Oakley Sound Systems

Voltage Controlled Mixer Module

User's Guide

V2.0

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

Introduction

The Oakley Voltage Controlled Mixer or VC-Mixer, for short, is a very flexible module that allows you to sum four separate signals together. Unlike a conventional mixer, the VC-Mixer allows you to set the mix levels by voltage control. The voltage controlled elements are four high quality pro-audio VCA devices made by THAT corporation. These ICs offer about the best performance outside of complex discrete designs.

This module can be built in a variety of options. The two main options are:

1. VC-Mixer in a 2U panel.

This is the ‘standard’ option. The module will act as a conventional mixer design, having four signal inputs, each controlled by its own level control. The output is thus the sum of all the input mixed together in proportions governed by the input level pots. A master output level pot is provided to set the overall gain of the module. With all pots at full, the gain of the module is approximately unity or 0dB.

Voltage control is provided by four additional sockets on the front panel. When a jack is inserted the associated level pot becomes an attenuator for the control voltage (CV) input. With the pot at full, an input voltage of 5V will give you 0dB gain on that channel. 0V will cut the signal off completely.

A peak overload LED is included, which is set to light when the mixer output levels exceed +/-10V. Also, provided is a circuit that drives an LED which can be used to give an indication of signal magnitude.

2. Quad VCA and VC-Mixer in a 3U panel.

This has all the features of the above, but has an additional four pots which act as CV attenuators. The main level pots are always prepatched to control the gain, and are not reconfigured when a jack is inserted into the CV input sockets. Thus each channel is now controlled by a single pot controlling gain and another to control the depth of CV modulation.

The additional pots are wired onto the VC-Mixer board via a 5-way 0.1” header. This can be soldered directly to all four pots if one wishes, or you can use one of the Oakley ‘Four-pot’ boards. This board is fully compatible with the VC-Mixer and as well as making the four additional pots board mounted, it will also allow each CV input to have its own reversible attenuator. The ‘four-pot’ board will make building easier and more flexible.

This User Guide will be more concerned with the 2U VC-Mixer. A separate User Guide is available for the 3U Quad VCA and Mixer module. However, references are made to both types in this User Guide to illuminate the differences between the two modules.

There can be other options too. The builder will have their own idea of how to implement them. Each voltage controlled amplifier (VCA) section is totally independent and has its own output. This means one could make a four gang module. Each VCA output may be normalised with the four channel mixer input to allow more flexibility.

Let us remind ourselves of what a VCA actually is. A VCA 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 Signal 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, ie. the VCA is closed or off, to about unity gain (0dB) at an input of 5V.

The VCAs used in this project have exponential gain control. This is different from the other VCAs you may have used in the Oakley Modular system. Those VCAs are linear, where the gain is directly proportional to the input CV level. That is, if you double the voltage to a linear VCA, the gain doubles. However, the ear and brain are not linear devices, so if you were to increase the CV input to a linear VCA slowly and in an even fashion, the perceived volume of the sound heard would not increase smoothly. The sound heard would increase in volume fast at first, and then hardly change at all. An exponential VCA however, produces a smoother change in volume when controlled by linear voltages.

The Oakley VC-Mixer will thus produce even changes in volume when a linearly changing voltage is applied to its control inputs. This makes automated fades very easy when used in conjunction with a midi-CV convertor or LFO. The VC inputs have a sensitivity of +60mV/dB, and will produce unity gain with a +5V input. Inputs in excess of 5V will produce gains of over unity should you require this, 5.6V will give a gain of +10dB for example.

The PCB is one of the biggest we sell. It is 146mm (high) x 108mm (deep). The four level pots can be board mounted for easier construction.

As with all Oakley projects the PCB is double sided with through plated holes, has a high quality solder mask both sides, and has component legending for ease of construction. The boards are electrically tested for defects.

Of Pots and Power

There are four control pots that are directly mounted on the PCB. The pots are the **Level** controls for each VCA channel. 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 pot, **Output**, is mounted off the board and is connected via a screened cable to the board. The PCB has four mounting holes, one in each corner should you require support if you are not using the pot brackets.

The board also had two LEDs which are also mounted separately from the board. Flying leads are used to connect the LEDs to the board.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 40mA 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 VC-Mixer circuit is spread over two pages. The first page shows the four channel mixer, output stage, peak and level circuitry and power supply. The second page shows the four identical VCA sections. In a break from conventionality let us start with page two first.

Since all three VCA sections are the same, we will have 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 90% of its original value by R31 and R27. This is to prevent overloading of the op-amp’s input. The TL072, in common with many other op-amps, tend to do very odd things when their input pins get pulled towards the supply voltages. R31 and R27 reduce the chance of this from happening.

The signal is now sent to the VCA core. The core is based around a THAT 2180 or 2181. These are exceptional audio VCA devices, and form part of a family of VCA that are manufactured by the THAT corporation. The VC-Mixer is designed to use any of these devices, but has been tested only with the 2181 and 2180. These parts basically do pretty much everything you need from a VCA. Used in conjunction with an op-amp output stage, they simply alter their gain depending on the voltage on their control pins. A full datasheet of either part may be downloaded from the website at http://www.thatcorp.com.

The output of U5 (pins 5, 6, 7) goes into the VCA via C13. This is an optional part. You can fit this capacitor to remove any DC component from your input signal. DC that is superimposed on your audio signal will end up being modulated by the VCA’s control voltage. This may be heard as a thump if the CV is modulated fast enough. Most Oakley modules have no appreciable DC offset, so its probably not worth fitting. C13, C2, C3 and C4 would be then replaced by wire links in this case. The advantage in not fitting these capacitors, is that you can use the VC-Mixer to control other control voltages. For example, you may want to control an LFO with a aftertouch CV before feeding it to a VCO’s pitch input. However, if you will never need to mix DC, that is, you want it for audio only, then you can fit those capacitors. In this case, make each capacitor a 1uF 63V polyester. On my ready made and pre-populated versions of this board, I do not fit the capacitors.

The THAT2181 parts are very good parts, and offer very low distortion and control voltage breakthrough. However, it is possible to trim the distortion down still further. PR1 provides this means. It is not essential to use this trimming in most applications, but it is there, should you require the finest fidelity from your signal.

If you use the THAT2180, you don’t need to trim for low distortion. The parts come pre-trimmed so the trimmer and associating 150K resistor can be omitted.

Please note that R5 must be fitted with either device and must be the value chosen. This resistor sets the operating current within the 2181 which is set to 2.4mA with a -15V supply.

U5 (pins 2,3,1) acts as a transimpedance amplifier. Simply put it converts the output current of the 2181 to a voltage. This output voltage is available at the op-amp’s output at pin 1 of U5. R6 provides the necessary isolation should the VCA output be used independently from the rest of the circuit. This isolated output is available at the square pad on the PCB marked A1.

The control signal for the VCA comes from another op-amp circuit. This is based around one half of U10 (pins 5,6,7). The sensitivity of the 2181’s control pin is just -6mV/dB. That means every 6mV drop in control voltage at pin 3 of the 2181 will cause the gain to go up by 1dB. A gain change of roughly +10dB represents a doubling in perceived volume of the output signal.

This sort of sensitivity is too high and it goes the wrong way for a modular synth application, so we use the active circuitry around U11 to make it approximately +54mV/dB. U11 provides the necessary inversion and change of gain required.

GAIN1 is a pot that controls the level of the output signal. On the 3U panel the pot’s CW pin is permanently wired to a +5.0V reference voltage, via the CV1 pad. On the 2U panel, the CV1 pad is connected to the signal lug of the CV input jack socket. The 5.0V reference voltage is passed onto that lug via the socket’s normalising contact. This means that whenever a jack plug is not inserted the 5.0V reference is connected to the pot. When a jack is inserted, the jack will supply the control voltage only.

The Gain1 pot controls the level of the VCA by dividing down the reference voltage or CV signal. The amount it divides down the signal is dependant on the position of the pot’s wiper.

The THAT chips are slightly awkward in one respect, that is they produce the most gain when their control pin is at 0mV and the minimum gain at +540mV. Now we want a control range of 0V to 5V, with +5V as the maximum gain and 0V for its minimum gain. This means not only do we have to change the sensitivity of the CV and polarity, but we also have to offset the output by exactly 540mV. This offset is created by adding a -5.0V negative reference voltage to the control voltage created by the input CV. This comes from the signal -REF on the schematic. The voltage at -REF should be an exact opposite of the positive reference voltage at the top of the pots so as to cancel out perfectly when the pot is at its maximum setting. We will see later how we actually generate these two reference signals.

U10 acts as a summing stage, inverter, and scaler in one. R67 is present to reduce the effects of power up surges damaging the THAT chip. I don’t know whether this is a bit of overkill on my part, but having that 1K resistor there makes me happier. The feedback path includes R67, so that the voltage that drives the THAT chip is not affected by the resistance of R67. R3 and C12 dampen any possible oscillations that may occur inside the THAT chip.

Lets now turn to the first page of the schematics, the first thing that I want to concentrate on is the reference voltage section. These two little circuits are about half way down on the left hand side. Both are very simple and are based around the actions of a zener diode. Given a certain amount of current a zener diode will conduct with a certain voltage across it. Now this is true of pretty much all devices, but the zener diode actually maintains this voltage for a large range of currents. In this way, we can use it as a voltage reference. We have two of these circuits here, and they are pretty identical except one makes a negative voltage and the other, positive. In each circuit a resistor provides the current from the appropriate supply rail, and a big capacitor filters any noise that may be developed across the zener diode.

In my prototype I used an ordinary 5.1V zener diode. Now ordinary zener diodes are actually not very stable voltage references as they tend to drift with temperature and age. However, we are actually interested in the difference in the absolute value of the zener voltages. So, if one of them drifts because it gets warm, the chances are, so will the other one since it is in same location. Thus small temperature effects will cancel out. Any differential between the two voltages will produce a gain change in the VCAs. An appreciable 100mV difference would produce just less than 2dB of change which just discernible. However, I know that this wouldn't keep the most discerning of my customers happy, so you have an alternative. Instead of using ordinary cheap 5.1V zeners, you can use the LM336. This is a precision 5.0V zener has exceptional characteristics and is available for less than a pound each.

For the 2U panel design, R1 is provided to limit the current surge from C1 whilst inserting the CV jack plug before the normalising contact has been broken.

At the top of page two we see the four channel mixer section built around U9 (pins 2,3,1). This simple circuit is based around the traditional inverting summing circuit and adds the four voltages that occur at M1 to M4. The M pads are situated very close the A pads on the PCB. Thus it is a simple matter to link the A and M pads together to directly connect each VCA to the mixer stage.

The gain of the mixer is set to only 0.2. This means that even if all four inputs are close to clipping, the final summed output is still not clipping or distorting. U9 is a low noise DC accurate op-amp, and functions well as an all round DC and audio amplifier.

The output of the summing stage feeds the master volume pot, or **Output** control. This pot is mounted off board, and the three pot pins are connected to the board via three solder pads. The master volume pot is a 47K linear pot. The linear track on this pot would normally give us an uneven volume increase as it was turned up, but we are using a technique called wiper loading to create a pseudo log effect. By loading the wiper with a low resistance, the actual signal at the wiper will not increase linearly with an even pot rotation. R64 acts as the loading resistance, and input resistor for the next amplification stage at the same time.

This final stage of amplification does several things. One, it amplifies the signal back up to unity gain after the loss of gain through the summing stage. Two, inverts the signal back to the correct phase after the inversion through the summing circuit. Three, provides stability against capacitive and resistive loading effects thanks to C37 and R82. The output signal is available at the solder pad named **OUT**.

C37 and R82 form part of the stabilising network designed to keep the op-amp from oscillating under capacitive loads. A full explanation of this is beyond the scope of this User guide, but more information can be found on the Analog Devices website in their Applications Note AN-257. I was first introduced to this method when I worked at Soundcraft, and have since used it on many other Oakley projects.

The output signal is then sent to the peak and level indicating circuitry. R55 and R56 reduce the signal to half of its original value before the FWR. This will reduce any loading effects on the main audio output of U9.

The signal is full wave rectified by the circuitry based around U12 (pins 5,6,7). Full wave rectification can be described by the mathematical ‘absolute’ function. In other words, the output of the full wave rectifier (FWR) is always positive. If you present +10V to the input, you will get +10V. But if you present it with -10V you will also get +10V. Likewise, -5V turns into +5V, -3V into +3V. Now if you put an audio signal into this circuit, you will get a series of positive bumps that correspond to the up and downs of the audio signal.

U12 (pins 2,3,1) forms a buffer circuit. The output of the full wave rectifier is therefore protected by the odd load presented by the next set of circuits.

A comparator circuit, based around U12 (pins 10,9,8), is used to operate the **PEAK** LED. This is normally achieved more simply using just a transistor, but I had the spare op-amp available and the results from this sort of circuit are more predictable. The LED is designed to turn on when the output of the FWR reaches around 2.8V. At 2.8V the output of U9 is just below the clipping level, at around +/-10V. An op-amp running off a +/-15V rail will be able to output around 13V maximum, so enabling the **PEAK** LED to turn on at 10V gives you just the right amount of headroom. You should normally operate the Mixer so that the peak LED only very occasionally flickers with the peaks in the music.

A visual indication of the CV outputs is available from the **AMP** LED. This is driven from a current source provided by op-amp U12 (pins 13,12,14). The LED in the feedback loop will have a current that is determined solely by the voltage presented to the end of R63. A 5V CV output, will produce 5mA in the LED. Although in normal operation the LED is always forward biased, it may be subjected to odd negative swings on power up and power down. A normal diode is placed in parallel with the LED pads to prevent damage to the LED.

Looking at the bottom right quarter of the schematic is the power supply conditioning. This includes the power supply inlet, POWER and the power supply filtering components.

The swathe of capacitors in the middle lower section of the page are the decoupling capacitors for al the op-amps on the board. The boxes in the section to the left of them show the power supply connections to the op-amps. 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.

Two grounds are provided, one for the circuit itself, and one for the earthing of the jack sockets on the front panel.

Sourcing your Components

Most of the parts are easily available from your local parts stockist. I use Rapid, RS components, Maplin and Farnell, here in the UK. In North America, companies called Mouser, Newark and Digikey are very popular. In Germany, try Reichelt. 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 pots are Omeg ECO-16 or Piher PC16 types with matching 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. In the UK, Maplin, CPC and Rapid sell the Omeg pots at a very good price. But note that Rapid or CPC don't sell the pot brackets. RS sell the Piher range. The pot kit that I supply contains all five pots and the four pot brackets.

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. Two values that may need some extra explanation are the 5K1 and 3M3. The 5K1 will probably be only available in 1% tolerance. While the 3M3, although sometimes available in 1%, it will be generally cheaper to buy these as a 5% type.

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

The pitch spacing of the 1nF and 680nF polyester capacitors is 5mm (0.2"). These common types come in little plastic boxes with legs that stick out of the bottom. They come in a variety of styles and names, but they should be described as metalised polyester types. For either value try to get those with an operating voltage of 63V or at the most 100V. Higher operating voltages can be used but will probably be too large to fit on the PCB.

The PCB is another one the later Oakley boards 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. These axial parts are best suited to a spacing of 0.3", and the board is laid out accordingly.

The low capacitance (values in pF) ceramics have 5mm (0.2") lead spacing. For these 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. In the UK RS-Components sell the former, whilst plate types can be bought from pretty much anywhere.

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

All op-amp 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. Do not use SMD, SM or surface mount packages. They do not fit at all.

U9 is a low noise, high output current audio op-amp called OP275GP. It is made by Analogue Devices and is an excellent part. You can also use a 5532, although this device has a higher offset voltage. A TL072, or LF412 can also be used, and apart from the decreased output drive, they are both good devices to use in place of the OP275.

The three audio op-amps, U5, 6, 7 and 8, are listed as TL072. These op-amps are fine for most users, although you may prefer to use a better part. The LF412 offers an improvement in DC offset. The OP275 can be used, and offers theoretical improvements in noise and DC accuracy. However, I found the part oscillated in this application when used as is. I would therefore recommend that if you do use the OP275, you do not fit R35, R36, R37 and R38 and link them out instead.

The four VCA chips are THAT 2181 or THAT2180 devices. They are both available in various grades from http://www.profusionplc.com. Each grade offers different distortion figures, the LC being the worst at a very acceptable 0.02% THD at 1KHz. Rapid Electronics sell the THAT2180LC at a very reasonable price too.

If you use the THAT2180, please note that you do not need to trim for minimum distortion. The manufacturer of the ICs have done this for you. You therefore do not need to fit R4, 9, 14 and 19, and all four of the trimmer pots.

The four trimmers, if you need to fit them, 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. 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.

I would recommend that you use IC sockets for the four VCA chips. You need four seven pin SIL (single in line) sockets. You can buy SIL sockets in large strips that you can cut down. The other alternative is to buy two 14-pin DIL low profile turned pin sockets, and cut them in half. You can then trim off the extra bits of plastic with a craft knife.

The zener diodes can be either standard 400mW types in a DO35 package, or you can use the more exclusive LM336. The latter is harder to find, but Farnell have the part in stock.

I use bipolar LEDs in all my modules now. They can be fitted in anyway round and will not be damaged. I use 5mm LEDs, with Cliplite LED holders. The LEDs are available in most places, although Rapid sell them cheapest. The cliplites are available from Maplin.

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. All of the sockets must have normalising lugs if you are building the suggested layout. The Switchcraft 112 types have normalising lugs as standard.

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

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

Parts List

A quick note on European part descriptions. To prevent loss of the small '.'as the decimal point, a convention of inserting the unit in its place is used. eg. 4R7 is a 4.7 ohm, 4K7 is a 4700 ohm resistor, 4n7 is a 4.7 nF capacitor.

Please note that there are various options available for this board. a) DC or AC coupling of the VCA sections. b) THAT2180 or THAT2181 devices. c) 2U or 3U panel versions.

NB: 3U 'Quad VCA and Mixer' builders should refer to the specific User Guide for that module.

Links

For the 2U VC-Mixer option, link out the A and M pads in pairs. That is, link out A1 and M1, A2 and M2 and so on. You can use solid core wire or a clipping from a resistor.

Resistors

Resistors 1/4W, 5% or better.

100K	R27, R69, R70, R46, R28, R29, R47, R52, R51, R30, R75, R73, R57
100R	R3, R8, R13, R18, R65
10K	R31, R35, R36, R32, R33, R37, R38, R34, R54
11K	R68, R45, R50, R76
1K	R6, R67, R44, R11, R16, R49, R21, R72, R63
1K8	R60, R2
22K	R22, R23, R7, R24, R25, R12, R17, R26, R81
22R	R61, R62
2K2	R77
33K	R66, R43, R39, R40, R41
3M3	R79
47K	R78
47R	R1
5K1	R5, R10, R15, R20, R42, R56, R55, R80
6K8	R64, R58
75R	R82
7K5	R59

For users of the THAT2181, these additional components are required:

150K	R4, R9, R14, R19

Capacitors

100nF multilayer axial ceramic	C5, C7, C9, C36, C11, C33, C34, C35, C21, C23, C25, C22, C24, C19, C18, C20
100pF low-K ceramic plate	C38, C29, C30, C39
22pF low-K ceramic plate	C14, C15, C16, C17
33pF low-K ceramic plate	C37, C31
1n5, 100V polyester film	C12, C6, C8, C10
22uF, 25V electrolytic	C40, C28, C27, C1, C41, C26
680nF, 63V polyester	C32

For optional AC coupling the following parts need to be fitted:

1uF, 63V polyester	C13, C2, C3, C4

For DC coupling you will need to link out C13, C2, C3 and C4 with short wire links.

Semiconductors

1N4148 signal diode	D2, D4, D3, D7, D6
5V1 zener diode or LM336-5	D5, D1 (see text)

Integrated Circuits

TL072CN dual op-amp	U5, U11, U10, U7, U8, U6
TL074CN quad op-amp	U12
THAT2181LC or 2180LC	U1, U2, U3, U4
OP275 audio op-amp	U9

Other

4-way 0.156” Molex/MTA connector	POWER
47KA or 100KA linear variable resistor	Gain1, Gain2, Gain3, Gain4 [mounted on board] Output [mounted off board]
Leaded axial ferrite beads	L1, L2
5mm Orange LED and clip	AMP
5mm Red LED and clip	PEAK
Switchcraft 112APC 1/4” sockets	Nine of them for 2U panel
50 cm of 0.91mm solid core wire	
Five knobs	
Power lead MTA to MTA connector	
1m of multistrand hook up wire in a variety of colours	

For users of the THAT2181, these additional components are required:

100K horizontal mount trimmers	OFF1, OFF2, OFF3

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 seven 8-pin and one 14-pin DIL sockets. You also need four 7-pin SIL sockets.

Building the VC-Mixer Module

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

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

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

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

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

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

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

If you have used ordinary 5V1 zener diodes, you can mount these as you would an ordinary diode. If you have chosen to use the LM336, then this is more complex. As you will see the LM336 looks more like a transistor than a diode. It has three legs and is in a TO92 style housing. But don’t worry it will fit OK if you follow these instructions carefully.

Take the device and hold it with the legs upwards, and the flat of the body away from you. With a pair of wire cutters, snip off the left leg. Make your cut about 2mm away from the body of the diode. Now you should have a two legged device. (Its cruel, I know, but its better this way). The middle pin of the device is the cathode, and this should be **gently** positioned into the square pad of D1. The right hand leg, the anode, should go in to the round pin of D1. Make sure that neither leg touches the stump of the snipped leg. Do the same for the other LM336 for the D5 position.

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.

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.

The ICs all have pin at the top. Pin 1 is also the one with the square solder pad. The SIL packages for the VCAs, should have a slot or a dint in the upper surface of the package. This represents pin one of the IC. As such, all the slots should be towards the top of the PCB

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 four trimmers if you are using the THAT2181. Do not fit the four pots at this stage. The mounting of the pots requires special attention. See the next section for more details.

Mounting the Pots

If you are using the recommended Omeg ECO or Piher PC16 pots, then they can support the PCB with specially manufactured pot brackets. You will not normally need any further support for the board. When constructing the board, fit the pot brackets to the pots by the nuts and washers supplied with the pots. Now fit them into the appropriate holes in the PCB. But only solder the three pins that connect to the pot. I normally solder the middle pin first and then check if the pot is lying true. If it is not, simply reheat the middle pin’s solder joint to allow you to move the pot into the correct position. **Do not solder the pot bracket at this stage.**

Now remove all the nuts and washers from the pots and fit the board up to your front panel. Refit the washers and tighten the nuts, but not too tight other wise you will deform the pot bearing. If the pot feels rough when you turn it, the chances are that you have tightened the pot nuts too tight. Normally backing off the nut a bit is enough to free the shaft. Now carefully position the PCB at right angles to the panel. The pot’s own pins will hold the PCB fairly rigid for now. Then you can solder each of the brackets. This will give you a very strong support and not stress the pot connections.

The Omeg pots are labelled A, B or C. For example: 47KB or 100KA. Omeg uses the European convention of A = Linear, B = logarithmic and C = Reverse logarithmic. So a 47KA is a 47K lin pot.

The pots shafts may be cut down with a good pair of pliers, or a junior hack saw. Try not to bend or rotate the shaft as you are cutting.

The Omeg pots are lubricated with a thick clear grease. This sometimes is visible along the screw thread 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. If I do see any grease near the top of the bush, I tend to wipe it clean with a bit of kitchen paper before I mount the pot.

Connections

There are a lot of connections in this module, but it is easy to wire up if you take your time.

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 inserted. 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’ many of the socket’s lugs. This means that many of 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 left as you look at the rear of the panel. There are nine sockets in total.

The first lugs we are connecting together will be the ground or earth tags on the two main rows of sockets. We will ignore the main output socket for now. 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 across all the earth tags on the top row. Trim off any excess that sticks out on either end. Then do the same on the lower row of sockets. What you have now done is common each row’s earth tags together, but each row is still separate for now.

Now solder another thick piece of wire across the NC lugs of the top row of sockets. Make sure you keep the wire straight, and make sure the tip of any inserted jack plug doesn’t make contact with this wire. Do the same for the lower row of sockets. You should now have four parallel pieces of wire.

Fit the 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 needs to go to the pad on the PCB marked PN1, but don’t solder it yet. Now solder another piece of wire to the socket furthest left on the bottom row. This wire will also be going to the pad PN1. The two stripped ends of the wires should fit snug into the one hole, and can be soldered in together.

The commoned NC lugs on the top row of sockets can now be connected to the pad marked GD2. Whilst, the commoned NC lugs on the lower row can now be connected to REF.

The earth lug on the MIX OUT socket should now be connected to the pad labelled PN1.

Connect, with a piece of insulated wire, the following **signal tags** on these sockets to the respective pad on the PCB.

Socket Name	*PCB pad name*
IN1	IN1
IN2	IN2
IN3	IN3
IN4	IN4
CV1	CV1
CV2	CV2
CV3	CV3
CV4	CV4
MIX OUT	OUT

All pads on the lower part of the board should now be used except for the spare GND pad called GD1. This now completes the wiring of the sockets.

The master volume pot, OUTPUT, is the next part to be wired. In this module I mount the pots so that their pins face towards the bottom of the module. The pot has three pins, CW, CCW and wiper. CW stands for clockwise and CCW for counter clockwise. The CCW pin is the right hand pin when looking from the back with the module lying on its front with the pot pins facing down. The CW, the left hand pin. The wiper is in the middle. Each pin on the pot is connected to the relevant pad on the board. You can use either three separate pieces of insulated wire, or you can use screened cable. I use narrow gauge twin cored screened cable, the screen being connected to the CCW pin and pad.

The LEDs are simple to wire up, but it is made even easier if you use bipolar LEDs. These are LEDs that are manufactured so you do need to worry about the polarity of the device. Normal LEDs need to connected the correct way around, otherwise they don't work and they may be damaged. Bipolar ones can be put in anyway around, and they always work.

I always twist the two wires that connect LEDs together. This has two purposes; one, it keeps the connections tidy, and two, it minimises any LED switching noise from reaching the audio lines. In any case, keep your LED leads away from the audio leads.

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/PN1/PN2	3
-15V	4

The PN1 and PN2 pads on the PCB has been provided to allow the ground tags of the jack sockets to be connected to the powers supply ground without using the module's 0V supply.

Earth loops cannot occur through patch leads this way, although screening is maintained. Of course, this can only work if all your modules follow this principle.

At the rear of this user guide I have included 1:1 drawings of the suggested front panels. Actual panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £25 per panel. All you need to do is e-mail the chosen fpd file that is found on the VC-Mixer 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 channel is passing audio. Send an audio signal into the input socket for each input and make sure that its LEVEL 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. You should notice a small dead zone at the start of the LEVEL pot’s rotation. This is to ensure that the VCA is turned fully off with the pot at its minimum value.

The OUTPUT pot should behave as a master volume control. Its action should be smooth. Check that the LEVEL LED gets brighter the louder the output signal. It will flicker if the input frequency is lower than 30Hz or so.

The peak LED should light up if the output level goes above +/-10V. However, since the maximum gain of the mixer is 0dB or unity, it should not light up if you are only feeding it with a VCO output.

Now connect an envelope generator (EG) to the CV1 socket. Gating the EG should now control the VC-Mixer. Remember the exponential nature of the CV input will make it respond different to normal linear VCAs; the sound should fade in and fade out a lot faster. Turning the LEVEL pot down should control the overall volume level of the signal.

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

Trimmers

Remember you do not need to trim anything if you have used the THAT2180 VCA chips. This section only applies to folks you have used the THAT2181 devices.

There are four trimmers, or presets as we call them in the UK, on the PCB. Each one trims out the offset and distortion for that particular VCA channel. The actual setting will depend on the characteristics of the THAT2181 you have used and your power supply. If you swap the VCA 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. PR1 deals with VCA channel 1, PR2 with VCA 2 and so on.

Set the LEVEL pot to its middle position and the OUTPUT pot to its maximum. Listen to the audio output of the VCA by hooking up your mixer/amplifer to the MIX OUT socket.

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 bleedthrough almost disappears. It won't go completely, but you should be able to find a spot on the trimmer that minimises it. As it happens, the minimum point for CV breakthrough is also the point at which you get the minimum distortion.

Now repeat this for the other three VCAs.

Final Comments

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

If you cannot get your project to work, do get in touch with me, and I will see what I can do. Sometimes, it can be the simplest things that can lay out a project. I do offer a ‘get you working’ service. Send your completed non-working module back to me with £25 and I will fix it for you. You will also have to pay for the postage both ways and any parts I have to replace. Make sure you wrap it carefully and include a full description of the fault. If you are sending the item from outside the EU, then be sure to say on the customs label ‘item being sent for repair only’.

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

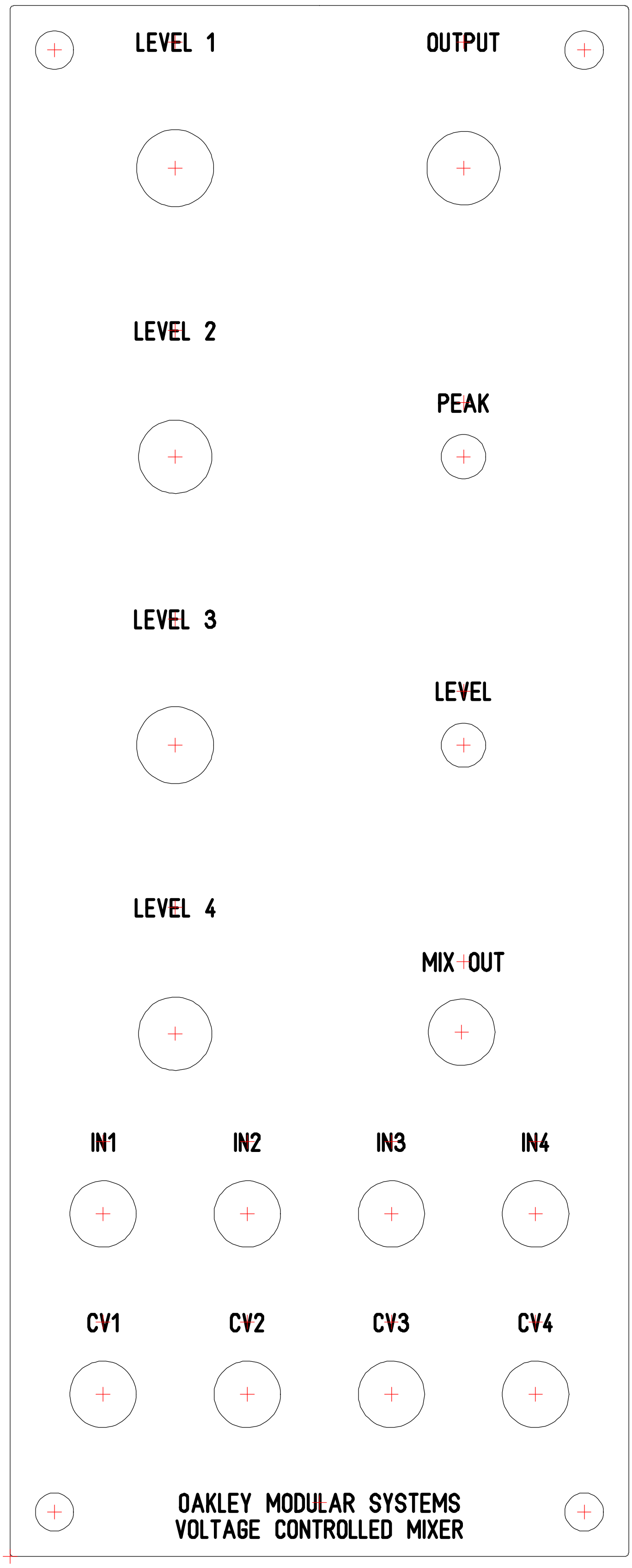
Please further any comments and questions back to me, your suggestions really do count. If you have any suggestions for new projects, feel free to contact me. You can e-mail, write or telephone me. If you telephone then it is best to do this on Monday to Friday, between 9 am and 6 pm, British time.

Last but not least, can I say a big thank you to all of you who helped and inspired me. Thanks especially to all those nice people on the synth-diy, Oakley-Synths and MOTM mailing lists.

Tony Allgood. April 2004

Version. 2.0

Formatted on Lotus Word Pro


LEVEL 1
OUTPUT
LEVEL 2
PEAK
LEVEL 3
LEVEL
LEVEL 4
MIX OUT
IN1
IN2
IN3
IN4
CV1
CV2
CV3
CV4
OAKLEY MODULAR SYSTEMS
VOLTAGE CONTROLLED MIXER