Oakley Sound Systems

5U Oakley Modular Series

VC-ADSR

PCB Issue 1

User's Guide

V1.0

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is my version of the classic envelope generator module primarily designed for use in modular synthesisers.

The envelope generator or EG for short, generates a rising and then falling voltage at its output when triggered by a gate signal. The gate is derived from a keyboard, switch or a midi-CV convertor elsewhere in the synthesiser system. The speed of the rise in output voltage is determined by the Attack control. The speed of the fall of the output voltage is determined by the Decay or Release controls. The output of an EG is traditionally patched to the VCA control voltage, to control the volume of the note when the keyboard is pressed, or to the VCF, where it dynamically alters the harmonic structure of the sound throughout the duration of the note played.

Traditionally there are three basic analogue EG types available:

Attack-decay (AD)

This is a one shot type of generator. The output will rise and then immediately fall. The A and D pots determine the time taken to rise and fall. Removal of the gate will cause the decay phase to start prematurely on most systems.

Attack-release (AR)

Very similar to the AD type, but the voltage is held high once the attack phase is completed until the gate is removed.

Attack-decay-sustain-release (ADSR)

This one is controlled by four pots. When the attack phase is initiated by the gate signal, the output will rise to a predetermined level. Then the decay phase starts and the output voltage will then fall to the level set by the sustain pot. The voltage will remain at this level for as long as the gate is high. But as soon as the gate is removed the release stage is initiated. This causes the output to fall at a rate determined by the release pot. The ADSR-EG can perform both AR and AD operations simply by turning the sustain pot full up or down respectively.

Other types of envelope generators are available, especially on digital synthesisers, but the three analogue types still seem to the most musician friendly.

The Oakley ADSR module also incorporates a versatile VCA on the output of the EG core. This part is used to control or modulate the level of the EG itself. For example, a velocity CV from the output of a midi to CV convertor, can automatically scale the ADSR output for touch sensitive control of the VCA or VCF. This VCA has an excellent audio response, so it can be used to modulate any audio input with the ADSR too. This combined module used to be called an Envelope shaper, but that term has dropped out of fashion these days. This combined module is very useful in reducing patch cords.
Unlike the original two issues of the Oakley ADSR/VCA board, the control input of the VCA is now permanently connected to the output of the envelope generator circuitry. This may

seem like a loss, but most builders built the standard design anyway. It also makes the module easier to build.

The ADSR core is now quite different to other ADSR units including my own issue 1 and 2 ADSR/VCA modules. This new module uses a high quality audio VCA chip to achieve voltage controlled time constants. In other words, the attack, decay and release times are now under voltage control. These voltages are derived directly from the front panel pots and are normally kept within the module circuitry. However, two four way headers make these four voltages available for an expansion card called the Oakley 'Four-pot' board. This additional board will allow external CVs to merge with the ADSR's pots, so full external control is available to the user. This upgrade feature may be added onto the ADSR/VCA module at any time, although requires the use of an additional 1U panel.

An LED has been fitted to this issue of the module. This lights up and its brightness follows the level of the internally generated ADSR signal. It is not affected by the CV or audio signal on the IN socket. The LED is driven from a maximum of 10mA at the attack peak.

Specifications

Minimum attack time:	0.75mS
Maximum attack time:	12 seconds
Minimum Decay/release time:	1.8mS to fall from 90% to 10% of initial value.
Maximum decay/release time:	15 seconds to fall from 90% to 10% of initial value.
Normalised output voltage**:	Attack peak at 5V from OUT+
Gate input to activate ADSR:	>3V

** No jack inserted into the IN socket.

Of Pots and Power

There are four main control pots on the PCB. The pots are **Attack**, **Decay**, **Sustain** and **Release**. If you use the specified Spectrol pots and Oakley mounting brackets, the PCB can be held firmly to the panel without any other 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 PCB has four mounting holes, one in each corner should you require additional support which you probably won’t.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 25mA. 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 current Oakley modular projects, yet is still compatible with the MOTM systems. See later for details. This unit will run from a +/-12V supply with a slight reduction in dynamic range and maximum attack and decay times.

Circuit Description

Power is applied to the board through the 4-way Molex connector, PWR. L1 and L2, small axial ferrite beads, provide some high frequency resistance, and along with C15 to C18 prevent the board from being effected by any noises on the power rails. They also help keep any noises going the other way too.

The **GATE** input requires a switch type signal that is either at around 0 volts when off, or any positive voltage greater than 3V when on. The Oakley ADSR can easily handle greater gate voltages without damage. D5 protects Q3 from any negative inputs. When a positive gate arrives, Q3 turns on and pulls its collector down to ground or 0V. R35 provides a little positive feedback to speed up switching times. R35 also allows slowly increasing CVs applied to the GATE input to trigger the envelope. For example you can use the output of another EG with a slow attack to create gate delay effects.

The inverted and beefed up version of the applied gate signal found at the collector of Q3 is sent to two destinations. Referring to the schematic, we see that one destination is another transistor, Q2. This is configured as another inverter. Thus the output of Q2 produces a copy of the gate signal that swings from 0 when off, to +15V when on. It is passed on to a CR network that acts as a differentiator. This circuit produces a positive voltage spike when the gate goes high. The duration of the spike is determined principally by the values of C13 and R23. D4 prevents a negative spike being produced when the gate goes low. The positive spike triggers an RS flip-flop circuit based around two NOR gates, U1.

A flip-flop is a sort of a one bit memory, or latch. Once triggered by a positive going pulse at pin 8, it stays latched. You can only reset it by removing the power or a reset pulse at its other input, pin 6. When the flip-flop is latched, pin 4 goes high and pin 10 goes low. Pin 4 goes to a control input on U2.

U2 is a 4052 CMOS dual 4-way switch or analogue transmission gate. This, quite simply, behaves like an electrically controlled rotary switch with four positions and two poles. You can consider each pole's operation as being a bit like walking into a room through one door and facing four others. You have to keep moving forward, and you must go through one of the doors in front of you. You can't go through two doors at once so you have to chose which one to go through. A four way switch is a bit like that. The electric current can enter the switch through the input pin and leave via any one of the four outputs. It can also do the other way round too, the switch doesn't care which direction the current flows. So you can also select which of the four 'inputs' can go to one 'output'. A two pole switch would have two identical switches, but both switches move together. That is, if you have selected position 1, then both inputs will have to go to their respective output 1. So you can't have one pole going to output 2 and the other to output 3.

Now a 4052 is an electrically controlled rotary switch and we are using it here as a two pole-three way switch. One of the four ways will go unused. It is configured so that each way of the switch are the inputs, and the switch wiper will be the output. The position of the switch is controlled by two ports, marked A and B on the schematic. The voltages, either 0V or 15V, at these points will determine which output is selected. I have separated the two poles and the control section of the 4052 into three parts on the schematic. Notice that the control part has some additional pins; these are the power supply and another unused control pin. C11 provides some local supply decoupling for U2.

The two signals that control which direction our switches are pointing are from the output of flip-flop and the processed gate signal. The switch will control whether the circuitry is in the A, D/S or R parts of the envelope's cycle. The decay and sustain parts are actually one of the same. At the start of the decay phase the output voltage of the EG is simply decaying to the sustain voltage. The sustain period merely being the final level of the decay process.

The core of the EG is a charge storage device. In other words, some sort of memory cell that we can charge up and discharge down. In a standard ADSR, this function is admirably achieved using a capacitor which is charged up and down via variable resistors. In this module we are using a more complicated circuit called a lag or slew generator. This circuit has two inputs, and one output. The first input tells the lag generator whether its output should go up or down. The second input controls the speed at which any change takes place. Each of these inputs is controlled by one pole of the four way switch. The lag generator is based around U4, a dual op-amp, and U8, a highly specialised transconductance device.

At the lag generator's heart there is a smaller circuit called an integrator. This part is made up of R39, U4 (pins 5,6,7) and C14. Just think of U8 being a piece of wire for now. You can charge it up by applying a voltage to R39, and discharge it with a voltage of the opposite polarity. The higher the voltage the faster the charging time. Apply no voltage to it, and the charge should stay where it is, and the voltage on the output of U4 will stay constant. Unfortunately, this is not quite the case. Stray currents and voltages generated by U4 tend to make the output unstable without other circuitry which we will come to later.

To make the integrator's time constant voltage controllable I have used a current output VCA in series with R39. This VCA is normally used in high quality mixing desks, but is great for this particular application too. Its three main features are:

1. Very low offset. This allows the integrator to be controlled over a very wide range without the problems of integrator error. This error voltage will be at its worse when the gain of the VCA is very small, ie. for long charge and discharge times. At low VCA gains, errors in the integrator are not able to be driven down by the feedback system. This doesn't tend to affect the attack phase, but it does affect the decay and release phases. So for very long release times, the output of the integrator will never fall to zero. Even so, the 2180LC parts I originally tried provided a worse case error voltage of only 15mV. That's small enough to be ignored in this application. However, more recently I have come across two samples of 2180LC that produced an unacceptable level of offset. But as we will see, this can be compensated for by a little modification to the original circuit.

2. Exponential voltage input. The VCA's control node controls the gain of the VCA in an exponential fashion. In other words, for every 18mV increase in voltage at pin 3, the time constant is doubled. The affect of this for ADSRs is tremendous. It allows control over the attack and decay times with superb accuracy. This is especially important in setting attack times. Now you can set any attack time between 0.75mS and 100mS with ease. This is great for making punchy bass and lead sounds without creating clicks at the start of every note.

3. Small size. In one small 8 pin SIL (single in line) pack, we get a VCA and the exponential convertor. This makes laying out the PCB much easier.

U4 is a dual op-amp, one half is used in the integrator itself, the other used as a differential amplifier. A differential amplifier is simply one that takes two inputs and subtracts them together. One of these inputs is the voltage signal coming from the ADSR logic circuitry via R14. This voltage will determine whether the integrator is charged up or down and to what level. The other input, via R12, to the differential amplifier is the output of the integrator. This is the feedback path. It provides two main purposes:

1. To reduce long term drift of the integrator output.
2. To turn the integrator into a lag generator.

Any integrator's output will continue to rise if a constant positive voltage is placed at its input. We want a different response; we need it to rise to a set value and stay there. By feeding back the output of the integrator and subtracting it from the input voltage, we can create an integrator that only charges up to that level determined by the input voltage. The functions makes sense when you throw some figures at it. Say the integrator output is at zero. If you now place a 10V signal to the input of the differential amplifier, the differential amplifier's output will be 10V. Since 10V - 0V = 10V.

This 10V is now passed to the integrator, which rises upwards at a rate determined by the VCA, U8. But as it does so, the level fed back to the differential amplifier rises too. This in turn is subtracted from the 10V input signal by the differential amplifier. Thus the amplifier output falls. When the integrator reaches 10V, the differential amplifier's output is now zero and the integrator stops rising. In fact, the speed of the rise has been slowing down all the time since the input voltage to the integrator has been falling too. So you get a nice exponential rising curve of increasing voltage.

As we have said, the lag generator has two inputs. One, the input, controls the direction of charging or discharging, and the eventual level of the output signal. The other, the control, affects the speed at which the output is allowed to change. Each of these voltages are controlled by one pole of U2, the electrically controlled switch.

Looking at the schematic we will see that the input to the integrator can be switched between one of three inputs. One is +15V via R7. This is switched in during the attack phase. The integrator will charge up, and would reach +15V if we were to let it. The decay-release phase is initiated when the second input is selected. This input is connected to the wiper of the sustain pot. The integrator will now discharge to the sustain voltage. The release cycle is initiated by the gate going low, and the third input is now selected. This is connected to ground, and the integrator now discharges to 0V.

The speed at which all these changes take place is governed by the control voltage to the VCA chip, U8. Again, three different voltages are present to allow for individual control over attack, decay and release times. The second half of U2 switches these into the control node. U3 (pins 5,6,7) buffers the voltage signal from the analogue switch. A buffer is a simple op-amp based circuit that monitors its input voltage and presents an exact copy of it to its output. The benefits of this is that the input signal does not 'feel' the affects of any loading that the output may be subjected to. In this case, the relatively low resistance of R25 and R24 would take too much current from the pots were it not for the buffer between them. The two resistors are required to cut down the 0 to 10V signals from the three timing pots to the small level required by the 2180. They have to be a low value to keep offsets in the 2180 getting too big.

In the original design, I made the control voltage at pin 3 of U8 vary between 0V and 500mV or so. 0V would produce the fastest times and 500mV the slowest. However, certain specimens of THAT2180LC produced too large an offset at very long release and decay times. I therefore added R49. This forces pin 3 to be more negative than it would have other wise been. So although the control voltage range is still 500mV, it now swings from around -150mV to 350mV. Simply put, this ensures that the VCA chip always has sufficient enough gain to compensate for any offsets in the integrator even at long release times. Unfortunately, this additional resistor does not have a pair of unique pads on the PCB. As such, you must ‘surface mount’ the resistor in the tried and tested way of industrial PCB re-workers all over the globe.

Monitoring the voltage at the output of the integrator is a comparator based around U3 (1,2,3) and Q1. When the voltage exceeds 10V or so, the comparator’s output goes from 0V to +15V. This tells the flip-flop that the attack phase is over and the **decay** phase is about to start. The latch is thus reset; pin 4 goes low and pin 3 goes high. The electronic switch, U2 changes state and the integrator is no longer charging up.

In some other EG circuits, the level at which the attack phase stops is set by the CMOS logic gate’s threshold voltage. In other words, the buffered capacitor voltage is fed directly to the flip-flop. This, in my opinion, is not very good for two reasons. One, the slowly rising voltage at the input to CMOS logic is not a good idea since it produces a large heat dissipation is the device. Secondly, it can lead to false triggering.

The benefit in having a reasonably precise comparator is that you can set the end of the attack phase to suit your own purposes. The timing capacitor is being charged from a +15V source through a resistor. This leads to the exponential rise in voltage over time. I prefer the attack phase to be nearly linear for a more punchy sound. By setting the maximum attack peak to a little below 5V, we get a very good approximation to a straight line. However, it does limit the maximum attack time. By using a 14V peak, we can get really long maximum attack times, but a very exponential response, and in my opinion very unrealistic sounds. By using a value of 10V, we get a nice compromise between punchy attack and long maximum attack times. Feel free to adjust the value of R4 which controls the peak and the maximum sustain value and do some experimenting yourself.

The decay phase is controlled by the flip-flop. Pin 4 goes low when the attack peak has been reached. But it also goes high when the gate signal is removed. This is the signal that tells us that the **release** phase has started. C12, R22 and D3 form a differentiator which apply a pulse when the gate goes low. This is combined with the attack peak detected signal to reset the flip-flop. U1 (12,13,11 & 1,2,3) acts as the OR gate which combines the signals.

When the gate drops, the envelope generator will start the release phase. This can happen at any point in the ADSR cycle. The output of Q3, the inverted gate signal will go high. This alters the selection within electronic switch U2 and the integrator is discharged to ground at the speed governed by the Release pot. The flip-flop will be reset, as we have already discussed, to prevent the attack and decay pots from affecting the timing capacitor. With no gate present, the envelope generator is continually in release mode thus keeping the capacitor always discharged.

The output of the ADSR now goes to the VCA section. This is shown on the second sheet of the schematic. It is the same basic circuit as used on the Oakley Triple VCA module. However, there are a few differences to the way we generate the Iabc current as we shall see.

The IN socket on the front of the module is the signal that is to be controlled. Its either a CV or audio, and it enters the VCA by the pad named IN. The signal is reduced, or ‘potted down’ to about 90% of its original value by R18 and R19. 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. R18 and R19 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 R33. I say ‘roughly’ because we deliberately add a small offset voltage to pin 14 through the trimmer OFF and R34. 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.

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. 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, but although useful, it still not that great for your final VCA stage. The BA6110 and CA3280 go a stage 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.

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 changes to the design but the principle of this 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, R31 and the trimmer GAIN. No current flows into pin 6 at all, as the op-amp makes sure that both currents are always evenly matched. R31 and GAIN control just how many mA from the OTA are turned into volts at the output of the op-amp. C10 creates a simple high frequency roll-off to aid stability. A 1K resistor protects the opamp’s output from abuse and unstabilising capacitive loading.

Another op-amp is wired as a simple inverting circuit. This tips the output of the VCA upside down. +5V becomes -5V. -1V becomes +1V. This is ideal in an EG so you can have negative modulation sweeps. If you are using the VCA for an audio application is has less of a use, but it should be noted that there is a phase difference of 180 degrees between the two outputs.

The Iabc current is controlled with the circuitry based around Q4. Q4 turns the output of the ADSR's integrator into an current suitable to drive the OTA. This circuit is very simple and does have a few conditions attached to it. Notice that the base is held at around 220mV by R46 and R47. This means to turn on that transistor and create any Iabc at all, you need at least 300mV at its emitter. This means that if the output of the ADSR's integrator falls below 300mV, it will be ignored. This is actually a double edged sword. Its good because the decay and release tails have a definite finish. The VCA is most certainly off when the ADSR output falls below 300mV. And offsets within the ADSR will have no effect on VCA performance since they are most certainly smaller than 300mV. But its a bad thing when you look at the attack response. Basically, the very first portion of the attack phase is ignored. However, to put this in perspective its only the first 300mV out of 10V. That's only 3% and you will not notice this in all but the longest attack times.

But, its not perfect is it? No, its not, but if it makes you feel better, most of the classic analogue synths have a similar circuit; from the humble Rogue to the big Oberheims.

R48 sets the sensitivity for the CV input and 27K will make the overall gain of the VCA roughly unity at 100% Sustain level. However, fine tuning of the VCA's gain can be done with the GAIN trimmer.

R32 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.

The last bit of active circuitry is the LED driver. This provides a visual indication of the ADSR output. The LED, I use a green one, is driven from a current source provided by op-amp U5 (pins 2,3,1). The LED in the feedback loop will have a current that is determined solely by the voltage presented to the end of R17. A 10V ADSR output, will produce 10mA 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, D1, is placed in parallel with the LED pads to prevent damage to the LED. Note, that if you use a bipolar LED any damage is impossible anyway even without the D1.

The little two resistor network at the bottom of the schematic creates a +5V signal. Available at solder pad NC, this is added to the VCA’s signal path when no jack plug is inserted into the CV input. This allows the ADSR output to be always available with a fixed attack peak of +5V even with no modulation input. Thus the envelope generator can be used as if the VCA were not present.

Components

Most of the parts are easily available from your local parts stockist. I use Rapid Electronics, RS Components, Maplin and Farnell, here in the UK. The Equinoxe was designed to be built solely from parts obtainable from Rapid Electronics and myself only. Rapid’s telephone number is 01206 751166. They offer a traditional ‘paper’ catalogue 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 pots are now Spectrol 248 series pots with 1/4” shafts. These are high quality sealed conductive plastic potentiometers. Rapid and Farnell sell these parts in the UK. The pot brackets are especially made for us, and are only available from Oakley Sound Systems. We also sell the pots should you find it difficult to get them yourselves.

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. Some values, like 5K1, are only available as 1% metal film types.

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 polyester capacitors is 5mm (0.2”). I use metalised polyester film types. These come in little plastic boxes with legs that stick out of the bottom. Try to get ones with operating voltages of 63V or 100V. You can also use multilayer ceramics for the three 1nF capacitors.

The low capacitance (values in pF) ceramics have 5mm (0.2”) lead spacing. For these two 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.

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 X7R types from Rapid.

The horizontal preset or trimmer resistor is just an ordinary carbon type. No need to buy the expensive cermet types. Carbon sealed units have more resistance to dust than the open frame types. Citec and Piher-Meggitt make a suitable type to use here. Pin spacing is 0.2” at the base, with the wiper 0.4” away from the base line.

L1 and 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: 108-267.

The BC550 and BC560 devices are discrete low noise transistors. You can replace them with BC549 and BC559 types respectively, although the voltage rating of the BC550 and BC560 is higher. Quite often you see an A, B or C suffix used, eg. BC549C. This letter depicts the gain or grade of the transistor (actually hfe of the device). The VC-ADSR is designed to work with any grade device although I have used ungraded BC550 and BC560 throughout in my prototypes.

All ICs are dual in line (DIL or DIP) packages except for the VCA. These DIL ICs are normally, 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 LM13700 is a dual operational transconductance amplifier, and is some times not to be found in the op-amp section of your parts catalogue. It may be down as ‘special’ or ‘OTA’. This part can be substituted with a LM13600 or NE5517 with no loss of performance.

The THAT2180LC is available from Rapid Electronics or www.profusionplc.com. I guess that the better 2180LB and 2180LA would offer even lower offsets should you need it.

As with most of the Oakley modular series the input and output sockets are not board mounted. You can choose what types of sockets to use. I used the excellent Switchcraft 112APC 1/4” sockets.

The LED should be a 5mm diameter bipolar LED. The LED clips I use I get from Maplin in the UK. They have a built in lens and hold the LED firmly to the front panel. For green LEDs, its best to get green lens. Now there's a surprise!

IC sockets are recommended for U1, U2 and U8. 8 pin SIL sockets can easily be made by cutting a 16 pin DIL socket in half and trimming off the rough bits.

If you want to expand your ADSR/VCA at some point then it is best to fit the two four way headers on the board now. These two headers are 0.1" pitch vertical headers with friction lock. If you are not fitting the 'Four-pot' board, then these headers will need to be shorted out. You will need four 'jumper links'. These are the small links that one sees on computer motherboards and hard disk drives. Rapid sell these jumpers as part number 22-0692. If you are not likely to be fitting the four pot board at any time, then simply link out the appropriate places on the VC-ADSR board. See the connections section for more details.

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

Parts List

This is an early issue of the documentation, I have checked the parts list, but I can miss things. If in doubt, check against the circuit diagram, this is always correct. Please e-mail me if you find any discrepancies.

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	R26, R18, R23, R38, R45, R22
100R	R24, R16, R33, R46
10K	R27, R19, R10, R9, R1, R41
12K	R3, 6, 32, 49**
15K	R31
1K	R40, R28, R17
1M	R35, R34
22K	R39, R21
27K	R48
2K2	R25
390K	R14, R15
3M3	R8
47K	R12, R13, R30, R29, R44, R11, R2
4K7	R7, R37, R36
5K1	R43, R5, R4, R42
62K	R20
6K8	R47

****R49 is a modification to the original PCB layout and does not have a legended location on the board. See the section in the User Guide: 'Building the ADSR/VCA board'.**

Capacitors

100nF multilayer ceramic	C1, C4, C5, C8, C9, C7, C15, C16, C11, C2
100pF ceramic plate	C10
1nF, 63V polyester	C3, C12, C13
2u2, 63V electrolytic	C17, C18
220nF, 63V polyester	C14
470pF ceramic plate	C6

Discrete Semiconductors

1N4148 silicon signal diode	D1, D4, D3, D2, D5
BC550 NPN transistor	Q3, Q2, Q1
BC560 PNP	Q4
LED 5mm green bipolar	LED

Integrated Circuit Semiconductors

LM13700 dual OTA	U7
4001 Quad NOR gate	U1
4052 DP4T analogue switch	U2
THAT2180LC audio VCA	U8
TL072 dual FET op-amp	U6, U3, U5, U4

Other

4-way 0.156” Molex/MTA connector	PWR
4-way 0.1" vertical header	A-D, A-R
Jumper links	Four off (see text)
10K linear single gang variable resistor	SUSTAIN
50K linear single gang variable resistor	ATTACK, DECAY, RELEASE
Four Oakley-Spectrol pot brackets to suit	
10K carbon trimmer (horizontal)	GAIN
100K carbon trimmer (horizontal)	OFF
Leaded or taped ferrite beads	L1, L2
1m of multistrand hook up wire	
Four knobs	
Four decent quality jack sockets, eg. Switchcraft 112	

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, one 16 pin, and four 8 pin DIL sockets. You also need one 8 pin SIL socket.

Building the ADSR/VCA Board

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 use 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 traditional uncontrolled 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.

R49 is a modification to the original circuit design. This part has been added to ensure very low offset in the integrator. Some specimens of the THAT2180LC were not quite as good as I expected and this modification compensates for that. R49 is a 12K resistor and should be soldered from the right hand pad of R24 to the top pad of C15. The resistor can be soldered onto the top surface or the underside of the board. However, I would recommend that you solder it to the top side of the board so that lies flat on the board.

It will then lie parallel to the bottom edge of the board, ie. horizontally. If done neatly, it will should be directly on top of the legend U8.

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.

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 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.

The polyester capacitors are like little blue or red 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 0.2" pitch ceramic plates need to be treated with a little respect. Don't bend them to much once you have soldered them in. Do trim down the leads with wire cutters, even if they don't have that much to chop off.

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.

All the ICs have pin 1 to the top of the board. This also applies to the THAT2180. Be extra careful with this part, pin 1 is denoted by a chink in the top of the casing. This should go to the top of the board. Pin 1 is also depicted by a square solder pad.

I would make the board in the following order: resistors, IC sockets, small non-polar capacitors, transistors, electrolytic capacitors. Then the final water wash. You can now fit the trimmer to the board with no-clean or ordinary ‘ersin’ flux solder. Do not fit the pots or the LED at this stage. The mounting of the pots and the LED requires special attention. See the next section for more details.

Mounting the Spectrol Pots and LED

NOTE: This procedure is rather different to that of the Omeg 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	3 off
10K linear	M248 10K M	1 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 itself. Do this for all four pots.

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. The washers should go on the pot’s bush at the front of the module and the second nut on top of this. Again, do not over tighten.

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

The pots are lubricated with a light 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.

The LED should be able to be soldered directly into the board if its leads are long enough. You’ll have to bend the leads at ninety degrees near the body of the LED. It doesn’t matter which lead goes into which hole of the LED pad since we are using bipolar LEDs.

If your LED does not have sufficiently long leads to reach to board from the panel hole, then you may have to wire it to the board with some small pieces of insulated wire. Keep the wires as short as possible without being taut. Use a little heatshrink tubing to insulate the LED’s leads from rubbing together.

Connections

This module is very easy to connect up. There are just four 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 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’ 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 left as you look at the rear of the panel. There are just four sockets in total.

The first lugs we are connecting together will be the ground or earth tags on the two horizontal rows 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 across the two 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.

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 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 furthest left on the bottom row. This wire will be going to the pad PN2. Your earth tags are now commoned together.

Connect, with four pieces of insulated wire, each signal tag to the respective pad on the PCB. The pads that are going to be connected are OUT, INV, GATE and IN. I have used slightly different names for the front panel sockets. The table below shows which is connected to which:

PCB	**Front Panel**	**Socket Connection**
IN	‘IN’	Signal lug
NC	‘IN’	NC lug
OUT+	‘OUT+’	Signal lug
OUT-	‘OUT-’	Signal lug
GATE	‘GATE’	Signal lug

Use small lengths of insulated wire to make your connections. There is no need to use screened cable.

Leave the NC tags unconnected on the GATE, OUT+ and OUT- sockets. Now with another piece of insulated wire connect the NC tag on the IN socket to the NORM pad on the PCB. This will allow the ADSR outputs to function even without any CV input.

If you wish to use the Oakley CV-gate normalising system, you will want to add your gate input to the GATE socket's NC lug.

The PN1 and PN2 pads have 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. For a suitable power distribution board you may want to consider the Oakley ‘Dizzy’ PCB.

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/PNL	3
-15V	4

If you are intending to upgrade your module for full voltage control then you will have fitted the two 0.1" headers. These can now be connected to the 'Fout-pot' board as described in the 'Four pot's User Guide.

For those who have fitted the 0.1" headers, but are not intending to upgrade immediately, you will need to use those little jumpers. These fit across each pair of pins on the headers, that is pin1/2 and pin3/4.

If you are not intending to ever upgrade your module, then before you can use the module you must short out the headers. With four small pieces of uninsulated wire, resistor clippings are fine, you will need to short out four pairs of pins on both headers. Each header must have the following pins shorted together, pins 1 and 2, and then pins 3 and 4. Pin 1 is depicted by the square pin.

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-Apparatebau of Berlin, Germany. The cost is about £18 per panel. All you need to do is e-mail the fpd file that is found on the MultiLadder 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 3mm 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. Make sure that the LED is not lit. If its on, switch off and check all the parts again thoroughly. If your LED is off, and there is no smoke rising from the board (yikes!!), then we are ready to apply a gate input.

Use a gate signal from a midi-CV convertor, or a LFO's square wave output. Turn all the pots to their minimum value. The LED should briefly blip on for every low to high gate transition. Increase the decay pot, and hopefully, the LED blips will get brighter and last longer.

Now increase the sustain pot. This should increase the LED brightness, and it should stay on for longer. It should now stay on for the time the gate is high.

Increase the attack time, you should notice the LED ramping up to full brightness.

Now connect an audio signal of some sort, any will do, but a simple sawtooth wave is quite sufficient. You should connect it to the IN socket. Connect the OUT+ to an input channel on your mixing desk or other audio input. Hopefully, you should find that when the ADSR is gated the audio is heard. Play with the A, D, S and R pots to make sure they do the usual things. If you are not familiar with their action, I suggest you re-read the section at the front of this document.

Make sure that OUT- also produces audio. There should not be any noticeable difference in audio quality from the two outputs.

Make sure that attack, decay and release times can be changed from the near instant to around 10 seconds or so.

Setting Up

There are just two trimmers to be set before you are completely finished. The GAIN trimmer allows you to trim the output of the VCA to the desired gain. In the suggested layout, with no CV input, the ADSR output is trimmed to give +5V peak attack and sustain levels. Connect a positive gate signal to the Gate input. Set A, D, and R to their minimum positions, and set S to the maximum. Adjust LEVEL until you get +5V from the ‘OUT+’ output. Just to check you should get -5V from the ‘OUT-’ output.

Apply a 100Hz or so, square wave to the gate input. Now connect a zero volt signal to the CV input. You can easily get this by hooking up a non-oscillating filter's output to the CV input. Now attach the ADSR output to your final mixer or amplifier. With the volume quite low, adjust OFF until any audible 100Hz buzz is minimised. Turn the amplifier up to fine tune the control. You won't get it absolutely silent, but it will be near enough.

That's it you're ready to go.

Final Comments

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

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

Occasionally, there may be an error in the parts list. I have checked the documentation again and again, but experience has taught me to expect some little error to creep past. The schematic is always the correct version, since the parts list is taken from the schematic. So if there is any problem, use the schematic as the guide. If you do notice any error, please get in touch. You will be credited on the 'Updates and Mods' page, and you may get a free PCB if its a real howler.

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. January 2004

Version 2.0

Formatted on Lotus Word Pro