

# **Oakley Sound Systems**

**One of Three**

## **Voltage Controlled Oscillator Module**

**PCB Issue 4**

**User's Guide**

**V4.4**

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

## Introduction

**This User Guide is for the Issue 4 board set only. Check that you have an Issue 4 board. The issue number is written at the bottom left hand corner of the main board**

The Oakley 'One of Three' is a voltage controlled oscillator module that was inspired in part by the VCO design of the later MiniMoogs. In this module I decided to create a VCO that sounded good and performed well enough to keep up with the digital oscillators of modern day synthesisers.

This is the fourth issue of this popular project and module. The new issue mainly contains mechanical updates and still features the same powerful analogue sound that won the last three issues much acclaim.

The VCO features sawtooth, pulse, triangle and a low distortion sine output. The duty cycle of the pulse output may be controlled directly by a PCB mounted pot, and also an external voltage, the sensitivity being controlled by another PCB mounted pot. The frequency of the VCO can be controlled by two PCB mounted pots, one for fine, the other for coarse adjustment. These four pots are designed to be fitted to the main board, and when special pot brackets are used, the PCB can be firmly supported to the front panel.

The output levels for the sawtooth, sine and triangle are standardised at +/-5V, ie. 10V peak to peak.

The pulse wave output is also 10V peak to peak, but the peak levels vary with pulse width. The Oakley VCO uses an interesting technique to maintain the average voltage over one cycle to zero volts. This new circuit essentially adds an offset to the pulse output to compensate for the non zero average voltage for any pulse wave that isn't a square wave. Ordinarily, one would see average DC values varying from +5V to -5V as the PW is swept from one end to the other. In the new VCO, this average level is kept at zero. This means that for narrow pulses, you now have a wave that goes from just below 0V [down] to just below +10V [up]. For square waves, you have the usual +5V up and -5V down. For wide pulses, you have a wave that is just below 0V [up] and just below -10V [down]. The reason for this is that fast pulse width modulation no longer adds thumping to the audio output. ie. fast EG sweeps of PW will sound great. Although, no switch is provided to turn this feature off, you can simply omit one resistor and it will function like other VCOs. All outputs have an output impedance of roughly 1K.

The pulse output is also switchable between 'centre modulated' and 'edge modulated' forms of pulse width modulation. Most VCOs will only offer edge modulated, in which only one edge of the pulse wave is affected by the pulse width pot or modulating CV. The Oakley VCO allows you to modulate either one edge (edge) or both (centre). In fixed pulse width applications this generally makes no audible difference. However, when used with fast moving modulating CVs the difference in timbre is apparent.

A high impedance synchronisation input is provided to prematurely reset the VCO waveform. With this you can force the VCO's operating frequency to that of an external sawtooth signal, say from another VCO module. This input is level sensitive, so hard sync is possible with inputs of about +3V or above. Inputs below this will cause only occasional synchronisation leading to interesting harmonic structures.

Please note; that to cause hard sync effects the master 'sync' signal must be a sawtooth (falling ramp shape) waveform of 10V peak to peak. The slave VCO will not sync to a ramp waveform which typically has a rising ramp and does not feature the fast rising edge the VCO needs to lock on

to. These ramp waveforms are erroneously called sawtooth by some manufacturers. However, a simple inverting circuit will be sufficient to turn the waveform around the right way so it can be used as a sync master.

The VCO supports the standard 1V/octave exponential voltage to frequency relationship. However, a linear control input is also provided for constant depth frequency modulation.

Temperature compensation is performed by using a matched transistor pair in the exponential convertor, and a high quality temperature sensitive resistor.

## The Oakley CV/gate bussing

You really should think about it if you have a medium sized Oakley system. Using the Dizzy board (also available from Oakley Modular!), it allows Keyboard CV and Gate signals to be piped around the back of the modular's case along with the power supply rails. Any VCO and VCF can be connected to the KEY-CV line, and this will save you having to patch KEY-CV to every module that needs it. Inserting any patch lead into the 1V/OCT socket will override the CV bus line connection. The gate signals are treated similarly to the CV line but for use with the ADSRs and other EG modules.

The Oakley CV/gate buss uses a common three way 0.1" header to carry the two signal lines. A third, as yet unused connection is also present for future expansion.

The new VCO issue supports the Oakley CV/gate buss natively. Previous issues had the CV buss being connected to the module via a wire tail attached to the 1V/octave socket's normally closed (NC) lug. The new socket board features an optional three way header that can be fitted to allow direct connection to the CV/gate buss on an installed Oakley Dizzy system. If not required the header can simply be omitted altogether and a solder link put in its place. Or, the header can be fitted and a simple two way jumper, like those used on computer motherboards, can be fitted to allow to the buss to be implemented at a later date.

## The new issue PCB set

The main PCB is 108 (height) x 143 (depth) mm in size. All three boards use double sided copper traces and have through plated holes. The solder pads are large and are easy to solder and de-solder if necessary. They have a high quality solder mask on both sides for easier soldering, and have clear legending on the component side for easier building.

If you are building the standard design there are no components mounted off the boards. All components including sockets and pots are soldered directly to the boards.

Previously, many Oakley modules have had the sockets, switches and extra pots wired to the board by individual wires. This module allows all the socket wiring to be done via the socket PCB and two MTA solderless or Molex connections. If you are building this module in the standard Oakley format this new system will reduce assembly time and possible wiring errors.

Some people will wish to use this Oakley design in a non standard format, such as fitting it to another manufacturer's rack or one of their own invention. This is perfectly easy to do. Simply do not use the socket board and wire the main board to the sockets as per usual.

I have provided space for four of the control pots on the main PCB, whilst the other three pots are fitted to their own board. If you use the specified pots and brackets, the PCBs can be held firmly to the panel without any additional mounting procedures. The pot spacing is 1.625" and is the same as vertical spacing of the MOTM modular synthesiser.

There are detailed instructions later in the document about how to build the boards. The whole project takes around 3 hours to build and test.

## Oakley VCO Issue Changes

For those interested the Oakley VCO has evolved slowly over the last six years:

**Issue 1:** The start; the VCO is based primarily on the third series Minimoog VCO with plenty of additions. eg. in built 10V reference, sync, different exponential convertor and summing stage, sinewave output, pulse width selection, output buffering, Omeg pots.

Temperature compensation was generally provided by a +3000ppm/K 900mW film resistor. This was later upgraded to the KRL wirewound types for last ten or so boards that were sold.

**Issue 2:** Boards had the following features added:

Added -10V reference to go with the +10V internal reference voltage already on issue 1. This stabilises absolute pitch drift so that supply rail potential doesn't affect the pitch of the VCO. All pitch pots use the +/-10V references.

The range of the TUNE trimmer was reduced and this was coupled with a reduction in the quiescent operating frequency.

Added SYNC input buffer circuit. This has the advantage of providing a high impedance input. The old issue had a 10K input resistance which caused a slight drop in amplitude to the other (master) VCO connected to the SYNC input. An additional benefit is that unwanted cross coupling between two or more slave VCOs was eradicated. With the older VCOs, any two slave VCOs tended to sync to each other as well as the master. Interesting effect though.

I swapped the positions of the power supply regulator with the CV summer circuitry. This enabled me to move the two tuning multiturn trimmers to the edge of the board. The old issue 1 board had these parts in the centre of the PCB, and they were impossible to trim when the board was fitted into the Oakley Orbital monosynth. Moving the trimmers to the edge means that we can now use side adjustable trimmers in the Orbital to facilitate easy tuning with the boards in situ. This does not affect the modular set up in any way, in that we will continue to use top adjustable trimmers as before. As it turns out, no issue 2 boards ever made it into the Orbital project.

The pitch spacing of the polyester capacitors was changed from 0.3" to 5mm. This was in line with other Oakley boards.

Added separate triangle and sine wave offset trimmers. This meant that switching to centre on the PWM source switch would not affect the mark-space ratio of the square wave output.

**Issue 3:** Boards had the following features added:

Sinewave purity increased.

Two high frequency tracking compensation networks instead of just one. The standard Sergio De Franco technique, used on the earlier issues, is now fixed to compensate only for the FET discharge time. Dave Rossum's technique for compensating for the bulk emitter resistance ( $R_{be}$ ) of the NPN pair is now added with variable depth controlled by the new HFT trimmer.

Constant zero average output voltage from the pulse wave. This is achieved by considerable changes to the older simple design.

The board has four Spectrol pots instead of three Omeg ones. This makes the board less cramped since the board is now longer. The solder pads for the jack sockets now are on standard sized solder pads. 0.1" headers are still provided for those of you who like them, although the main reason for keeping these is the Orbital. The new board layout required a new modular faceplate to be made for the module.

**Issue 4:** Boards had the following features added:

Major mechanical changes. The module is now constructed from three PCBs and features solderless interconnects. This facilitates easy construction. The board set can still be used with the issue 3 front panel design.

Spectrol 248 pots are now used throughout the module.

Change to the high frequency tracking (HFT) circuitry to include Rene Schmitz's improved take on the Rossum method of compensating for  $R_{be}$ .

Change to the way the fine tune pot's CV was handled. The single metal film 10M resistor was replaced by a simple resistor network. 10M 1% resistors are getting hard to come by these days.

Replaced the now deleted CA3080 from the sine shaping circuitry. The new board simply uses one half of an LM13700 device.

Easy to fit to the Oakley CV/gate buss using an optional three way header fitted to the new socket board.

Latest PCBs are Ni/Au finish and are RoHS compliant.

## Power Supplies

The design requires plus and minus 15V supplies. These should be adequately regulated. Although quite large perturbations in the supply will not cause any pitch changes, waveform linearity and DC offset will be affected by an unregulated supply. The current consumption is about 30mA per rail. Power is routed onto the PCB by a four way 0.156" MTA or Molex type connector. You could, of course, wire up the board by soldering on wires directly. The four pins are +15V, ground, panel, -15V. The panel connection allows you to connect the metal front panel to the power supply's ground without it sharing the modules' ground line. More about this later.

## Circuit Description

The VCO circuit can be thought of as several little subsections connected together. Lets look at these little sections separately.

The first thing to look at is the power supply. This is shown on the bottom of the second sheet of the schematics. The VCO could have been powered by +/-15V, but I wanted the VCO to sound like a minimoog. To do this, I have used the same voltage levels within the VCO core as the original. Thus we need to generate stable +10V and +5V supplies. I still used the +/-15V supply for most of the op-amps as this would not have any affect on the waveforms.

Power enters the board on a 4-way Molex connector. It is filtered initially by two small ferrite inductors, and decoupled by C20 to C23. These prevent stray high frequency pulses from entering or leaving the unit by the power supply lines. A LM723 voltage regulator IC is used to provide the 10V. This is an old IC design, some 30 years, but it is a superb device and has very low noise. The only problem is that you have to set the output manually. PSU is a trimmer to set the voltage to be exactly 10.00V. The PCB has been laid out for a cermet trimmer for long term stability.

Additional power supply decoupling is provided to each set of op-amps and ICs. This prevents any reset pulses from travelling down the power supplies and soft synching the other VCOs in your system.

To find the next part of the power supply we need to look at the first sheet of the schematics. Two sections on this page generate the stable +5V and -10V required by the VCO.

U2 (pins 5, 6, 7) provides the +5V supply, by simply buffering half the +10V rail. An OPA2277 is specified to provide a very stable output.

U3 (pins 1, 2, 3) is a simple inverter circuit that produces a very stable -10.00V supply for the tune pots and trimmers. Together with the stable +10.00V supply, this prevents any perturbations in the +/-15V supply rails from affecting the tuning of the VCO. In reality, the tolerance of the 22K resistors may mean that the output is not exactly -10.00V. This is perfectly OK, since we are not interested in the actual value so much, but the stability. Those 22K resistors are metal film and their value will not change appreciably over the years or at differing temperatures.

The VCO's pitch is determined by a variety of sources. Two 1V/octave CV inputs, and two pots on the front panel, and a trimmer to set initial frequency. U2 (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 R44 and the V/OCT trimmer. For the two 1V/octave inputs the gain is roughly -0.7. The V/OCT trimmer is adjusted to give a rise of one octave in output frequency when the keyboard input voltage goes up by 1V. The output of the CV summer is then fed to the exponential convertor via R29 and R30.

The exponential convertor is based around on half of U3 (5, 6, 7) and the matched NPN pair of U4. Its output is a current that is proportional to the exponent of the voltage applied at the base of the left hand transistor of U4. The circuit gives the VCO a sensitivity of roughly -18mV/octave, so R29 and R30 reduce the output of the CV summer to this level. However, it is worth noting that R30 is a temperature sensitive resistor. The resistance of R30 will go up with temperature at a rate of 0.33% for every degree Celsius. This should counteract the temperature effects produced by the semiconductor junctions in U4. To get the best temperature stability R30 is mounted right on top of U4. 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, U4 and U3, 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.

The op-amp method also has another bonus, it allows an additional current to be injected into the inverting pin of U3. This current will directly control the output current of the exponential convertor. R15 allows an input CV to control this current. We now have a linear frequency modulation input, whose sensitivity is set by R15. Connecting this input to the wiper of a pot, allows you to control the sensitivity of this input directly from the VCO's front panel.

Note that this input is not a true linear input control. This VCO, and most other modular VCOs, don't have that capability. What we have here is a constant modulation index input. This is linear modulation, but not linear control. The precise description of this statement is probably beyond the scope of this document, but in a nut shell it means this: The linear input will not act with a strict volt per hertz relationship. That is, for every 1V, the output frequency will not rise by a fixed amount. But it will act so that audio rate FM will create a constant depth of modulation on any given frequency controlled by the usual 1V/octave inputs. This means that the for audio rate FM effects (clangs, bells and the like) you can get very good results with the linear FM input.

R3 has been chosen to set the operating frequency of the VCO at approximately 1kHz when the voltage at the base of the left hand transistor is zero. The exponential convertor works at its most accurate when the voltage across the two bases is zero. And since the human ear is particularly sensitive to frequency changes at around 1kHz it is good to make the most accurate part of the VCO at this frequency.

The output of the exponential convertor is a current. It is this current that controls the core of the oscillator. Contrary to many people's ideas, the core of a VCO is typically a linear CCO. That is a current controlled oscillator. A doubling of current to (or from in this case) the CCO will produce a doubling of output frequency.

The core of this VCO is a traditional sawtooth oscillator. C14 is the timing capacitor that is charged by the output of the exponential convertor. Since one end of C14 is tied to +5V, the voltage at the other side of C14 moves towards zero volts as current is 'sucked' out of it. The higher the current the faster the voltage drops. This voltage is 'sniffed' by one half of dual op-amp U5, which produces a replica of this voltage at its output. Q4 aids the output of U5, as well as providing a suitable offset for the triangle generator circuit (see later).

U7 (pins 1, 2, 3) is a comparator, and it is 'watching' the output of U5. When the voltage across C14 reaches -5V, the output of U5 (pin1), normally at -15V, suddenly flies upward towards 10V. This sudden level change passes through C13, and turns on JFET, Q3. This rapidly shorts out C14, and the voltage across the capacitor drops to zero and both of its pins are at +5V once again. This means that the voltage at the input of U8 is also at +5V, and the charging process begins again. C10 controls the time that the FET is on.

The sync input enables an external voltage to trip the comparator early. This will cause premature shorting of the capacitor, locking the fundamental of the VCO to an incoming external sawtooth signal. However, the fall time of the VCO's saw waveform will still be set by the input CV to the

VCO. Thus, you will not get traditional sawtooth waveforms, but half completed sawteeth. The sonic affects of these are marvellous, especially if you sweep your VCO with an envelope generator whilst locked to another VCO at fixed frequency. The controlled VCO is often called the 'slave', while the fixed VCO is called the 'master'.

The incoming sync signal is first buffered by Q1, a simple emitter follower. This is shown on the second page of the schematics. R11, C1 and C2 provide decoupling to prevent power supply noise from accidental triggering of the VCO. The buffered signal is then passed to a simple differentiator based around C9 (back on page one again). This part in conjunction with D3 and R24 only allow only fast moving rising edges of the input waveform to reset the VCO core. In theory this should allow any pulse wave to be used as sync waveforms, but the best sync sounds can be obtained when a falling sawtooth master signal is used.

This last statement is very important if you want to use sync effectively. Use sawtooth outputs from Oakley VCOs to sync other Oakley VCOs. MOTM VCOs produce rising ramps not falling sawtooth waveforms. So you if you want to use a MOTM VCO as the master to sync the slave Oakley VCO, then you must invert the MOTM's output first. You could use an Oakley MultiMix (PCB currently available from Oakley Modular) to do the inversion.

The suggested layout of the Oakley VCO includes a depth control for the sync input. This will allow you to create partial syncing effects. This effect is very difficult to describe in words and has to be heard to be believed. But very very complex timbres can be produced this way.

At high frequencies the VCO can go a little flat due to the finite time it takes to reset C14, and errors in the exponential convertor. By lowering the maximum peak voltage of the sawtooth waveform, the capacitor has less charge to loose before the output of U5 reaches zero. Thus the frequency is higher than it would have been. Using a resistance in series with C14, we have a voltage drop developed across the resistor that increases with frequency, due to the increased current through the exponential convertor. This method was first postulated by Sergio De Franco, and is usually called 'De Franco' compensation because of this. This works very well, but it does mean that the amplitude of the sawtooth waveform decreases slightly with increasing frequency. In the first two VCO issues I made this resistor variable. I was using the De Franco resistance to compensate for both reset time and errors in the exponential convertor. In the issue 4 VCO, this resistance is fixed by R37, and is compensating for reset time only.

The exponential convertor causes errors due to 'bulk emitter resistance' or  $R_{be}$  in the NPN pair. The SSM2210 is a very good matched pair with low values of  $R_{be}$ , but it still has an effect at very high frequencies of the VCO.  $R_{be}$  is not a real resistor, but it acts as if a resistor was connected in series with the emitter of the transistor.

In the early issue VCO boards, this loss of high frequency performance was cancelled out by the effect of the De Franco resistor. But really, this is a bit of a bodge since the De Franco is designed to compensate for a fixed timing error (the FET reset time), whilst  $R_{be}$  causes an error related to frequency. Over the limited range of an audio VCO, this is usually perfectly OK and De Franco is used in many commercial synth VCOs as the only high frequency compensation technique. For issue three I decided to go one better. The newer issue VCOs use a fixed De Franco resistor for the reset time, and also use the Rossum technique to compensate for the exponential generators error.

Dave Rossum, founder of E-Mu systems in the 1970s, was an important player in the development of the SSM chips. The SSM2030 VCO chip, which was not as good in comparison to the later Curtis CEM3340, was groundbreaking when it came out. Dave's method of HF compensation for the SSM2030 involved gently pulling the base of the first transistor of the NPN pair lower when the

collector current through the second NPN gets bigger.

In the issue 3 VCO I slavishly copied Dave's idea as presented in Electronotes and indeed the Prophet V many years ago. However, due to work done recently by René Schmitz this simple method can be improved upon with the same number of components. As in the original Rossum idea a fixed emitter resistor, R17, can be used to effectively measure the emitter current, since the voltage on its bottom end will fall as collector current rises. Previously we took this voltage, passed it through a diode to compensate for the base-emitter voltage drop ( $V_{be}$ ) of the matched pair and then fed a small proportion of the resultant voltage back to the base of the left hand transistor. This works well enough.

However, René's idea improves on this on two counts. Firstly, the diode in the pure Rossum method doesn't exactly compensate for the  $V_{be}$  because the current flowing in the diode and the base-emitter junction are different. In the new circuit, both semiconductors see the same current as they are fed from identical resistive sources; R17 and the HFT trimmer are both the same value. Therefore, the voltage drop across D2 and  $V_{be}$  should track each other reasonably well, even over a change in ambient temperature. Secondly, the new method feeds the base of the left hand transistor with more or less the same resistance. That is the extra resistance due to the HFT circuitry the left hand transistor base 'sees' is pretty much fixed at the value of R20 only. Thus altering the HFT trimmer should not result in a change in the overall operating frequency of the VCO as it did in the old method.

Ideally the diode should be matched with the transistor array. However, HFT compensation due to  $R_{be}$  is only a small consideration at audio frequencies so we need not get too bogged down with the higher orders of errors.

Going back to the VCO core: The sawtooth output from U5/Q2 is amplified by the other half of U5 (pins 5, 6, 7), before being sent to the output pad. The final output is roughly +5V to -5V, ie. 10V p-p.

The triangle shaper is essentially a full wave rectifier, whose operating point is about 2.5V. If the operating point is not exactly half the peak value of the sawtooth output of U5, then the triangle wave will have discontinuities in it. This leads to a slightly brighter or harsher sound than the perfect 'textbook' triangle waveform. SHP-T is a trimmer that allows precise enough adjustment in the operating point to eliminate these glitches.

The created rectified wave is filtered before amplifying and level shifting by U6. The SYM-T trimmer sets the DC offset of the wave output to match that of the sawtooth output. The final output is again 10V p-p.

The sine wave shaper circuit, shown on page two of the schematics, acts upon the triangle wave output. The rounded peaks of the sinewave are created by deliberately overdriving the inputs of one half of U8, a dual operational transconductance amplifier (OTA). These devices are normally used as VCAs, but in this VCO it is merely 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 R32 and R48. SHP-S adjusts the amount of overdrive. Since U8 gives a current output, it must be turned into a voltage and this is done by U9 (pins 5, 6, 7).

R18 provides a special negative feedback path within the shaper circuit. This increases the non-linearity of the whole stage at the critical peaks of the output waveform.

SYM-S 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 for musical uses it is adequate.

The final output, and the final circuit block is the pulse conditioner. This is also found on page two of the schematics. The first section of this is the CV summer and is based around U9 (pins 1, 2, 3). This circuit simply sums together the PWM (pulse width modulation) CV input and the voltage from the wiper of the Width pot. The output of the CV summer will go from around -5V to +5V when being controlled by the Width pot alone.

The pulse wave generator itself is based around the second half of the comparator, U7 (pins 5, 6 & 7). This compares either the triangle wave or sawtooth wave output with a voltage set by the output of the CV summer. If the waveform is higher than the voltage from the CV summer, the comparator's output goes low. If it is lower, the output goes high. Thus, the output is either low or high, and spends very little time in between. This creates a rectangular waveform, where the proportion of time spent high or low is controlled by the WIDTH pot and/or the external CV. If the width pot is set to its middle position, and no input CV is applied, the output waveform should be a square wave. The output of the comparator swings from -15V to ground (or zero volts).

The output of the comparator is out of phase with the audio input. Q2 inverts this signal, so that the generated pulse wave will go high when the saw or triangle wave is high. It would be possible to generate the correct phase by simply wiring the comparator's inputs pins differently. Thus you wouldn't have to invert the output since it would already be the right way around. Earlier issues of the VCO, in fact, did this. However, this caused a less than perfect output waveform. The positive feedback provided by R40, which is needed for good comparator action, would cause the sawtooth or triangle wave to be superposed to the output waveform. This was considered not to be a problem, but with the new issue I decided to do things differently. And perhaps more correctly.

What follows after the comparator is, I think, new to modular VCO design. The inverted output of the comparator is now summed with two voltages in U9 (pins 12, 13 & 14). One comes from the +15V supply via R74. The other is the output of the CV summer. The former allows the -15V/0V output swing of the comparator to be normalised between +5V and -5V. The other allows the output to be offset against zero volts as the pulse width is varied. When the CV summer has zero volts at its output, the generated pulse waveform is a square wave. No additional offset is added and the waveform from U9's output moves between +5V and -5V. As the pulse width changes, then the output of the CV summer moves away from zero. This is added to the waveform and the pulse output waveform moves above or below zero volts.

Analysing this dynamically changing output reveals that the average output voltage is actually always zero. This is because the offset added compensates for the altering amounts of time the waveform spends in the high (or low) states.

The summing circuit also inverts the waveform, so once again, we create the wrong phase. A simple op-amp inverter circuit, U9 (pins 9, 10 & 11) turns the pulse wave the right way round again.

If you do not want this dynamically varying output signal, then all you need to do is omit R71. The source of the audio source for the comparator comes from either the triangle or sawtooth waveforms. This selection is another unique feature of the Oakley 'One of three' VCO. Both will sound the same with fixed pulse widths. But, they do sound different when the width CV is modulated quickly.

## Components

Most of the parts are easily available from your local parts stockist. Rapid Electronics, RS Components, Maplin and Farnell, are popular here in the UK. The VCO Module was designed to be built mainly from parts available from Rapid Electronics. Rapid's telephone number in the UK is 01206 751166. They offer a traditional 'paper' catalogue, on-line ordering service, and take VISA card orders over the telephone.

In North America, companies called Mouser, Newark and Digikey are very popular. In Germany, try Reichelt, and in the Nordic countries you can use Elfa. All companies have websites with their name in the URL.

The resistors can be 5% carbon 0.25W types except where stated. However, I would go for 1% 0.25W metal film resistors throughout, since these are very cheap nowadays. If you do want to use 5% types, the please note that some of the resistors **have to** be 1% or better types. Failure to use good quality parts in these locations will affect the tuning stability of the VCO. These critical parts are indicated in the parts list.

Some of you may want to use 0.1% precision resistors in these critical positions. These are quite pricey and may be best to use in the most critical positions only. These would be: R32, R44, R29. It is my opinion that this is not worth doing and that 1% types are perfectly adequate.

All the electrolytics (abbreviated to 'elect' in the parts list) should be 25V or 35V, except where stated, and radially mounted. Don't chose too high a voltage. The higher the working voltage the larger in size the capacitor. A 220V capacitor will be too big to fit on the board. 35V is a good value to go for.

The pitch spacing of all the non-polar capacitors is 5mm (0.2"). This may differ from some of the older Oakley boards you have built.

The polyester capacitors are actually metallised polyester box types. These have a lead pitch of 5mm (0.2") and come in little plastic boxes in a variety of colours. Typical working voltages will be 63V, 100V or even 400V. They key thing to look out for is physical size and lead pitch. Most of Rapid's are either 63V and 100V and all will fit on the board.

The ceramic capacitors should be 'low-K' ceramic plates. These are sometimes called COG types. The lead spacing is 0.2" or 5mm. Do not chose cheap and nasty ceramic types, usually 'high-K', obtainable from some surplus places. These can lead to a noisy audio output. Alternatively, you can use polyester parts in place of the larger values of ceramics.

C14 is a close tolerance (2% or better) polystyrene capacitor or better. The PCB is laid out for EXFS/HR series made by LCR. Farnell and others sell these. They are good components at a good price.

The matched pair, SSM2210P, can be difficult to trace, although Farnell and RS-Ltd do carry stock. You can use the LM394 or MAT-02 instead. You could also match your own pair from discrete transistors, but thermal matching is very hard to do.

U2 and U3 are both good dual op-amps. The OPA-2277 from Burr-Brown is to be recommended. Other alternatives include the OP270 from AD or the LT1013 from Linear Technology. I have used the audio grade OP275 with success. I guess you could use a LF412, but they do drift more with temperature.

The LM1458 is a real oldie, and some places don't stock it any more. In the UK, Farnell and RS sell the little chap for peanuts.

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

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; TL072CP. Do not use SMD, SM or surface mount packages.

The BC550 or BC549 devices are discrete low noise transistors. Quite often you see an A, B or C suffix used, eg. BC549C. This letter depicts the gain or grade of the transistor (actually  $h_{fe}$  of the device). The 'One of three' VCO is designed to work with any grade device although I have used BC550 throughout in my prototypes.

The positive temperature coefficient (PTC) resistor is worthy of a mention. Meggitt make a +3000ppm/K type that is a very good part. It is available from Farnell and the part number is 732-278. However, the actual theoretical value of compensation should be closer to around +3500ppm/K when used in the circuit that I have chosen. The Meggitt part thus falls slightly short of this, and may cause the VCO to lose 'V/oct' tracking over the full keyboard range unless the ambient temperature is pretty stable.

It is therefore preferable to get something closer to the ideal value. The ones I recommend are 1W wirewound resistors with a nominal TC of +3500ppm/K. You can buy these from Oakley Modular, KRL-Bantry in the US, or from Senso's Vintage Planet ([www.senso.dds.nl/index.html](http://www.senso.dds.nl/index.html)). These are sold as equivalents to the old Tel Labs Q81 series. Once fitted to the VCOs, they give better tuning stability and allow the modular to 'warm up' quicker. They are also physically larger than the Meggitt type which means they are more stable in air drafts too.

The PTC resistor is fitted to the board so that it lies across the top of the SSM2210. A bit of thermal compound (eg. Thermpath) could be used to keep the two devices in contact. You'll need to carefully bend the legs of the TC to make sure it lies flat on the surface of the 2210 and still fits through its solder pads.

The JFET is the J112, this is very common and is the same FET die as the original MiniMoog. But take note, the Mini used the E112, and that has a different pin out. Some US customers have found the J112 quite difficult to trace. However, it can be found at Radio Shack.

The board mounted pots are Spectrol 248 series with Oakley pot brackets. You could use any 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.

Oakley Modular carry a stock of all the pots you need as the VCO pot kit. One thing to note if you are buying knobs for these pots. The older Oakley modules used Omeg pots which come with a 6mm shaft, while the Spectrol ones have a 6.35mm shaft.

The toggle switch is a standard SPDT (single pole, double throw) switch. The types I use are made by APEM in France, and have a flat toggle. RS-Components Ltd (320-916) and Farnell (1082301) sell them in this country at a very good price.

L1 to L2 are leaded ferrite beads. These are little axial components that look like little blackened

resistors. They are available from most of the mail order suppliers. Find them in the EMC or Inductor section of the catalogues. Farnell sell them as part number: 9526820 (latest RoHS compliant component).

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 four trimmers, SYM-S, SYM-T, SHP-S and SHP-T, are standard sealed carbon units. These are adjusted from the top and, as such, are called horizontally mounted types. Piher and other companies make suitable types. Lead spacing is 0.2" for the track ends, and the wiper is 0.4" above a line joining the other two pads. I use Iskra ones available from Rapid.

The PSU trimmer is a special single turn cermet type. This type of trimmer has a different PCB footprint than the other types. Lead spacing is 0.2" for the track ends, and the wiper is 0.2" away above a line joining the other two pads.

The eight way jumper interconnect is a one piece assembly bought ready made from several places. I buy mine as an eight way 55 mm or 85mm long piece. These come as pre-stripped and even tinned with solder too. Make sure you get the 0.1" (2.54mm) pitch variety. The Rapid part number is 22-1650 for the 55mm and 22-1655 for the 85mm one.

For the 0.1" interconnections I use the 26 awg MTA parts. These are made by Amp, now part of the massive Tyco empire. To use these effectively you need a special tool to poke the wires into the special ' housings ' The housing contains specially shaped contacts that cut through the insulation of the wire so you don't need to do any stripping. Just simply push down on the wire with the tool to lock it into place in the housing.

You can also use Molex strip and crimp systems. More about this one later on.

<i>Name</i>	<i>Farnell Part number</i>	<i>Amp's part number</i>
Four way housing	1098712	640442-4
Four way header	588-593	640456-4
Five way housing	1098713	640442-5
Five way header	588-600	640456-5
Six way housing	1098714	640442-6
Six way header	588-611	640456-6
Handtool	589-494	59803-1

Input and output sockets are now board mounted in the standard module assembly. You could of course use any types if you would like to wire your sockets up with individual wires.

I use the Switchcraft 112A 1/4" sockets in all my ready made modules. These excellent parts are also used on the Moog and MOTM modulars. The version you need to fit in the socket board is the 112APC. This part is stocked by most suppliers. Both Rapid and Farnell sell is at a reasonable price. Rapid also sell a Far Eastern clone of the 112APC part that is considerably cheaper, but

doesn't look as nice. Please note that the standard 112A will not fit easily into the boards as it has solder tags.

UK builders should know that there is now a 'Oakley Preferred Parts List' online. This gives the part codes of our most used parts. It can greatly speed up ordering times. This can be found at [www.oakleysound.com/parts.pdf](http://www.oakleysound.com/parts.pdf).

Sometimes people like to substitute parts in place of my own recommendations. Feel free to do this, but remember that there is normally a good reason why I have selected that particular part. If you do find that, say changing an op-amp with another one, makes an improvement, please do let me know either via the Oakley-Synths list or directly to me.

## VCO issue 4 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, 6n8 is a 6.8 nF capacitor.

### Resistors

All 5% carbon 1/4W or better. Components designated as 1% should be 0.25W 1% metal film or better.

22R	R50, 42, 41, 60, 7, 75
100R	R11
390R	R19, 23
470R 1%	R37
1K	R55, 62, 24, 56, 39, 67
1K +3500ppm/K Temp co.	R30
2K	R17, 25
2K2	R5
2K2 1%	R22
5K1	R68
5K6	R65
6K2	R59
6K2 1%	R21
8K2	R16, 69
10K	R80, 78, 27, 70, 66, 48, 46, 49, 35, 10, 34
10K 1%	R2, 57, 58
18K	R79
22K	R82, 73, 9, 71
22K 1%	R4, 6
33K	R72, 77, 76
39K	R47
39K 1%	R29
47K	R12, 33, 54
56K	R63
62K	R18, 81
68K	R74
68K 1%	R44
100K	R36, 38, 13, 15, 61, 52, 51
100K 1%	R1, 32
150K	R64
180K 1%	R28
220K	R8
270K	R53
390K	R14, 20
390K 1%	R40
680K 1%	R3, 45
750K	R83
1M	R26, 84, 43
1M 1%	R31

## Capacitors

100nF 63V polyester	C11, 12, 2, 23, 22
100pF Low-K ceramic	C13, 16
10uF, 25V elect	C3
18pF Low-K ceramic	C10
1nF 100V polyester	C4
1nF 1% polystyrene	C14
220pF low-K ceramic	C9
22uF, 25V elect	C15, 5, 7, 17, 6, 19, 24, 26, 1
4u7, 35V elect	C18, 20, 21
470pF low-K ceramic	C8, 25

## Discrete Semiconductors

1N4148 signal diode	D2, 4
BAT42 Schottky diode	D3
BC550 NPN transistor	Q1, 2, 4, 5 (You can use the BC549 too)
J112 J-FET	Q3

Note: there is no D1.

## Integrated Circuits

LM1458 dual bi-polar op-amp	U6
LM13700 transconductance amp	U8
LM2903 dual comparator	U7
LM723 voltage regulator	U1
OPA2277 low drift dual op-amp	U2, U3
SSM2210P matched NPN pair	U4
TL074 quad bi-fet op-amp	U9
TL072 dual bi-fet op-amp	U5

## Trimmers

100K multiturn trimmer	TUNE
10K multiturn trimmer	V/OCT
2K multiturn trimmer	HFT
1K horizontal carbon trimmer	SHP-T
1K cermet trimmer	PSU
22K horizontal carbon trimmer	SHP-S
47K horizontal carbon trimmer	SYM-S, SYM-T

## Pots

50K Linear Spectrol 248 pot	DEPTH, WIDTH, TUNE-2, TUNE-1 on main board
Oakley/Spectrol pot brackets	EXP, LIN, SYNC on pot board
Spectrol nuts	7 off for the above
	7 off for the above

## Miscellaneous

SPDT toggle switch	PWM type	1 off
4-way 0.156" MTA header	PWR	1 off
3-way 0.1" header	Oakley Buss optional connector	1 off
4-way 0.1" MTA header	MAIN-2 (Main PCB and socket PCB)	2 off
4-way 0.1" MTA housing	MAIN-2 cable	2 off
5-way 0.1" MTA header	POTS (Pot PCB and socket PCB)	2 off
5-way 0.1" MTA housing	NSE cable	2 off
6-way 0.1" MTA header	MAIN-1 (Main PCB and socket PCB)	2 off
6-way 0.1" MTA housing	MAIN-1 cable	2 off
8-way jumper lead	LINK interconnect	1 off
Leaded ferrite beads	L1, L2	

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 two 14-pin DIL sockets, and seven 8-pin DIL sockets.

## Populating the Main Circuit Board

Oakley Modular PCBs are now supplied with the RoHS compliant Ni/Au finish. This is a high quality finish but does possess slightly different soldering characteristics to the traditional lead based HASL finish. Handle the boards with care, and avoid touching the Ni/Au that can cause premature tarnishing of the finish. Shelf life is hard to predict but we recommend soldering in all the components less than one year from when you receive your board.

These boards can be soldered with either leaded or lead free solder. However, you should be aware that lead solder is toxic. Always wash your hands after handling solder and never put solder, or any products containing solder, in your mouth.

**We are not responsible for any accidents caused whilst working on these boards. It is up to you to use your board responsibly and sensibly.**

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!

For construction of the PCB I use water washable flux in solder. The quality of results is

remarkable. In Europe, Farnell sell Multicore's Hydro-X, a very good value water based product. You must wash the PCB at least once an hour 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. 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 better 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' 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 25W iron with a Variac power supply running at 205V. 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 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. When all the diodes are in place, double check all are pointing the right way.

The polyester film capacitors are like little blue 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 ceramic capacitors are strange flat plates made from pot. Be careful with these and make sure you have bought the ones with 0.2" lead spacing. Forcing the smaller 0.1" ones into these larger pads will break them. Another thing to watch out for is the identification markings on these capacitors. For example n47 is actually 470pF.

The smaller 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, 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.

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. And that any water is thoroughly dried out before you power up. Sockets harbour little pools of water in their pins which can lead to some very odd effects. Several sharp taps face down onto a towel will loosen off any remaining globules of liquid.

The transistors are all in the same type of packaging and therefore look the same. Only the numbers on the side will allow you to tell them apart. 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 parallel to the flat side of the transistor.

There are three types of trimmer used in this project. In two of these types we are actually using two different values. Look very carefully at each one before soldering them into place, they are very difficult to remove if you put them in the wrong place. If your multiturn pot is marked as 104, then this is a 100 000 ohm or 100K pot. 103 is a 10K pot. These multiturn pots can go in any way around.

The last part, except for the pots, you will probably put on is the temp co resistor. This is to be fitted so that it lies across the top of U4. The leads of the resistor should be bent so that it goes in the pads provided. Put a small blob of heatsink compound on the top of U4. Then push the resistor onto it. Clean up any squeezed heatsink compound from around the resistor, and make sure none of it goes on the pins of U4. Now you can solder the resistor in place.

The 0.1" headers are fitted on the main board so that pin 1 is towards the right hand side of the board. The clip on the side of the header will match with the picture on the PCB legend and should be towards the top of the board.

The 0.156" header should go into the board so that the plastic lock is on the left hand side.

I would make the main circuit board in the following order: resistors, IC sockets, small non-polar capacitors, diodes, transistors, electrolytic capacitors, and connectors. Then the final water wash.

You can now fit the trimmers using no clean or ordinary rosin flux solder.

Do not fit the pots at this stage. The mounting of the pots requires special attention. This will be covered later in this User Guide.

## Populating the Socket Board

You have one socket board to populate and the method is a little unusual.

On the board the first things to solder are the headers. These are fitted to the **BOTTOM** of the board and are soldered from the top side. This is obviously opposite to what you are normally used to. The legending is on the top of the board, and the bottom of the board is marked as such in

copper on the underside.

Fit both the headers so that pin 1 is the square pin. The friction lock on the header should correspond to the legend on the top, ie. the opposite side, of the board.

If you are intending to fit the Oakley CV/gate buss to your synthesiser system then you will need to fit a three way header in the location CV-BUSS. This is to be fitted on the bottom of the board like all the other headers. If you have no intention of ever using the CV/gate buss then you should link out pins 1 and 2 of the three way header with a piece of wire. You can use a resistor lead clipping to do this job.

It is possible to fit the header and then use a two way jumper to temporarily link pins 1 and 2 together. This way you can add the Oakley Buss at a later date very easily.

The sockets will be fitted on the top of the board, and therefore be soldered on the bottom of the board. You may well find your own way of soldering the sockets, but the way I do it is as follows:

Fit all your sockets into one of the boards. The bevel edge should align with the picture on the board legending. Do not solder them at this stage. Take your front panel and align this over the sockets.

Now carefully place your front panel with PCB and sockets upside down onto your bench [or kitchen table!]. The holes where the sockets will be should hang over the edge of the bench so that the sockets aren't forced back up through the holes. You'll also probably need a small counter weight to stop the panel from falling over the edge. Now allowing the PCB to rest flat on top of the sockets, you can begin to solder all the pins to the board.

Once all the sockets are soldered, you now need to turn your attention to the toggle switch. Now, it would be great if we could have a PCB mounted switch, but we haven't and as far as I can see nobody makes one quite like what we need. So we have to kludge a little here. All the same, you'll still end up with a nice neat solution.

Remove the socket PCB from the panel and fit your switch in the correct hole. Use the one of the nuts and the spring washer on the inside of the panel. Use the second nut to secure the switch to the panel. It doesn't matter which way the switch goes in so long as the toggle moves up and down. You do not have to use the other washer that the switch kit comes with. Now solder a short length of uninsulated solid core wire to each of the three tangs. You can use some resistor lead clippings if you wish. Make sure the wire leads stick up at right angles from the panel surface.

Now refit the socket PCB with all of its sockets back onto the front panel. Carefully align the three wires through the three solder pads in the PCB. Now you can then flip the panel over and fit all the sockets' washers and nuts. Once this is done, you can solder the three switch leads onto the PCB. Clip off any excess lead length that sticks out above the solder.

## Mounting the Board Mounted Pots

**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:

<b>Value</b>	<b>Marked as</b>	<b>Quantity</b>	<b>Location</b>
50K linear	M248 50K M	4 off	Main PCB
50K linear	M248 50K M	3 off	Pot board

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. Leave off the washer for later. 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 PCBs. 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.

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 a cotton wool bud.

You can now present the front panel up to the completed boards to check that they fit. If it does, I then mount the boards 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.

Note that the pot shafts of the pots will not need cutting to size. They are already at the correct length.

## Connections

The first thing that will need doing is the interconnection between the pot and main board. This is done with a 55mm or 85mm long 8-way jumper connection. Simply thread all eight wires into six holes of the LINK pads from the underside of the pot board. The other end of the jumper can then be gently coerced into the LINK pads of the main board. Solder each end of the jumper. You must solder the pot board from the top of that board, and the main board from the bottom of that board.

That's it that the two boards joined up. Wow, that was easy!

If you are using the recommended MTA interconnections this section will be very easy too. All the wiring between the sockets and the main board is done with one 4-way jumper, one 5-way jumper and one 6-way jumper. Here you will be using either the MTA system or the slower, but cheaper, Molex system.

Make up the 4-way interconnect first. This should be made from wires 100 mm long. Make sure you get pin 1 going to pin 1 on the other housing, pin 2 to pin 2, etc. This cable will connect to the

headers called MAIN-2 on each board.

The second lead is a 5-way interconnect. This is made up to be 145 mm long. This should connect the POTS headers on the socket board and the pot board.

The third lead is a 6-way interconnect. This is made up to be 90mm long. This will connect the MAIN-1 headers on the main and socket boards.

You can use some cable ties to bundle the wires in each interconnect together. This is best done once all the interconnects are in position.

## The Molex Alternative to MTA

For those of you who want to use the cheaper Molex system, the following information may be useful:

A quick note on the female plugs; these are sometimes called housings, since they aren't plugs themselves but merely housings for the individual crimp terminals. Terminals have to be bought in packs of one hundreds, but this is OK, because they are not expensive. These are normally designed to be crimped but they can be easily soldered with care.

Make each wire the correct length. I normally strip back the wire by just 2 to 3mm. Place all the bare wire into the crimp on a heatproof surface. I use 12mm MDF board to protect my bench top, which although not at all burn proof will take plenty of heat from a soldering iron without major damage. Rest a pair of pliers on top of the wire to hold it in place. Slip the crimp under the wire, so that the wire's insulation butts up to the edge of the terminal. Then solder in place. Sometimes I find I need to gently squash the crimp part of the terminal so that it will fit into the housing. This is easier to do before you solder it, although it can be done after with care.

Do not use the water washable flux solder in this application. You must use either good old fashioned ersin or rosin flux based solders or the newer so called 'no-clean' types. I actually prefer the rosin based ones for this because I find they flow better. Once you have soldered it, wait a bit for it to cool, and then push it into the housing until it clicks. If it doesn't go in, then take it out and bend the crimp slightly backwards. Now try again.

## 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.

<i>Power</i>	<i>Pin number</i>
+15V	1
Module GND	2
Earth/PAN	3
-15V	4

The PAN pad 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 modules 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.

## Front Panel

At the rear of this user guide I have included a 1:1 drawing of the suggested front panel layout. Actual panels can be obtained from Schaeffer of Berlin, Germany, or Front Panel Express in the US. The cost is about 20GBP per panel. All you need to do is e-mail the appropriate fpd file that is found on the VCO web page on Oakley Sound site to Schaeffer or FPE, and they do the rest.

The database panel is black with white **engraved** legending. The panel itself is made from 3mm thick anodised aluminium. The fpd panel can be edited with the Frontplatten Designer program available on the Schaeffer or FPE web site.

The pot spacing on the PCB is equivalent to the vertical spacing of the MOTM series of modules, but the socket and horizontal spacings are very different.

## Trimmers

In the UK we call them 'presets'. The first thing you need is a trimmer tool. Screwdrivers are a pain, especially with multiturn trimmers. Spectrol and others make trimmer adjusters for less than a pound. But don't use the adjuster as a normal screwdriver, you'll break it.

There are quite a few trimmers on the PCB. Lets run through them all in turn:

**PSU:** Set this to give 10.00V with respect to local ground at TP1.

Before your adjust the V/OCT trimmer you need to make sure the effects of the HFT trimmer are minimised. To do this, switch the module off and measure the resistance between the HFT trimmer's pins 2 and 3. Pin 1 is the square one, and pin 2 is the middle one. Turn the trimmer until you get the lowest value you meter will read. It'll probably go right down to zero ohms.

The other trimmers are best set up in the modular with your chosen power supply. It is also a good idea to let it warm up for at least fifteen minutes.

**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.

Now comes the tricky bit. Plug your midi-CV convertor or 1V/oct keyboard into the 1V/octave input of the VCO. Play a lowish note on the keyboard, then go two octaves higher. Adjust V/OCT until the interval is EXACTLY 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 VCO 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. Once you get the hang of it, its easy. I can do it in about one minute but I've had a lot of practice.

Now leave it on for 30 minutes, and then check the scaling. Adjust if necessary.

**HFT:** Turning the HFT will affect the pitch across the whole range of the VCO, but just like the V/OCT trimmer you need to make sure you can get one octave spread across one octave of your keyboard. This time though we are **superfine** tuning the interval at the top end of your keyboard only. If you only have a small keyboard use the Coarse pot to get the VCO really high. Anything over 2kHz (high whistle) is fine. Again ignore pitch, its the interval we are wanting to get right. Once you have set up the perfect octave, then check down at the lower end that everything is still responding to 1V/octave. It is? Good. If it isn't, go back and repeat the adjustments above.

Remember, if you have skimped on the V/OCT trimming, no amount of tweaking of the HFT will get it to play in tune.

**TUNE:** This sets the range over which your VCO acts. Set the Coarse pot to its minimum setting, and set the Fine tune pot to its middle position. Play the lowest note on your keyboard. Now adjust the TUNE trimmer until the note that you can hear is equivalent to your lowest note minus three semitones or so. That is if your bottom note is a C, then adjust TUNE so that you get a A. Now use your Coarse Tune pot to set the correct note. You may want to compare it to another keyboard module, to make sure that you have set it to the right octave.

**SHP-T:** Play an A an octave above middle C. Adjust SHP-T so that the triangle output sounds smooth. You will know when it is right. If you have a scope, adjust SHP-T so that the sharp steps disappear from the output.

**SYM-T:** Set the PWM TYPE switch to EDGE. Listen to the pulse output at any frequency that's comfortable to listen to. Now, turn the PULSE WIDTH pot to give you a square wave. This will be somewhere near the middle of its travel. A square wave sounds hollower and less buzzy than the pulse wave, so it should be easy to find. Now set the PWM type switch to CENTRE. It'll probably change the tone of the output signal. Adjust the SYM-T trimmer so that you can again hear the square wave. Flick between the two modes to confirm that you have a square wave in each switch position.

**SYM-S and SHP-S:** Plug the sine output into an amplifier and use your ears for this one. Set the VCO to make a lowish note, around 200Hz will do. Adjust SHAPE until the sine output sounds pure. Its shouldn't be too buzzy or too hollow. Then go back and adjust the SYM-S 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.

Now you have set up your VCO, and you are ready to go.

## Final Comments

I hope you enjoy building and using the Issue 4 “One of Three” VCO module.

If you have any problems with the module, an excellent source of support is the Oakley-Synths Group that can be found at <http://launch.groups.yahoo.com/group/oakley-synths/>

If you can't get your project to work, then Oakley Sound Systems are able to offer a “get you working” service. If you wish to take up this service please email 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 postal costs, any parts used and my time at 20GBP per hour. Most faults can be found and fixed within one hour. The minimum charge is 20GBP plus return postage costs.

Your comments and questions are important to both Oakley Sound and Oakley Modular.. In the first instance, please use the Oakley Synths Group (address above) where a wealth of experience resides. Please do not contact me or Oakley Modular directly with questions about sourcing components or general fault finding.

Last but not least, can I say a big thank you to all of you who helped and inspired me. Thanks especially to Jürgen Haible, René Schmitz and Ian Fritz for exponential convertor advice, Andy Hartley who built the second ever ‘One of three’ oh so many years ago, and Steve Ridley for ‘the perfect VCO’.

Tony Allgood  
Cumbria, UK January 2007

Version 4.4

Formatted on Open Office 2.0

Errors will always occur in the preparation of a document. Please forward any errors found to me so I can correct them.

Copyright belongs to Tony Allgood. No part of this document may be copied by any means for whatever purpose other than to build this unit.

