value. The LED should briefly blip on for every low to high gate transition. Increase the decay pot, and hopefully, the LED blips will get brighter and last longer.

Now increase the sustain pot. This should increase the LED brightness, and it should stay on for longer. It should now stay on for the time the gate is high.

Increase the attack time, you should notice the LED ramping up to full brightness.

Now connect an audio signal of some sort, any will do, but a simple triangle wave is quite sufficient. You should connect it to the VCA IN socket. Connect the VCA OUT to an input channel on your mixing desk or other audio input. Hopefully, you should find that when the ADSR is gated the audio is heard. Play with the A, D, S and R pots to make sure they do the usual things. If you are not familiar with their action, I suggest you read the section at the front of the User Manual.

Make sure that the ADSR socket also produces an output. This output is not shaped by the VCA IN signal and should be a standard envelope generator output of 0 to +5V. You can test it by routing it to a VCF or VCO.

Final Comments

If you have any problems with 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 are in the UK, 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 those nice people on the Synth-diy and Analogue Heaven mailing lists and those at Muffwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK