

Oakley Sound Systems

5U Oakley Modular Series

VC-LFO

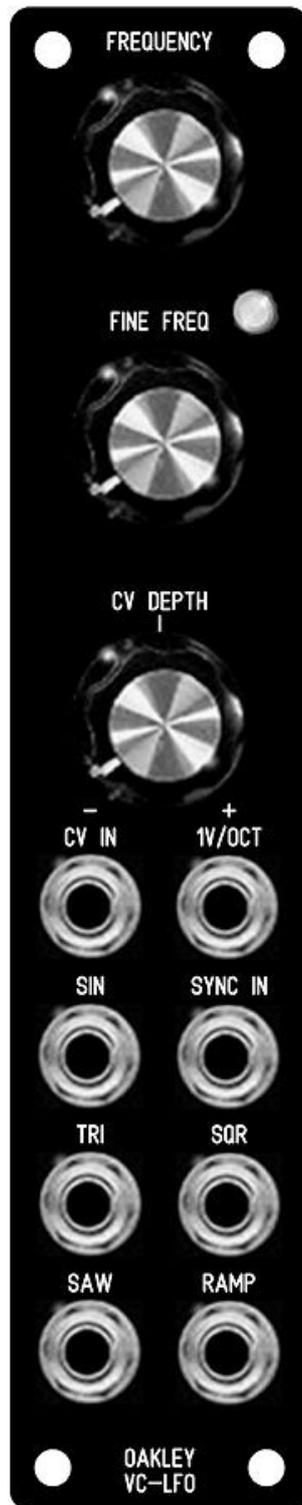
Low Frequency Oscillator

PCB Issue 3 & 3.1

Builder's Guide

V3.4

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom



The suggested panel layout of the issue 3 VC-LFO module. This is different to the older issue 1 VC-LFO panel in that we lose a switch and gain an additional pot.

Introduction

This is the Project Builder's Guide for the issue 3 VC-LFO 5U module from Oakley Sound.

Please note that the earlier issue 3 PCB is incorrectly marked as an issue 2. The way to tell them apart from an original issue 2 board is that the true issue 3 boards are fitted with a HFT trimmer in top right hand corner. Issue 3.1 boards are correctly labelled.

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

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

<http://www.oakleysound.com/lfo.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>.

The issue 3 VC-LFO PCB



The original issue 3 PCB mounted behind a natural finish panel. Note the use of the optional Sock8 socket board to facilitate the wiring up of all those sockets.

I have provided space for the three 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 and many of our other modules.

The design requires plus and minus 15V supplies. The power supply should be adequately regulated. The current consumption is about 40mA for each rail. Power is routed onto the PCB by a four way 0.156" MTA156 type connector or the special five way Synthesizers.com MTA100 header.

The PCB has four mounting holes for M3 screws, one near each corner. These are not required if you are using the two 16mm pot brackets.

The board size is 104mm (high) x 104mm (deep).

The board has been laid out to accept connection to our Sock8 socket board. This small board speeds up the wiring of the eight sockets and reduces the chances of mistakes.

Circuit Description

The VC-LFO circuit consists of several sections spread over two pages; the CV summer and exponential convertor (page 1), the VC-LFO core (page 2), sine shaper (page 1), sawtooth shaper (page 2), LED driver (page 1) and power supplies (page 2).

Looking at the bottom of page 2 of the schematic is the power supply section. This includes the power supply inlets, PWR and PSU, and the power supply filtering components.

U1 provides a stable +5V supply for the DG403's digital input circuitry. The original VC-LFO boards had no such supply and the DG403 was driven from +15V only. However, when Vishay bought out Siliconix, the original manufacturer of the device, some of the specifications changed, in particular the lower threshold voltage for the analogue switch control. This meant that the Vishay specimens of DG403 when used in the VC-LFO did not correctly respond to sync pulses. Vishay DG403 devices required a higher voltage than what provided by my sync input circuit and +5V gate signals were not able to reset the LFO core reliably. By driving the logic part of the DG403 from a 5V supply this insensitivity to input levels is removed.

Two grounds are provided, one for the circuit itself, and one for the earthing of the jack sockets on the front panel. There are eight ICs on this PCB, and each requires power, and the mass of capacitors nearby are power supply decoupling. Both these sections of the circuit are separated from the main circuit to avoid cluttering up the main parts of the diagram. On the PCB itself those decoupling capacitors are actually as close the action as they can be. The closer those caps are to their parent IC, the more effective they become. They are like little reservoirs of charge to provide the current to the IC when it needs it. And they can usually supply it faster than the power supply itself, if only for a very short period of time.

The VC-LFO's pitch is determined by a variety of sources. Two CV inputs, one pot and a switch on the front panel, and a trimmer to set initial frequency. U2a (pins 1, 2, 3) is built as a voltage summer. It takes the voltages from the six sources and adds them together. The gain of the summer is set by the input resistors and R7. The output of the CV summer is then fed to an inverting amplifier of adjustable gain via the V/OCT trimmer. The V/OCT trimmer is adjusted to give a rise of one octave in output frequency when the KEY-CV input voltage goes up by 1V. The output of this amplifier is then fed to the exponential convertor via R47 and TC.

The exponential convertor is based around on half of U3b (pins 5, 6, 7) and the matched PNP pair of U7. Its output is a current that is proportional to the exponent of the voltage applied at the base of the left hand transistor of U7. The circuit gives the VC-LFO a sensitivity of roughly +18mV/octave, so R47 and TC reduce the output of the CV summer to this level. However, it is worth noting that TC is a temperature sensitive resistor. The resistance of TC will go up with temperature at a rate of 0.35% for every degree Celsius. This should counteract the temperature effects produced by the semiconductor junctions in U6. To get the best temperature stability TC is mounted right on top of U7. This way the temperature of the two devices should be the same.

It is possible to get an exponential response from a single transistor, but that has problems as V_{be} , the junction voltage, changes with ambient temperature. The 'temp co' resistor cannot compensate for this change in the transistor's operating current. So the now classic circuit with two perfectly matched transistors and an op-amp, U7 and U3b, is used. Changes in the V_{be} for one half of the transistor pair are mirrored in the other. The op-amp then matches the current in the first transistor with the same current in the other one. So the collector current in the first transistor will effectively control the collector current in the second. And it is the current drawn by the second transistor that controls the frequency of the VCO.

output of the integrator passes straight to the schmitt trigger. This connection can be broken when the sync pulse is applied, but more about this later.

The schmitt trigger is a simple circuit block based around the other half of the OTA. It is configured as an inverting schmitt trigger and its output, at pin 12 is either high at +1.8V, or low at -1.8V. If the output of the Schmitt is initially low, it requires an input of less than -1.8V at the output of the integrator to make it go high. Conversely, the integrator will need to produce an output of +1.8V to make the Schmitt go low again.

To make any oscillator you normally require an output to be fed back into the input. In the VC-LFO, the non inverting integrator is fed by the output of the inverting schmitt trigger. Thus, a 'high' at the output of the schmitt causes the integrator to rise. When the integrator's output reaches a certain point, the schmitt switches state and the integrator's output falls. The schmitt trigger changes state once again, and the process repeats itself...

C12 effectively controls the overall range of frequencies produced by the VC-LFO. You may want to try different values for this capacitor. Lower values of C12 will give you a faster oscillator. Indeed, the VC-LFO will make a very nice audio only VCO if you make C12 1nF.

The two back to back zener diodes, D3 and D4, strapped across the input of U8b prevent the input voltage at pin 13 exceeding the maximum allowed by the LM13700. In ordinary operation this will never happen, but in some cases, if your power supply comes up too slowly, the integrator output can get too positive (or too negative) for the briefest of moments and the schmitt trigger 'latches up'. This nasty bit of behaviour is when the input devices within the LM13700 get turned on too hard and draw excessive current through the input pins. If latch up has occurred then only by removing the input signal to pin 13 (with an active sync pulse) or by powering down the module can the input be unlatched. The two diodes prevent any such latch up occurring in first place no matter what the power supply does on power up.

A tri-colour LED is used to give the user an idea of LFO speed. It will change colour depending on whether the integrator is charging or discharging. It is driven from two transistors, Q1 and Q2, directly from the square wave output.

The triangle output of the module needs to be increased in level before sending out to the front panel. The raw triangle signal is taken from the integrator's output and passed into an inverting amplifier, built around U9b, to increase the +/-1.8V signal to the required +/-5V output level.

The square wave output is derived from the output of the schmitt trigger. It is then fed to an non inverting amplifier, built around U5a, to give use the standard +/-5V output.

When a pulse or gate appears at the 'sync' input, the pulse or gate is shaped by C3, R6 and D2 to give a short positive spike. This controls the two channels of the DG403. Normally, as I mentioned earlier, this part of the DG403 allows the integrator output to pass straight to the Schmitt trigger. However, when a spike is present at pin 10 of the DG403, two things happen. Firstly, the integrator output is cut off from the schmitt trigger by the opening of the FET switch between pins 5 and 6. R20 will then force the schmitt trigger's input low and the Schmitt's output will be set low. Secondly, the integrator capacitor will be shorted out, by the closing of the FET switch between pins 8 and 9. This will set the integrator's output at zero volts. When the 'sync' spike fades away, the integrator will start to ramp positive due to the positive voltage from the schmitt trigger. If we did not force the Schmitt low then the integrator would have randomly ramped either up or down.

The sawtooth and ramp shaping circuitry looks complex, but it can be broken down quite simply.

U5b acts as a dual mode circuit, it is either a buffer or an inverter. It acts on the amplified triangle wave output. What mode it is in is determined by the switch within one of the channels of the DG403. When the internal switch is closed, U4 acts as an inverting amplifier with gain -1. Pin 5 is effectively tied to ground by the action of the switch. But when the switch is opened, pin 5 is attached to the triangle wave input via R23 and R26. These two resistors form a parallel combination of exactly 11K. U4 now acts as a unity gain buffer, ie. an amplifier with a gain of +1.

The switch in U3 is controlled by the square wave output. That means when the integrator is discharging it is switched off and U4 is in 'buffer' mode. When the integrator is charging, U4 is in inverting mode. The triangle input is then switched rapidly from being buffered and unaffected, to being inverted. This chops the regular up and down waveform of the triangle into two sections, one as before, the other tipped upside down. This creates two sections of ramping up, ie. a ramp waveform but of twice the original frequency. By adding a square wave signal to this ramp signal in the correct proportions we get another ramp waveform but at the original frequency.

U10b (pins 5, 6, 7) is a standard inverting summing circuit. It combines the square wave and twice frequency ramp signal and inverts the sum to get a sawtooth. This is our sawtooth waveform and this goes through R37 to connect to the output socket. A ramp waveform is simply created by inverting the sawtooth output with an inverting op-amp based around U10a (pins 1, 2, 3), shown on page one.

The sine wave shaper circuit, also shown on page one of the schematics, acts upon the raw triangle wave output of the integrator. The rounded peaks of the sine wave are created by deliberately overdriving the inputs of a differential amplifier built around the NPN pair in U7 and recovery amplifier U2a. The non linearities of the input stage being utilised to squash the top and bottom peaks of the triangle wave input. The gain of the shaper is fixed partly by R57. SHP adjusts the amount of overdrive.

R58 and R59 provide a special feedback path, called emitter degeneration, within the input circuit. This causes the input devices to be tickled even further into overdrive at the critical peaks of the output waveform.

The trimmer, OFF, provides compensation for any small imbalance in the NPN transistors. Any differences in base-emitter voltages between the two transistors will make them soft clip asymmetrically. By adding a small voltage of the opposite polarity to one of the input pins, we can cancel the effects of the offset.

The sine wave created by this process does not give us a perfect textbook example of a sine wave. However, the wave is very low in harmonics and has a nice rounded shape. This makes it perfect for modulation purposes and sounds good too.

Issue 3 & 3.1 VC-LFO Parts List

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>.

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. M 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 0.25W 1% metal film (MF) types unless stated.

22R	R1
390R	R45, R52
820R	R2, R3
1K	R38, R37, R36, R34, R40
2K	R30
3K	R41
3K3	R43
4K7	R57
5K6	R29, R39
10K	R33, R42, R5, R18, R35
18K	R53
20K	R58, R59
22K	R23, R27, R4, R26, R24
30K	R21
33K	R15, R50, R32
39K	R46, R47, R44, R25, R28
47K	R16
51K	R22, R54, R49, R11, R48
68K	R7, R55
100K	R20, R6, R19, R14, R12
180K	R9
220K	R13
470K	R10, R17, R51, R56
680K	R8
1M	R31
1K +3000ppm/K PTC	TC*

* This is fitted across U7 and soldered only after U7 has been placed. See picture earlier on in this document.

Capacitors

100nF multilayer axial ceramic	C4, C10, C6, C5, C13, C14, C7
150pF ceramic C0G 2.5mm	C3
33pF ceramic C0G 2.5mm	C8, C11
470pF ceramic C0G 2.5mm	C9
4n7, 63V polyester film	C12
2u2, 63V electrolytic	C1, C15, C16
22uF, 25V electrolytic	C2

Semiconductors

1N4148 signal diode	D1, D5, D7
BAT42 Schottky diode	D6
2V7 500mW zener diode	D3, D4
6V2 500mW zener diode	D2
BC549 NPN transistor	Q1, Q2
Red/Green tri-colour LED	LED1 (Mounted off board)

Integrated Circuits

78L05 +5V 100mA regulator	U1
LM13700N dual OTA	U8
TL072CN dual op-amp	U10, U6, U3, U5, U2
LT1057CN8 dual op-amp	U9
DG403 dual analogue switch	U4
THAT340 matched NPN/PNP pair	U7

IC sockets can also be used. You need two 16 pin DIL sockets, one 14 pin DIL socket, and six 8 pin DIL sockets.

Pots

All pots 16mm Alpha or equivalent.

47K or 50K linear	FINE, CV1-DEPTH, COARSE
Alpha pot brackets	2 off

Trimmers

500R 6mm horizontal (eg. Bourns 3386F)	SHP
2K 6mm horizontal (eg. Bourns 3386F)	HFT
50K 6mm horizontal (eg. Bourns 3386F)	STEP
100K 6mm horizontal (eg. Bourns 3386F)	OFF

100K multiturn (eg. Bourns 3296W)
20K multiturn (eg. Bourns 3296W)

TUNE
V/OCT

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 6-way	UPR – for connecting to sockets
Molex/MTA 0.1” header 8-way	LWR – for connecting to sockets
Molex/MTA 0.1” housing 6-way	UPR – for connecting to sockets
Molex/MTA 0.1” housing 8-way	LWR – for connecting to sockets
Molex/MTA 0.1” housing 3-way	LED1 – optional connecting technique for the tri-colour LED.
5mm clear LED lens	LED1
Switchcraft 112APC 1/4” sockets	Eight off mounted either on the Sock8 board or on panel
Two knobs	
Power lead MTA to MTA connector	

Additional components required if using optional Sock8 board

Molex/MTA 0.1” header 6-way	UPR
Molex/MTA 0.1” header 8-way	LWR
Molex/MTA 0.1” housing 6-way	UPR
Molex/MTA 0.1” housing 8-way	LWR
112APC Switchcraft 1/4” socket	SK1, SK2, SK3, SK4, SK5, SK6, SK7, SK8

If using Molex KK you'll also need at least 28 crimp terminals.

Suitable lengths of wire to make up the two interconnects and three cable ties.

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.

<i>Power</i>	<i>Pin number</i>
+15V	1
Module ground (0V)	2
Socket ground	3
-15V	4

Pin 1 of the LWR header is connected to pin 3 of the PSU header and 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.

MU and 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 the pin in location 2 is removed. In this way location 3 is actually pin 2 on my schematic, location 4 is actually pin 5 and so on.

<i>Power</i>	<i>Location number</i>	<i>Schematic Pin number</i>
+15V	1	1
Missing Pin	2	
+5V	3	2
Module ground (0V)	4	3
-15V	5	4
Socket Ground *	6	5

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

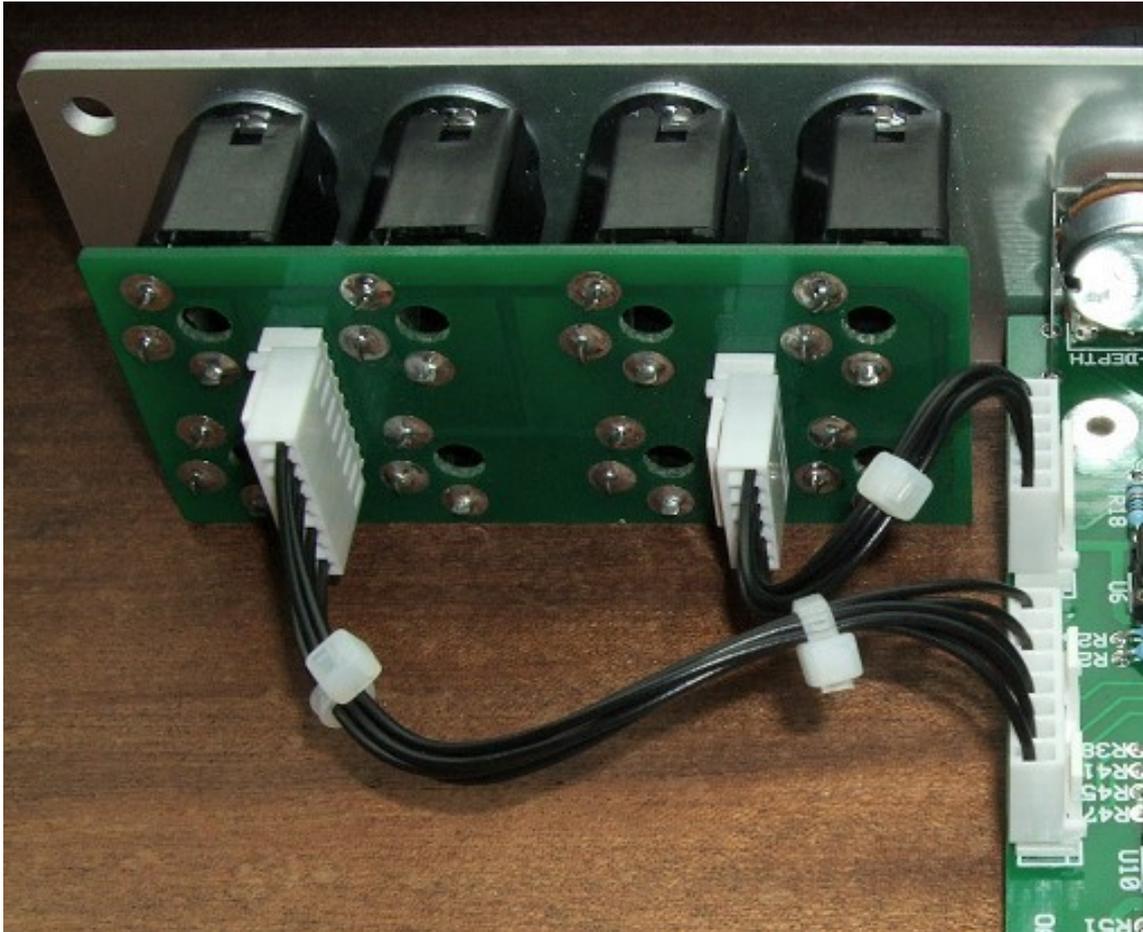
If fitting the PWR header and using it with a standard MU power distribution system, you will also need to connect together the middle two pads of the PSU header on the main board. This link connects the socket and panel ground with the module ground. Simply solder a solid wire hoop made from a resistor lead clipping, or bit of solid core wire, to join to the two middle pads of PSU.

* Issue 3.1 boards now connect the normally unused pin 6 of the MU connector to socket ground. With the link on PSU not fitted, and using an Oakley MU Dizzy distribution board with a five way power cable, will allow the socket ground to be kept separate from module ground to prevent ground loops.

Using the Sock8 board

This is the simplest way of connecting all the sockets to the main board. The Sock8 board should be populated in the way described in our construction guide found on the project webpage. There are only two headers, UPR (for upper) which is six way, and LWR (for lower) which is eight way. Both headers are fitted to the bottom side of the board. L1, the optional wire link, on the Sock8 board is not fitted.

You need to make up two interconnects. The six way one should be made so that it is 90mm long. The eight way should be made to be 140mm.



Both board interconnects made up and cable tied into place for neatness. This is the cheaper but perfectly decent Molex KK 0.1" header system.

Wiring the sockets manually

If you have bought Switchcraft 112A sockets you will see that they have three connections. One is the earth or ground tag. 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 LWR. Pin 1 is the square solder pad.

The NC lugs of CV, 1V/OCT and SYNC IN sockets should be connected to module GND. This can be found on pins 2, 3, and 5 of the UPR header.

All the other connections are connected to the signal lugs of the sockets. The tables below show the connections you need to make:

UPR

<i>Pin</i>	<i>Pad name</i>	<i>Socket Connection</i>	<i>Lug Type</i>
Pin 1	SYNC-IN	Connect to SYNC IN	Signal lug
Pin 2	module ground	Connect to SYNC IN	NC lug
Pin 3	module ground	Connect to 1V/OCT	NC lug
Pin 4	KEY_CV input	Connect to 1V/OCT	Signal lug
Pin 5	module ground	Connect to CV	NC lug
Pin 6	CV1-IN	Connect to CV	Signal lug

LWR

<i>Pin</i>	<i>Pad name</i>	<i>Socket Connection</i>	<i>Lug Type</i>
Pin 1	Panel ground	Connects to all sockets	Ground lugs
Pin 2	RAMP-R	Connect to RAMP	Signal lug
Pin 3	SAW-R	Connect to SAW	Signal lug
Pin 4	SQR-R	Connect to SQR	Signal lug
Pin 5	Not connected		
Pin 6	SIN-R	Connect to SIN	Signal lug
Pin 7	Not connected		
Pin 8	TRI-R	Connect to TRI	Signal lug

Connecting the Tri-colour LED



Using a three way Molex KK housing to connect the tri-colour LED to the circuit board. The LED is held onto the panel with a clear Ciplite lens, or equivalent, available from various places.

You may be able to wire the LED directly to the circuit board if yours has long enough leads. Most of the ones I have seen have shorter legs than ordinary 5mm LEDs so I have to use flying wires to connect the LED to the board. Tri-colour LEDs have three legs and each one should go to the solder pad directly beneath it when it is mounted into the panel.

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

I normally wire my coloured LEDs so that they go red when the square wave output is positive.

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 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 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

© December 2010 – updated September 2020

No part of this document may be copied by whatever means without my permission.