# Oakley Sound Systems

# VC-LFO

## Issue 1

# User's Guide

## V1.1

Tony Allgood B.Eng PGCE
Oakley Sound Systems
PENRITH
CA10 1HR
United Kingdom

# Introduction

The Oakley VC-LFO is a versatile yet easy to use voltage controlled low frequency oscillator module that can also be used as an audio VCO over a limited range. The unit features a low distortion sinusoid, sawtooth, ramp, triangle and square outputs. All are available simultaneously. A sync input will reset the waveforms to a fixed point when a sharp rising edge is received. This would normally come from a sawtooth or pulse wave input from another LFO or VCO, but it can also come from the system's Gate or trigger signals.

The module features two CV inputs. CV1 is controlled by a single pot on the front panel. When the pot is turned right of its central position, the pot acts as a normal attenuator; increasing the sensitivity of the CV input to a maximum of 0.5V/octave. Left of centre, the pot will act in inverting mode. Fully anti-clockwise the input will respond to -0.5V/octave. CV2 is fixed at 1V/octave.

A bicolour LED indicates the output status and frequency. An optional range switch determines whether the unit is to be run in audio or very low frequency mode. A large degree of overlap is available for really wide sweeps.

**Specifications**

The operating range of this VC-LFO is 0.002Hz to 11kHz via the CV inputs.

Using the front panel pot alone the following frequencies are obtainable:

**Hi Range:** 0.016Hz to 2kHz.

**Lo range:** 0.002Hz (cycle time of 8 minutes) to 150Hz

These frequencies can be altered over a limited range by the onboard trimmer, or more significantly by small change in the value of the timing capacitor.

As an audio VCO the usable range that conforms to 1V/octave (to +/-3cents) is over four octaves from approximately 32Hz to 750Hz. For accurate musical use over this frequency we recommend that you use the Oakley VCO.

As with all Oakley projects the PCB is double sided with through plated holes, has solder mask both sides, and has component legending for ease of construction. The PCB is 4.1" x 4.1" in size.

# Of Pots and Power

There are just two control pots that are directly mounted on the PCB. The pots are the main frequency control and the CV2 modulation depth attenuator. If you use the specified pots and brackets, the PCB can be held firmly to the panel without any additional mounting procedures. The pot spacing is on a 1.625" grid and is the same as the vertical spacing on the MOTM modular synthesiser.

The VC-LFO board features Spectrol 248 pots instead of the Omeg pots previously used. The pots maybe different to the Omeg E16 pots we use on other modules. They are a different shape and require different solder pads and mounting brackets.

The PCB has four mounting holes, one in each corner should you require support if you are not using the pot brackets.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 25mA per rail. Power is routed onto the PCB by a four way 0.156” Molex type connector. Provision is made for the two ground system as used on all Oakley modular projects, and is compatible with the single ground MOTM systems. See later for details. This unit should run from a +/-12V supply with a slight change in operating frequencies and output levels. However, this has not been tested.

## 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), 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 inlet, PWR and the power supply filtering components.

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. U1 (pins 1, 2, 3) is built as a voltage summer. It takes the voltages from the five sources and adds them together. The gain of the summer is set by the input resistors and R14. The output of the CV summer is then fed to an inverting amplifier of trimmable gain via the V/OCT trimmer. The V/OCT trimmer is adjusted to give a rise of one octave in output frequency when the CV2 input voltage goes up by 1V. The output of this amplifier is then fed to the exponential convertor via R32 and TC.

The exponential convertor is based around on half of U2 (5, 6, 7) and the matched PNP pair of U6. Its output is a current that is proportional to the exponent of the voltage applied at the base of the left hand transistor of U6. The circuit gives the VC-LFO a sensitivity of roughly +18mV/octave, so R32 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 U6. 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 Vbe, 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, U6 and U2, is used. Changes in the Vbe 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.

The output of the exponential convertor is a current. It is this current that controls the core of the oscillator. Contrary to most people’s ideas, the core of a VCO is typically a linear CCO. That is a current controlled oscillator. A doubling of current to the CCO will produce a doubling of output frequency. The core of this VC-LFO creates a triangle wave from which all the other waveshapes are created. This is different to the standard sawtooth core design used by the Oakley VCO.

The core itself is built from two main parts. An integrator and a schmitt trigger. The output from each feeds into the next, and then right round again. We will start by looking at each bit in turn.

The heart of the VC-LFO core is half of a 13700 IC, U7. This is a dual *operational transconductance amplifier*, or OTA for short. The OTA is different in several respects to a usual op-amp. Firstly, its output is a current, not a voltage. Secondly, its gain is controlled by a current injected into the Iabc pin. ‘I’ is for current, ‘abc’ stands for amplifier bias current. The bigger this current the higher the gain of the OTA. However, the current into the Iabc pin of the 13700 must not exceed 2mA otherwise damage will result to the OTA. R44 limits the Iabc to a safe value.

Together with op-amp U8 (pins 1, 2, 3),  U7 forms the inverting integrator whose time constant is controlled by the Iabc. Any positive voltage applied to R39, will cause the voltage to fall at the output of the op-amp. The speed at which the voltage falls is controlled by C1 and the Iabc current. If the applied voltage to R39 is negative the op-amp’s output will rise. It is the integrator’s output that will be used as the source for the triangle output.

The DG403 is configured as four electronically controlled switches. They are arranged in pairs, so that when one switch of the pair is closed the other is open. When a switch is closed the signal can pass through pretty much unaffected. One channel of the DG403 (pins 5 & 6) is wired so that the 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 an op-amp U4 (pins 1, 2, 3) and a push-pull transistor network, Q1 to Q4. Its output, at the collectors of Q3 and Q4 is either high at +15V, or low at -15V. If the output of the Schmitt is initially low, it requires +5V at the output of the integrator to make it go high. The integrator will need to produce an output of -5V 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 integrator is fed by the output of the schmitt trigger. Thus, a ‘low’ 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....

C11 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 C1 will give you a faster oscillator.

A bi-colour LED is controlled by the output of the op-amp directly. It will change colour depending on whether the integrator is charging or discharging.

The transistor push-pull circuit is used to guarantee a symmetrical square wave output from the schmitt trigger. If we had used the op-amp output directly, it is likely to be at an unspecified level (around +/-13V or so) and its edges would not have been so clean. Using the transistors, we can derive the square outputs from the supply rails. R1 and C3 provide a little filtering from noise on the positive rail, and also prevent any switching noise from getting back onto the rail from the transistor circuit. R9 and C4 do the same for the negative rail. It should be noted that the symmetry of the square wave will depend on the symmetry of your power supply.

The one problem with using an ordinary op-amp as a schmitt trigger is its slow response speed. It takes a finite time for the op-amp’s output to swing from one output state to the other. Simply put any op-amp doesn’t like being forced to its limits, that is into what is called ‘output saturation’. The external transistor circuit can help in this matter, but it too has its own response time. The effect of this slowness to respond allows the integrator to overshoot when the core is running at high frequencies. This means that the triangle waveform will get larger in amplitude as the frequency gets bigger. Fortunately, these effects are only pronounced at high frequencies and shouldn’t bother us too greatly when using the module as a modulation source. However, they do have a knock on effect of making the VCO’s 1V/octave response go flat. This ultimately limits the usefulness of this particular core as musical audio VCO to around 800Hz.

The triangle output of the module is taken from the integrator’s output via R37. The output level is controlled by the Schmitt trigger’s threshold points and is set by R21 and R20 at +/-5V.

The square wave output is derived from the output of the push-pull circuit. It is then fed to an inverting amplifier of gain -0.33. This gives us the +/-5V output we require.

When a pulse or gate appears at the ‘sync’ input, the pulse or gate is shaped by C2, R3 and D1 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. R22 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 negative voltage from the schmitt trigger. If we did not force the Schmitt low, then the integrator may ramp down or up. A good VC-LFO should have a predictable behaviour, and ramping up is the most sensible.

The sawtooth and ramp shaping circuitry looks complex, but it can be broken down quite simply. U4 acts as a dual mode circuit, its either a buffer or an invertor. It acts on the 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 a 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 R25 and R28. These two resistors form a parallel combination of exactly 5K. 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 charging it is switched off and U4 is in 'buffer' mode. When the integrator is discharging, 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 squarewave signal to this ramp signal in the correct proportions we get another ramp waveform but at the original frequency.

U9 (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 is goes through R36 to connect to the output socket. A ramp waveform is simply created by inverting the signal again with an inverting op-amp based around U9 (pins 1, 2, 3).

The sine wave shaper circuit, also shown on page one of the schematics, acts upon the triangle wave output. The rounded peaks of the sinewave are created by deliberately overdriving the inputs of the other half of U7, the operational transconductance amplifier (OTA). This time the OTA is acting as a soft clipping unit. The non linearities of the OTA's input stage being utilised to squash the top and bottom peaks of the triangle wave input. The gain of the shaper is fixed with R47 and R49. SHP adjusts the amount of overdrive. Since U2 gives a current output, it must be turned into a voltage. U8 (pins 4, 5, 6) does this job.

R45 provides a special feedback path within the shaper circuit. This causes the OTA to be tickled even further into overdrive at the critical peaks of the output waveform.

OFF provides compensation for the OTA's own offset voltage. This offset voltage means that the OTA will 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 is perfect for modulation purposes and sounds good too.

## Sourcing your Components

Most of the parts are easily available from your local parts stockist. I use Rapid Electronics, RS Components, Maplin and Farnell, here in the UK. Rapid's telephone number is 01206 751166. They offer a traditional 'paper' catalogue as well as an on-line ordering service.

In North America, companies called Mouser, Newark and Digikey are very popular. In Germany, try Reichelt, and in Scandinavia you can use Elfa. All companies have websites with their name in the URL. In the Netherlands try using www.telec.com.

The board mounted pots are Spectrol 248 conductive plastic types and are held onto the board with specially made Oakley pot brackets. Two pot brackets and an extra pair of nuts are required and these are provided with the 'pot mounting kit'.

You could use any pot type you want, but not all pots have the same pin spacing. Not a problem, of course, if you are not fitting them to the board.

In the UK, Farnell, CPC and Rapid Electronics sell the Spectrol pots at a very good price.

For the resistors 5% 0.25W carbon types may be used for all values. But I would go for 1% 0.25W metal film resistors throughout since they are very cheap these days, and are more useful for any other Oakley projects you may want to build.

All the electrolytic capacitors should be 25V or 35V, and radially mounted. Don't chose too high a working voltage like 63V. The higher the working voltage the larger the size of the capacitor. A 220V capacitor will be too big to fit on the board.

The pitch spacing of the 1nF and 4n7 polyester capacitors is 5mm (0.2"). These common types come in little plastic boxes with legs that stick out of the bottom. They come in a variety of styles and names, but they should be described as metalised polyester types. For either value try to get those with an operating voltage of 63V or at the most 100V. Higher operating voltages can be used but will probably be too large to fit on the PCB.

The PCB is another Oakley board to feature spacing to incorporate axial ceramics for the power supply decoupling. These are good components with an excellent performance. Various types exist but I tend to use the better quality C0G types from Farnell.

The low capacitance (values in pF) ceramic capacitor, C14, has a 5mm (0.2") lead spacing. For these parts use low-K types, these are the better quality ones with higher stability and lower noise. They are sometimes described as NP0 or C0G types. You can chose either radial multilayer types, or ordinary plate types. RS-Components sell the former, whilst plate types can be bought from pretty much anywhere.

L1 and L2 are radially mounted ferrite beads. These look a little like black resistors. They are usually in the EMC or Filtering section of your components catalogue. Farnell make a good one, part number: 108-267

The BC560 transistors can be pretty much any PNP transistor that corresponds to the same pin out. For example: BC558, BC559 etc. The BC550 transistors can be pretty much any NPN transistor that corresponds to the same pin out. For example: BC548, BC549 etc. Quite often you see an A, B or C suffix used, eg. BC549B. This letter depicts the gain or grade of the transistor (actually hfe of the device). The VC-LFO is designed to work with any grade device although I have used ungraded BC558 throughout in my prototype.

U6 is specified as an SSM2220. This is a high quality PNP matched pair. It is available in the UK from Farnell and RS. However, for standard LFO applications you can substitute this part with two BC560 transistors. Take care in the mounting of these. More detail is given later in this document.

TC is a positive temperature coefficient (PTC) resistor. This means it resistance goes up with temperature. Its there to keep the LFO's operating frequency relatively stable as the ambient temperature changes. Its a bit of a luxury because its not essential to the working of the LFO. If you don't want the expense of fitting it, you can use an ordinary 1K resistor for TC. The PTC I use for this job is Farnell part number 732-278. Its a 1K +3000ppm/K 900mW device.

All ICs are dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP or a CN in their part numbers. For example; TL072CN and LM13700N. Do not use SMD, SM or surface mount packages. They do not fit at all.

The LM13700N may be substituted with either the older 13600 (still made by JRC) and Phillips' NE5517.

The three standard trimmers are sealed carbon units but are smaller fitting than some Oakley boards you may have built before. These are adjusted from the top and, as such, are called horizontally mounted types. Spectrol, Bourns and other companies make suitable types. Lead spacing is 0.2" for the track ends, and the wiper is 0.2" away. Rapid, Farnell and Maplin sell these parts at reasonable cost. You can use the more expensive cermet types if you wish, but stability is not critical for this application.

The multiturn trimmers are the ones that have the adjustment on the top of the box. Spectrol and Bourns make these. Some types are 22 turns, while others are 25 turns. Either will do. They should have three pins that are in a line at 0.1" pitch. Don't chose the 10-turn ones with the adjustment on the end, they won't fit on the PCB.

The toggle switch is a SPDT (single pole, double throw) type. Get a good quality one. Apem and C&K make good ones. Farnell part number: 147-772.

Input and output sockets are not board mounted. You can choose whichever type of sockets you wish. I use the excellent Switchcraft 112 as used on the Moog and MOTM modulars. At least three of the sockets must have normalising lugs if you are building the suggested layout. The Switchcraft 112 types have normalising lugs as standard.

I am now able to supply four way MTA power leads at either 40cm or 60cm lengths.

Finally, if you make a circuit change that makes the circuit better, do tell me so I can pass it on to others.

# Parts List

A quick note on European part descriptions. 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, 4n7 is a 4.7 nF capacitor.

**Resistors**

Resistors 1/4W, 5% or better.

| | |
|---|---|
| 100K | R19, R31, R33, R20, R12, R4, R3 |
| 10K | R24, R10, R6, R9, R25, R8, R44, R7, R26, R28, R29 |
| 180K | R15, 16, R50 |
| 1K | R38, R52, R13, R37, R36 |
| 1K +3000ppm/K PTC | TC ** See text ** |
| 1M | R35 |
| 220K | R47 |
| 22K | R39 |
| 22R | R1, R2 |
| 2K2 | R34, R5 |
| 330R | R42 |
| 33K | R21, R22, R11, R46, R40 |
| 390R | R48, R51 |
| 39K | R30, R27, R32, R43, R49, R41 |
| 390K | R17 |
| 470K | R23, R54 |
| 51K | R18, R56 |
| 62K | R45 |
| 68K | R14, R53 |
| 8K2 | R55 |

**Capacitors**

| | |
|---|---|
| 100nF multilayer axial ceramic | C12, C6, C7, C8, C9, C5, C1, C13 |
| 1nF, 100V polyester film | C10, C2 |
| 220pF low-K ceramic plate | C14 |
| 2u2, 63V electrolytic | C16, C15 |
| 22uF, 25V electrolytic | C3, C4 |
| 4n7, 63V polyester | C11 |

**Semiconductors**

| | |
|---|---|
| 1N4148 signal diode | D2, D3 |
| BAT42 schottky diode | D1 |
| BC560 PNP transistor | Q2, Q3 |
| BC550 NPN transistor | Q1, Q4 |

| | |
|---|---|
| Red/Green bi-colour LED | LED |

**Integrated Circuits**

| | |
|---|---|
| LM13700N dual OTA | U7 |
| TL072CN dual op-amp | U8, U5, U1, U2, U9, U4 |
| DG403 dual analogue switch | U3 |
| SSM2220 dual matched PNP pair | U6 |

**Other**

| | |
|---|---|
| 4-way 0.156” Molex/MTA connector | PWR |
| 50KA or 100KA linear variable resistor | FREQ, CV1 |
| Leaded axial ferrite beads | L1, L2 |
| 100K multiturn trimmer | TUNE (optional) |
| 20K or 25K multiturn trimmer | V/OCT |
| 100K horizontal trimmer | OFF |
| 20K or 25K horizontal trimmer | SHP |
| 47K or 50K horizontal trimmer | STEP |
| SPDT toggle switch | SW1 |
| Switchcraft 112APC 1/4” sockets | Eight of them |

1m of multistrand hook up wire in a variety of colours
20 cm of 0.91mm solid core wire
Two knobs
Power lead MTA to MTA connector

For the suggested layout you need eight decent quality jack sockets, eg. Switchcraft 112. I use the PCB mount types, but you can use solder tag types if you prefer.

You may well want to use sockets for the ICs. I would recommend low profile turned pin types as these are the most reliable. You need seven 8-pin and two 16-pin DIL sockets.

## Building the Oakley VC-LFO Module

Occasionally people have not been able to get their Oakley projects to work first time. Some times the boards will end up back with me so that I can get them to work. To date this has happened only a few times across the whole range of Oakley PCBs. The most common error with four of these was parts inserted into the wrong holes. Please double check every part before you solder any part into place. Desoldering parts on a double sided board is a skill that takes a while to master properly.

If you have put a component in the wrong place, then the best thing to do is to snip the component’s lead off at the board surface. Then using the soldering iron and a small screwdriver prize the remaining bit of the leg out of the hole. Use wick or a good solder pump to remove the solder from the hole. Filling the hole with fresh solder will actually make the hole easier to suck clean!

I always using water washable flux in solder these days for my board manufacture. In Europe, Farnell and Rapid sell Multicore's Hydro-X, a very good value water based product. You must wash the PCB at least once every two or so hours while building. Wash the board in warm water on both sides, and use a soft nail brush or washing up brush to make sure all of the flux is removed. Make sure the board is dry before you continue to work on it or power it up. I usually put the board above a radiator for a few hours. It sounds like a bit of a hassle, but the end result is worth it. You will end up with bright sparkling PCBs with no mess, and no fear of moisture build up which afflicts rosin based flux. Most components can be washed in water, but **do not** wash a board with any trimmers, switches or pots on it. These can be soldered in after the final wash with conventional solder or the new type of 'no-clean' solder.

I have found that if you are using a very hot soldering iron it is possible to run your iron so hot as to boil the flux in the 'water washable flux' or some types of 'no-clean' solder. This is not a good idea as it can create bubbles in the solder. If you prefer to have a fixed temperature iron, then it is best to get a 18W one for this purpose. I use an ordinary Antex 240V 25W iron with a Variac power supply running at 200V. This seems to work well for me.

All resistors should be flat against the board surface before soldering. It is a good idea to use a 'lead bender' to preform the leads before putting them into their places. I use my fingers to do this job, but there are special tools available too. Once the part is in its holes, bend the leads that stick out the bottom outwards to hold the part in place. This is called 'cinching'. Solder from the bottom of the board, applying the solder so that the hole is filled with enough to spare to make a small cone around the wire lead. Don't put too much solder on, and don't put too little on either. Clip the leads off with a pair of side cutters, trim level with the top of the little cone of solder.

Once all the resistors have been soldered, check them ALL again. Make sure they are all soldered and make sure the right values are in the right place.

The diodes can be treated much like the resistors. However, they must go in the right way. The cathode is marked with a band on the body of the device. This must align with the vertical band on the board. In other words the point of the triangular bit points *towards* the cathode of the diode.

TC is the temperature sensing resistor and it is normally mounted on top of U6. This should obviously be done after you have fitted U6 so can be left until later. The TC device effectively straddles the IC package with its long leads bent accordingly. You can use heat conductive paste to bond the two devices together, but I think this is unnecessary in this application.

IC sockets are to be recommended, especially if this is your first electronics project. Make sure, if you need to wash your board, that you get water in and around these sockets.

For the transistors match the flat side of the device with that shown on the PCB legend. Push the transistor into place but don't push too far. Leave about 0.2" (5mm) of the leads visible underneath the body of transistor. Turn the board over and cinch the two outer leads on the flip side, you can leave the middle one alone. Now solder the middle pin first, then the other two once the middle one has cooled solid.

Sometimes transistors come with the middle leg preformed away from the other two. This is all right, the part will still fit into the board. However, if I get these parts, I tend to ‘straighten’ the legs out by squashing gently all the three of them flat with a pair of pliers. The flat surface of the pliers’ jaws is parallel to the flat side of the transistor.

The matched pair of PNP transistors in U6 can be substituted by two discrete devices in non critical applications. The board is laid out to show the DIL8 package of the SSM2220, but it is easy enough to fit two separate BC560 transistors if you wish. The transistors should be mounted using the top three pins on each side of the DIL layout. Both transistors should be fitted so that their flat side faces left, ie. the emitters should be in pin 3 for the left hand one and pin 6 for the right. The TC resistor can simply lay inbetween the two transistors, flat against the board like an ordinary resistor.

The polyester film capacitors are like little coloured boxes. Push the part into place up to the board’s surface. Little lugs on the underside of the capacitor will leave enough of an air gap for the water wash to work. Cinch and solder the leads as you would resistors.

The axial multilayer ceramics can be treated like resistors. Simply bend their legs to fit the 7.5mm (0.3”) spacing holes.

The electrolytic capacitors are very often supplied with 0.1” lead spacing. My hole spacing is 0.2”. This means that the underside of these radial capacitors will not go flat onto the board. This is deliberate to allow the water wash to work effectively, so don’t force the part in too hard. The capacitors will be happy at around 0.2” above the board, with the legs slightly splayed. Sometimes you will get electrolytic capacitors supplied with their legs preformed for 0.2” (5mm) insertion. This is fine, just push them in until they stop. Cinch and solder as before. Make sure you get them in the right way. Electrolytic capacitors are polarised, and may explode if put in the wrong way. No joke. Oddly, the PCB legend marks the positive side with a ‘+’, although most capacitors have the ‘-’ marked with a stripe. Obviously, the side marked with a ‘-’ must go in the opposite hole to the one marked with the ‘+’ sign. Most capacitors usually have a long lead to depict the positive end as well.

I would make the board in the following order: resistors, multilayer ceramics, diodes, IC sockets, polyester and ceramic plate capacitors, transistors, electrolytic capacitors. Then the final water wash. You can now fit the five trimmers, but do not fit the three pots at this stage. The mounting of the pots requires special attention. See the next section for more details.

## Mounting the Pots, LEDs and Switch

**NOTE:** This procedure is rather different to that of the Omeg/Piher pots you may have used on older Oakley boards.

The first thing to do is to check your pot values. Spectrol do not make it that easy to spot pot values. Your pot kit should contain:

| Value | Marked as | Quantity |
|---|---|---|
| 50K linear | M248 50K M | 2 off |

Alternatively, you may have been supplied with 100K pots which would be marked as M248 100K M.

Fit the pot brackets to the pots by the nuts supplied with the pots. You should have two nuts and one washer per pot. Fit only one nut at this stage to hold the pot to the pot bracket. Make sure the pot sits more or less centrally in the pot bracket with legs pointing downwards. Tighten the nut up carefully being careful not to dislodge the pot position. I use a small pair of pliers to tighten the nut. Do not over tighten.

Now, doing one pot at a time, fit each pot and bracket into the appropriate holes in the PCB. Solder two of the pins attached to the pot bracket. Leave the other two pins and the three pins of the pot itself. Now check if the pot and bracket is lying true. That is, all four pins are through the board, and the bracket should be flat against the board's surface. If it is not, simply reheat one of the bracket's soldered pads to allow you to move the pot into the correct position. Don't leave your iron in contact with the pad for too long, this will lift the pad and the bracket will get hot. When you are happy with the location, you can solder the other two pins of the bracket and then the pot's pins. Do this for both pots and snip off any excess wire from the pot's pins at this point.

You can now present the front panel up to the completed board to check that it fits. However, I usually fit the sockets before I do this, and wire up the ground tags first. After this is done, I then fit the LED to the front panel. Then I mount the board up proper. You need to add the washer between the panel and the nut. Again, do not over tighten and be careful not to scratch your panel.

The pots shafts of these two pots will not need cutting to size. They are already at the correct length.

The Spectrol pots are lubricated with a light clear grease. This sometimes is visible along the top of the mounting bush of the pot body. Try not to touch the grease as it consequently gets onto your panel and PCB. It can be difficult to get off, although it can be removed with a little isopropyl alcohol on cotton wool bud.

The board was designed so that the LED could be soldered in correctly without using any additional flying leads. You first need to bend the legs of the LED. Hold the LED so that the flat on the bottom edge of the package is to your left and the legs are sticking out towards you. Now bend the legs downwards in a tight 90 degree bend at a point 8mm away from the bottom of the LED's body. Now take the LED and fit it into the solder side (the bottom) of the board, but do not solder it in yet. Fit the clear LED 'cliplite' clip into its hole and make sure its firmly pressed home. Present the board up to the panel and guide the LED into the cliplite. The LED's leads should sit about 2mm above the board with the bend directly above the LED's solder pads. Once you have tightened the pot nuts, you can solder the LED in. Snip of any excess wire from the component side of the PCB.

The switch mounting is simple. Fit the switch making sure one of the nuts and the spring washer go on the inside of the panel. Use the remaining switch nut on the front panel. Using solid core uninsulated wire, connect the three tags on the switch to the board. Each switch tag should go to the solder pad directly below that tag. I usually solder each wire to the switch first, letting the wire dangle into the hole below. Try to keep the wire straight. Once all three wires have been soldered neatly to the switch tags, you can then solder the other end of the wires to the board.

## Connections

This module is easy to wire up if you take your time. There are eight sockets in the suggested layout.

If you have used Switchcraft 112 sockets you will see that they have three connections. One is the earth 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. The tags are actually labelled in the plastic next to the tag. The signal lug is called 'T' for tip, the NC lug is labelled 'T/S' for tip-switched.

In this module we are going to 'common' the sockets' ground lugs. This means that the sockets' lugs are going to be joined together. I normally do this part of the wiring without the PCB or pots in place.

Fit all the sockets onto this module so that the bevel on the side of the socket is facing top right as you look at the rear of the panel.

The lugs we are connecting together will be the ground or earth tags on the two vertical columns of sockets. I use 0.91mm diameter tinned copper wire for this job. Its nice and stiff, so retains its shape. Solder a length of this solid core wire right down the three earth tags on the right most column. Trim off any excess that sticks out on either end. Then do the same on the left most column. What you have now done is common each column's earth tags together, but each column is still separate for now.

Fit the VC-LFO PCB against the front panel if you haven't done so already. Solder a piece of ordinary insulated wire to the earth lug on the socket furthest on the left on the bottom row. The other end of this wire needs to go to the pad on the PCB marked PN1. Now solder another piece of wire to the earth lug of the socket at the bottom of the right hand column. Connect this to PN2. Your earth tags are now commoned together since all the two PN pads are connected together on the PCB.

Now its time to wire up the socket's signal lugs to the board.

| ***Socket Name*** | ***PCB pad name*** |
|---|---|
| CV1 | CV |
| CV2 | KEY |

| | |
|---|---|
| SIN | SIN |
| SYNC | SYNC |
| TRI | TRI |
| SQR | SQR |
| SAW | SAW |
| RAMP | RAMP |

The final wire is to the NC tag on the CV2 pad. This should be connected to the pad on the board called GND. This is connected the module ground line and effectively shorts the CV2 input when it is not being used.

**Power connections**

The 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 GND | 2 |
| Earth/PAN | 3 |
| -15V | 4 |

The PN pads on the PCB has been provided to allow the ground tags of the jack sockets to be connected to the powers supply ground without using the module's 0V supply. Earth loops cannot occur through patch leads this way, although screening is maintained. Of course, this can only work if all your modules follow this principle.

At the rear of this user guide I have included 1:1 drawings of the suggested front panel. Actual panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £20 per panel. All you need to do is e-mail the chosen fpd file that is found on the VC-LFO web page on my site to Schaeffer, and they do the rest. The panel is black with white **engraved** legending. The panel itself is made from 3 mm thick anodised aluminium. The fpd panel can be edited with the Frontplatten Designer program available on the Schaeffer web site.

# Trimmers

There are four trimmers, or presets as we call them in the UK, on the PCB. You do not need any special equipment to set these correctly, but a scope is helpful for the STEP and a digital tuner is very useful for setting the V/octave trimmer.

The three waveshaper trimmers are situated on the bottom right hand side of the board. We will trim these first.

**OFF and SHP:** Plug the sine output into an amplifier and use your ears for this one. Set the VC-LFO to make a lowish note, around 150Hz will do. Adjust SHP until the sine output sounds pure. Its shouldn't be too buzzy or too hollow. Then go back and adjust the OFF trimmer to get it really pure. You may need to bounce back and forth between the two trimmers until you get a good sound. Its not essential to get this right, just set it so you get a nice sounding output. It's easier to do than to explain.

**STEP:** This one affects the sawtooth and ramp outputs. The easiest way to set this is to use an oscilloscope, but you can set it by ear if you haven't got one of these. If you have a 'scope, then monitor the sawtooth output. You'll probably notice a small step, or glitch, halfway down the ramp part of the wave. Set the VC-LFO to the highest frequency on the LO setting. Adjust STEP until the little glitch disappears. It will reappear when the VC-LFO is operating at high frequencies due to the slowness of the schmitt trigger, but this is fine for our purposes.

If you have no 'scope then you need to use a lowish frequency saw output (0.5Hz or so) to modulate a VCO or oscillating VCF. You can use the 1V/octave input on the VCO or VCF. You should hear a 'pee-ooh' sort of sound from the VCO or VCF as its frequency is swept downwards by the VC-LFO's output. If the STEP trimmer is set incorrectly, you'll hear a little jump as the frequency moves downward. Adjust the STEP trimmer so that the affect is minimised.

**V/OCT:** Use this to generate a perfect 1V/octave scaling. This trimmer will need to be adjusted along with the fine and coarse pots on the front panel. You will need a scope, or a digital frequency counter, or the best of all, a guitar/chromatic tuner. Some people use another keyboard or a calibrated VCO and listen to the beats, but that can take longer.

Plug your midi-CV convertor or 1V/oct keyboard into the 1V/octave input of the VC-LFO. Play a lowish note on the keyboard, then go two octaves higher. Adjust V/OCT until the interval is as close as you can to two octaves.

We are only setting the interval and not the actual frequency. It does not have to be a perfect C when C is being pressed on the keyboard. It could be an F or whatever. The key thing (groan...) is that we are setting the musical gap between the notes. If you do need to alter the pitch of the VC-LFO to help you, use the front panel controls only. Leave the TUNE trimmer until later.

For any interval, if you find the higher note is flat, then turn the V/OCT trimmer to make it flatter still. This actually reduces the range between the two notes. Conversely, if you find your interval is greater than an octave, turn the trimmer to make the top note even higher. I always adjust V/OCT on the high note of any interval, and only adjust the front panel Tune pot on the lower.

This will probably require some patience and plenty of twiddling of the front panel controls as well. But you will get there. It should be remembered that this is a VC-LFO and not a audio VCO. Thus, the circuit has been optimised for good waveform purity and not keyboard scaling. I reckon the design should be able to track over four octaves give or take a few cents. At higher frequencies, above 800Hz or so, it will go very flat. Even so, it should be good enough for most bass lines and bit beyond that.

**TUNE:** This trimmer sets the operating frequency of the VC-LFO. Turn the FREQUENCY pot to its maximum and monitor the triangle output on a scope or an amplifier. Adjust the

TUNE trimmer until you get a frequency of approximately 2kHz. If you don’t know what 2kHz sounds like, simply measure the voltage on the left hand side of R15 with a voltmeter. Adjust TUNE so that you get approximately -9V at this point.

## Final Comments

I hope you enjoy building and using the Oakley VC-LFO module. Please feel free to ask any further questions about construction or setting up.

If you cannot get your project to work, do get in touch with me, and I will see what I can do. Sometimes, it can be the simplest things that can lay out a project. I do offer a get-you-working service. Send your completed non-working module back to me with £20 and I will fix it for you. You will also have to pay for the postage both ways and any parts I have to replace. Make sure you wrap it carefully and include a full description of the fault. If you are sending the item from outside the EU, then be sure to say on the customs label 'item being sent for repair only'.

Occasionally, there may be an error in the parts list. I have checked the documentation again and again, but experience has taught me to expect some little error to creep past. The schematic is always the correct version, since the parts list is taken from the schematic. So if there is any problem, use the schematic as the guide. If you do notice any error, please get in touch.

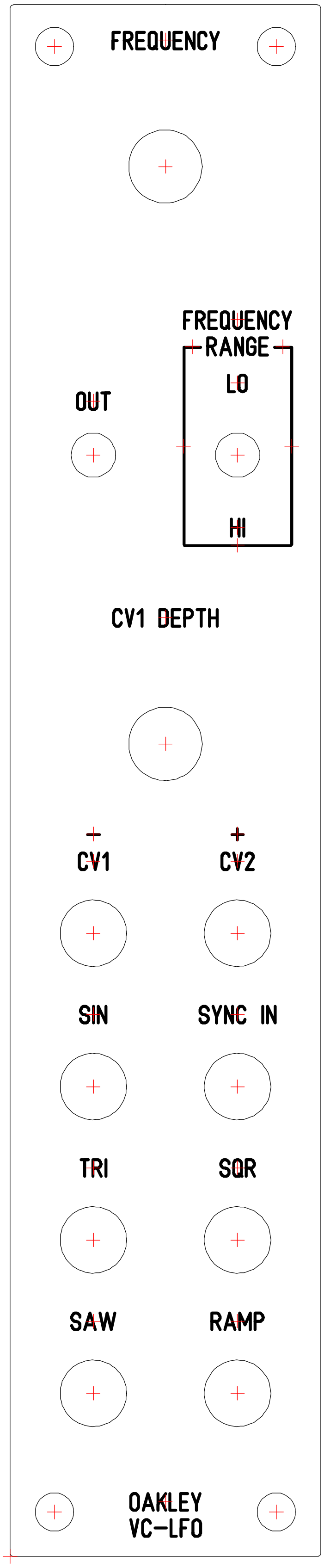
Please further any comments and questions back to me, your suggestions really do count. If you have any suggestions for new projects, feel free to contact me. You can e-mail, write or telephone me. If you telephone then it is best to do this on Monday to Friday, between 9 am and 6 pm, British time.

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, Oakley-Synths and MOTM mailing lists.

Tony Allgood.

May 2004

Version. 1.1
Formatted on Lotus Word Pro


FREQUENCY
FREQUENCY
RANGE
LO
OUT
HI
CV1 DEPTH
+
CV1
+
CV2
SIN
SYNC IN
TRI
SQR
SAW
RAMP
OAKLEY
VC–LFO