Oakley Sound Systems

Triple VCA Module

Board Issue 2

User's Guide

V2.3

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

Introduction

The Oakley Triple VCA features three separate linear direct coupled VCAs. Unlike the original 'Triple Basic VCA' project, this one is built on one single PCB. Construction is therefore faster and the finished product is neater.

The VCA (Voltage controlled Amplifier) is device used to control the level of one signal by the application of another. Traditionally, the *controlled* input to the VCA is called the INPUT, whilst the *controller* input is called the CV, or control voltage. A typical system will have the input as the audio output from a filter or oscillator, and the CV from an envelope generator. As the envelope generator's output voltage rises and falls, so the output of the VCA becomes louder and softer. The term *amplifier* is actually slightly different to the one you normally use. The gain actually varies from nearly zero, i.e. the VCA is closed or off, to about two, i.e. the output is double the input voltage.

The triple VCA features two pots per VCA channel. One controls the sensitivity of the CV, and the other sets the initial gain. The latter is used to partially open the VCA. Any positive CV applied will then open the VCA further, while a negative CV will actually cause the VCA to close.

A good VCA will be able to amplify any input including other CVs, as well as audio inputs. This means that the signal path must be direct coupled or DC coupled. No capacitors must be in the main signal path. You could use this VCA to adjust the output level of a low frequency oscillator. The CV for the VCA could be derived from the aftertouch output of a midi-CV convertor. Thus when the keys are pressed harder, you could introduce vibrato to the synth's main oscillators.

This VCA has a linear response. This means that doubling the CV will double the output amplitude. In general this is the most useful response, although other types are available from other designs. With the gain pot at its minimum, and the CV pot at its maximum. +5V at the CV input will produce a gain of around one.

The PCB is 108mm (high) x 108mm (deep). The three **Gain** pots can be board mounted for easier construction.

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.

Of Pots and Power

There are three control pots that are directly mounted on the PCB. The pots are the **Gain** controls for each VCA. 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 other three pots are mounted off the board and are connected via flying leads to the board and that channel's CV socket.

The new issue VCA board features Spectrol 248 pots instead of the Omeg pots previously used. The new pots 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 30mA 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 will run from a +/-12V supply with a slight reduction in dynamic range.

Circuit Description

The Triple VCA circuit consists of three identical VCA sections, and a common section.

Looking at the bottom right quarter of the schematic is the common section. This includes the power supply inlet, MOLEX POWER 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 nine 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.

Since all three VCA sections are the same, let use look at just one of them in detail. The top right one seems a good place to start.

The signal that is to be controlled enters the VCA by the pad named IN1. The signal is reduced, or ‘potted down’ to about 82% of its original value by R8 and R7. This is to prevent overloading of the op-amp’s input. TL072, in common with many other op-amps, tend to do very odd things when their input pins get pulled towards the supply voltages. R7 and R8 prevent this from happening.

The heart of this VCA is the LM13700 IC. 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.

Since the LM13700 is a dual device we use only one half of U4 as the gain control element. This is part that connects to the outside world via pins 14, 13, 12 and 16.

The OTA amplifies the *differential voltage* between inverting [-] and the non-inverting [+] pins by the amount set by the Iabc. The differential voltage is quite simply the voltage between the - and the + when measured on a voltmeter. Pin 13 of the OTA is the inverting input, so any positive going voltage here will produce a negative going current at the output. Pin 14 is held at roughly ground by R28. I say 'roughly' because we deliberately add a small offset voltage to pin 14 through the trimmer OFF1 and R27. This offset is very small, millivolts, but it is necessary to compensate for unwanted imbalances in the LM13700's input stage. If not corrected, the output of the VCA will have a small copy of the input CV at the output. One can never compensate for it completely, but we can do our best. Why do we want to get rid of it? Imagine a standard VCA whose job is to control the final output of a synthesiser. The input comes from a filter, and the CV comes from an envelope generator. If the envelope is very fast, a badly trimmed VCA will produce an audible click every time a note is pressed. Most people find this objectionable. As an aside, there is a late 70's Edgar Froese album ('Stuntman'), which has a synth rattling away in a sequence constantly clicking every time a note is triggered. It sends me mad every time I hear it. Edgar get that synth serviced now... oh, you sold it to get a DX-7. Oh well ...

But hang on a minute... why are there two OTAs in this circuit? This is where I take my hat off to Mike Sims who published a little touted VCA design in EDN magazine in 1995. His circuit proposed the use the other half of a dual OTA to provide a pre-distortion network.

A simple OTA like the CA3080 will become very distorted if the differential input signal exceeds 8mV or so. The knock on effect of noise and Iabc breakthrough, as well as the inherent distortion, is sufficient to render this device pretty useless for high quality VCA applications. Another OTA, the LM13700, provides a built in lineariser in the shape of a forward biased diode. This diode distorts the input in the opposite way to the OTA input stage. This cancels out some of the distortion, and although useful, it still not that great for your final VCA stage. The BA6110 and CA3280 go a step further by providing a more complex form of pre-distortion that works well enough for most of us. However the BA6110 is now deleted [shame on you R-Ohm!] and the CA3280 is expensive. But the 3280 is what I would have had to use if it weren't for a little playing around with Mike Sims' design.

Mike's design is indeed quite excellent and to my ears and my ageing test equipment, I reckon its better then the BA6110. I made a few changes to the design but the principle of the Oakley VCA is the same as Mike's original idea.

The output current of the OTA must be turned into a voltage to make it compatible with the rest of our synth. An op-amp in transconductance mode comes into use. This is quite simply an inverting amplifier with no input resistor. Any current flowing towards pin 6 of the TL072 (U1) will be matched by a current flowing through the feedback resistor, R4. No current flows into pin 6 at all, the op-amp makes sure that both currents are always evenly matched. R4 controls just how many mA from the OTA are turned into volts at the output of the op-amp. C7 creates a simple high frequency roll-off to aid stability. A 1K resistor protects the opamp's output from abuse and unstabilising capacitive loading.

The Iabc current is created with a clever little circuit based around one half of U7, another TL072 and Q1. This circuit is called a *voltage controlled current source*. Two CVs are combined as currents at pin 6 of U7 to give the Iabc we need.

R38 sets the sensitivity for the CV input and 18K will make the overall gain of the VCA roughly unity when a +5V signal is applied to the CV input. The CV input is buffered by the other half of U7 so that the input signal is not unnecessarily burdened by the VCA circuit.

The initial gain for the VCA is set by a pot labelled GAIN. The wiper of this pot travels between -0.6V and +15V. R36 sets the sensitivity to give the VCA a gain from nothing at all, ie off, to unity. The lower end is taken to just below 0V to guarantee the VCA is turned off. D1 and R35 act as a simple -0.6V supply for all three VCA sections. R1 and C3 provide a quiet +15V source for the top voltage of the Gain pots.

The voltage controlled current source acts so that any current heading towards pin 6 of U7 is offset by another current flowing towards the emitter of Q1. And any current flowing through the emitter junction will flow out of the collector towards the OTA. Actually, this is not strictly not true as some current is required to feed the base, but this is very small and can be ignored here. C19 provides stability to the circuit at high frequencies. D2 protects Q1 from reversed biasing of the base, which is possible at some input levels and at start up. R26 provides a very important job, it limits the maximum current into the OTA’s Iabc pin. This must not exceed 2mA ever and a 12K resistor sets the limit to just over 1mA.

Sourcing your Components

Most of the parts are easily available form your local parts stockist. I use Rapid Electronics, RS Components, Maplin and Farnell, here in the UK. The Triple VCA module was designed to be built from parts obtainable from Rapid Electronics and myself. Rapid’s telephone number in the UK is 01206 751166. They offer a traditional ‘paper’ catalogue as well as an 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 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, available from Oakley Modular. Three pot brackets are required and these are provided with the ‘pot mounting kit’. Oakley Modular also sell the pots should you find it difficult to get them yourselves.

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.

The three off board pots are also Spectrol 248 pots and these are mounted onto individual little boards called ‘potchips’. These potchips allow you to connect the pots to the main board with ease. These pot chips are provided with the ‘pot mounting kit’ that Oakley Modular sell.

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 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 metallised 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) ceramics have 5mm (0.2") lead spacing. For these six ceramics 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: 952-6820

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

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 the now defunct Phillips' NE5517.

The three trimmers 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" 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.

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.

Finally, if you make a circuit change that makes the circuit better, do let us know via the Oakley Synths mailing list at Yahoo Groups.

UK builders should know that there is now a ‘Oakley Preferred Parts List’ online. This can be found at **www.oakleysound.com/parts.pdf**.

Parts List

I strongly advise you to read the 'Buying Your Components' section above before you place any order for parts.

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

Resistors 1/4W, 5% or better.

100K	R7, R39, R12, R44, R17, R49
100R	R28, R31, R34
10K	R6, R40, R45, R11, R16, R50
12K	R26, R29, R32
18K	R38, R43, R48
1K	R2, R3, R25
1M	R27, R30, R33
22R	R1
22K	R8, R13, 18
2K2	R37, R42, R47
33K	R20, R22, R24
3K3	R35
36K	R4, R9, R14
56K	R36, R41, R46
62K	R19, R21, R23

Please note: Do not fit R5, R10 and R15. These components are used in the ring modulator option.

Capacitors

100nF multilayer axial ceramic	C6, C4, C11, C24, C12, C10, C22, C23, C16, C5, C17, C18, C1, C26, C27
100pF low-K ceramic plate	C7, C8, C9
1nF, 100V polyester film	C19, C20, C21
22uF, 25V electrolytic	C2, C3
2u2, 63V electrolytic	C25, C28

470pF low-K ceramic plate	C13, C14, C15

Semiconductors

1N4148 signal diode	D1, D2, D3, D4
BC560 PNP transistor	Q1, Q2, Q3

Integrated Circuits

LM13700N dual OTA	U5, U6, U4
TL072CN dual op-amp	U1, U7, U2, U8, U3, U9

Other

4-way 0.156” Molex/MTA connector	PSU
50KA or 100KA linear variable resistor	Gain1, Gain 2, Gain3 [mounted on board]
	CV1, CV2, CV3 [mounted off board]
100K horizontal mount trimmers	OFF1, OFF2, OFF3
Leaded axial ferrite beads	L1, L2
Switchcraft 112APC 1/4” sockets	Nine of them

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

For the suggested layout you need nine 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 six 8-pin and three 16-pin DIL sockets.

Building the Triple VCA Module

Warning:

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 five 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 leaded solder. The quality of results is remarkable, although you should remember that boards made this way are not RoHS compliant and would fall foul of the law should you decide to sell your unit. 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.

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.

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 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 three 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

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	6 off

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 all three 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. Although, I usually fit the sockets at this point, and wire up the ground tags first. After this is done, I then mount the PCB to the front panel. 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 three 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.

Connections

This module is easy to wire up if you take your time. There are nine 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. There are nine sockets in total.

The lugs we are connecting together will be the ground or earth tags on the three 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 middle column of sockets. Then do 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.

Now you need to common the NC lugs of the CV input sockets. You should be able to do this with another piece of 0.9mm wire. Simply lay it across (or through) the sockets' NC lugs so that all three lugs can be soldered together. You only need to do this for this column of sockets. We are not using the NC lugs on the other two sets of sockets.

Fit the VCA 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 top 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 top of the middle column. This wire will be going to the pad PN2. Now do the same for the right hand column and connect this to PN3. Your earth tags are now commoned together since all the PN pads are connected together on the PCB.

Solder another piece of insulated wire from the 0V1 pad to the commoned NC lugs on the CV input sockets. A spare ground pad (0V2) has been provided on the PCB, but this is not used in the suggested layout.

The suggested layout uses an additional three pots for CV input scaling. All three pots are 50K or 100K linear pots. Each pot has its own input 'pad'. And each input 'pad' is a pair of solder pads mounted in a square box on the board. These boxes are labelled CV1, CV2 and CV3.

Pin 1 of the input pad is the square pin and this will carry the signal to the VCA's control circuitry. Pin 2 is a direct connection to the module's ground supply, 0V.

The PCB is mounted into the front panel so that it takes the far left position looking from the front. This is unlike most MOTM modules, but is quite common with many Oakley modules. As you would expect, each of the three CV input pots should be wired to the **topside** of the board and soldered from the bottom side.

In this module we are going to be using the Oakley 'potchip' PCBs. You need to mount the three CV input pots so that their pins face away from the board. In other words, they face the opposite way compared to the board mounted pots. Each additional panel mounted pot is provided with two nuts, one 'potchip' PCB and a washer. One of the nuts should be threaded onto the pot's bush before the pot is mounted to the panel. This acts like a spacer and creates the correct pot shaft length for the knob when fitted. The washer should sit between the front of the panel and the second nut. Do not overtighten the pot nuts.

The potchip PCBs must be soldered onto the three pot pins so that the copper trace side of the potchip board can be seen facing upwards. The reason for mounting the pot upside down will now become obvious. You are able to solder the potchip pads simply from above. Each of the three CV input pots has its own potchip PCB.

The CV input pots are wired so that the counter clockwise (CCW) pin of the pot is tied to module ground. Thus, you take pin 2 (the round one) of the appropriate input pad to the CCW pin of the relevant pot. Pin 1 of the input pad (the square one) then goes to the central pin of the pot, the wiper.

NB: The CCW pin is the right hand pin when looking from the back of the pot with the pot pins facing down.

Using some stripped and tinned insulated wire connect each potchip board to the main board. The square pad on the pot chip PCB should go to the square pad on the VCA PCB. The small round pad on the potchip PCB to the round pad on the main VCA PCB.

This job can be made easier if you buy ready made wire jumpers at 0.1” (2.5mm) pitch. These can come in various lengths and different numbers of ‘ways’. We only need a two way jumper per pot, so you can use a 12-way one and cut this into the correct size with a knife or a pair of scissors. I use Rapid part number 22-1665.

The clockwise (CW) pin of each pot will be going to the appropriate jack socket’s signal lug. This requires a longish piece of wire, and its worth doing these connections last.

Now its time to wire up the socket’s signal lugs to the board. Not all signal lugs go to the board, some of them will go to the three panel mounted pots. We will do those three later

Socket Name	**PCB pad name**
IN1	IN1
IN2	IN2
IN3	IN3
OUT1	OUT1
OUT2	OUT2
OUT3	OUT3

All pads should now be used except for the spare 0V2 pad.

Now its time to wire up the CV input sockets’ signal lugs. These will be going to the CV input pots and not to the main board directly. You need three longish pieces of wire. Each one will go from the relevant signal lug to the big round pad on the respective potchip PCB.

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 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 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 VCA 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.

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

Apply power to the unit making sure you are applying the power correctly. Check that no device is running hot. Any sign of smoke or strange smells turn off the power immediately and recheck the polarity of the power supply, and the direction of the ICs in their sockets.

The next thing to do is to make sure that each VCA is passing audio. Send an audio signal into the input socket for each VCA and make sure that its Gain pot acts as a simple volume control. The volume should change from completely off to unattenuated in a smooth fashion as you turn the pot. Moving the relevant CV pot should do nothing at this stage. You should notice a small dead zone at the start of the Gain pot’s rotation. This is to ensure that the VCA is turned fully off with the pot at its minimum value.

Introduce a little bit of CV modulation at this point to check the CV input. A simple LFO waveform like a triangle is a good start. Set the gain pot to middle, and the CV pot should control the depth of modulation. In this case, you should hear the volume of the output signal rising and falling with the LFO. With the CV pot at its maximum, the LFO should be able to turn the VCA completely off.

Make sure all VCA channels work identically. If they do you have a working module and it is now time to calibrate.

Trimmers

There are three trimmers, or presets as we call them in the UK, on the PCB. Each one trims out the offset of the OTA used for that particular VCA channel. The actual setting will depend on the characteristics of the LM13700N you have used and your power supply. If you swap the OTA chips over or change your power supply, you'll need to recalibrate the module. But don't worry this one is easy, and you don't need any special equipment at all.

Each trimmer can be dealt with identically. OFF1 deals with VCA channel 1, OFF2 with VCA 2 and OFF3 with VCA3.

Set the Gain pot to its middle position. Set the CV pot to full clockwise, ie. fully on. Listen to the audio output of the VCA by hooking up your mixer/amplifier to the output of the VCA you intend to calibrate.

Send a square wave signal from a VCO or LFO to the CV input. You should hear that signal bleeding into the your VCA's output. It'll probably be very quiet, so turn your amp up so you can hear it clearly. Now turn the trimmer until the bleed through almost disappears. It won't go completely, but you should be able to find a spot on the trimmer that minimises it.

Now repeat this for the other two VCAs.

Final Comments

I hope you enjoy building and using your Oakley Triple VCA module.

If you have any problems with the construction of this module, the first place for support is the Oakley-Synths Yahoo Group:

http://launch.groups.yahoo.com/group/oakley-synths/

If you still 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 £20 per hour. Most faults can be found and fixed within one hour. The minimum charge is £20 plus return postage costs.

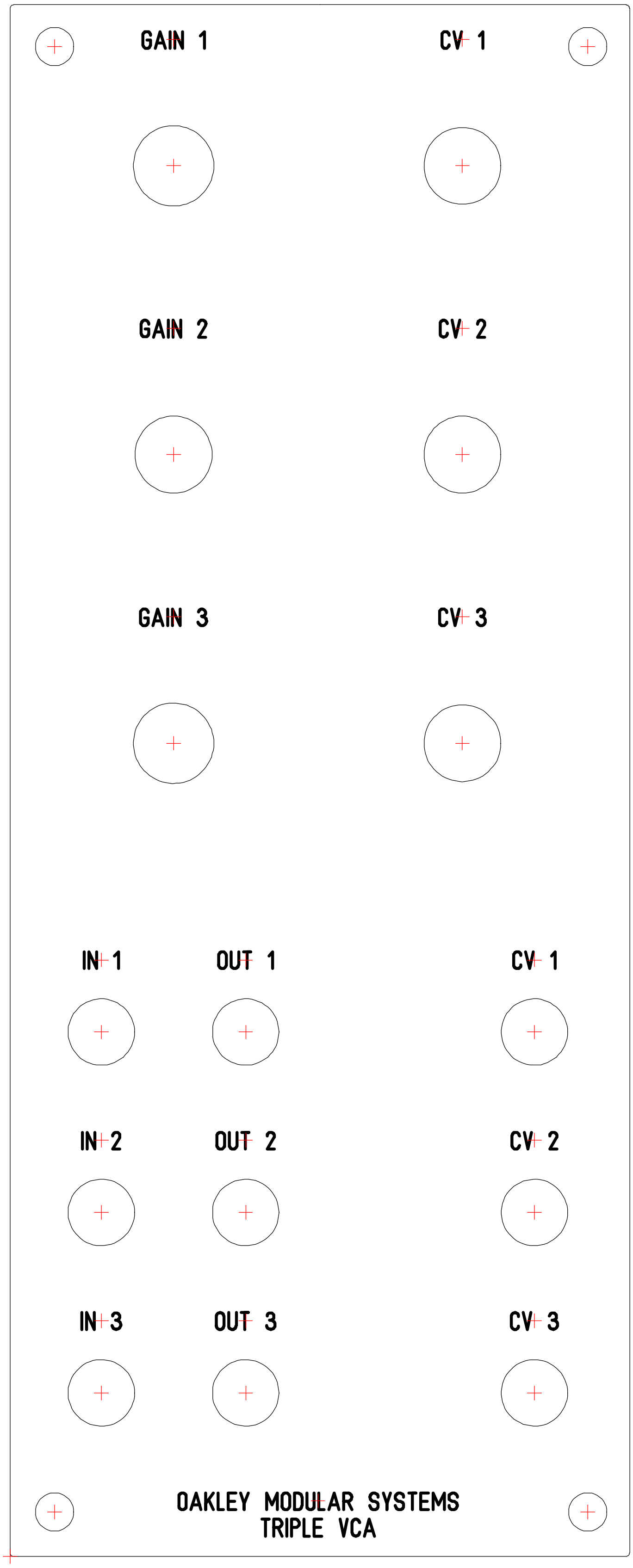
Your comments are important to both Oakley Sound and Oakley Modular. However, please do not contact me or Oakley Modular directly with questions about sourcing components or general fault finding. I would love to help but I really don't have the time these days to provide any sort of detailed customer support.

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
Cumbria, UK
February 2007

Formatted on Open Office 2.0

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


GAIN 1
CV 1
GAIN 2
CV 2
GAIN 3
CV 3
IN 1
OUT 1
CV 1
IN 2
OUT 2
CV 2
IN 3
OUT 3
CV 3
OAKLEY MODULAR SYSTEMS
TRIPLE VCA