Oakley Sound Systems

VC-ADSR Upgrade

Using and Building the Four-Pot PCB Issue 1

User's Guide

V1.3

Oakley Sound Systems
PENRITH
CA10 1HR
United Kingdom

Introduction

The Four-pot board is designed to be used with the VC-ADSR boards. It enables the four parameters, attack, decay, sustain and release, to be under full voltage control. The Four-pot board connects with the VC-ADSR board via two four way 'hidden' interconnects. The Four-pot board takes 1U of panel width and requires four socket inputs. It can be fitted into either its own 1U panel, or be part of a 2U panel shared with the ADSR/VCA board. For the sake of clarity in this document we will refer to the 1U panel normally as the Upgrade module and the 2U panel as the VC-ADSR module.

Please note in order to upgrade a ADSR/VCA module you need to have a VC-ADSR main board. Early versions of the Oakley ADSR/VCA module used passive methods of controlling the attack, decay and release times and are not suitable for upgrading. Your ADSR/VCA board should be clearly marked as 'VC-ADSR' if you want to have the ability to upgrade. Those marked as 'ADSR/VCA' will not work.

It is possible to replace the ADSR/VCA board with a VC-ADSR board since the two are functionally identical and have the same pot spacing.

The Four-pot board is also used on other Oakley Modular projects, the VC-Mixer and the Control-X modules. Each of these modules have different requirements and thus the Four-pot board will have different components fitted to match the module built.

Power Supplies

The design requires plus and minus 15V supplies. These are in addition to those provided for the ADSR/VCA board. Thus if you are building a VC-ADSR panel you must remember to allow two power leads for the whole module. Power should be adequately regulated. The current consumption is about 25mA per rail. Power is routed onto the PCB by a four way 0.156" MTA or Molex type connector. The four pins are +15V, ground, panel, -15V. The panel connection normally allows you to connect the metal front panel to the power supply's ground without it sharing the modules' ground line. In the VC-ADSR design, the panel connection on this board is not used. More about this later.

Circuit Description

The board consists of four identical attenuator and mixer circuits, and one precision 5V reference source. We will not be using the precision reference circuit in the VC-ADSR or Upgrade module, and the parts for this are not fitted.

Power is supply via the usual four way MTA or Molex connector. As is the custom for Oakley modules, I have used ferrite beads to act as high frequency filters on the power lines. Decoupling at the point of entry is provided by C14 and C16 for the positive rail, and C15 and C17 for the negative rail. Additional decoupling is also provided elsewhere on the board by the other capacitors shown individually on the schematic. These capacitors keep the power supply clean of noise, and provide a reservoir for the little bursts of current that the circuit takes in normal operation.

Two grounds are provided, one for the circuit itself (called module ground), and one for the earthing of the jack sockets on the front panel in the Upgrade module. C1 is linked out in this version of the Four-pot board, thus connecting REF with module ground, 0V. This is to provide a handy ground reference pad, REF, for the normally closed (NC) lugs of the four input sockets.

There are eight op-amp ICs, U2 to 9, on this PCB, and each requires power. The power supply to each IC is shown separately at the bottom of the schematic to avoid cluttering the main circuit pathways. All these ICs are simple dual op-amps and require both negative and positive supplies.

The four inputs feature identical **'reversible attenuator'** circuits. Let us consider the first reversible attenuator. This takes its input from either I1 and has its output at OP1. The circuit is made up from four individual op-amps.

I1 is a solder pad at the bottom of the board and comes from the ATTACK CV socket. U2 (pins 1,2,3) acts simply as a voltage buffer. This is a circuit that merely 'sniffs' the input voltage and creates a copy of that signal at its output on pin 1. We are free to take a bucket full of current from this output without affecting the input at all. A bucket full of current must be defined here as anything less 15mA or so... hardly a bucket then, is it Tony??

U3 (pins 1,2,3) is a different circuit to the one around U2 (pins 1,2,3). This is an amplifier with a gain of -1. This means that its output is the opposite polarity to the input voltage. The input voltage is sensed by R5, and the current flowing through R5, is matched by an equal current flowing in the series combination of R4 and PR1. The op-amp does this because it adheres to the golden rules. [Actually, it can only **try** to adhere to these golden rules] Now the golden rule in question is that an op-amp with negative feedback must move its output so that its two input pins are both the same voltage. The negative feedback is provided by the resistors R4 and PR1. So as the input signal tries to inject current into the op-amps inverting (-) pin via R5, the output will move against this by taking that current away through R4 and PR1.

So a positive input voltage at R5 will lead to the output going negative. The ratio of the resistors will determine the gain of the inverting circuit. Making both the resistors the same value will mean that the gain of the op-amp is -1. That is 5V at the input gives us -5V at pin 7. PR1 allows the user to trim the feedback resistance to exactly match the input resistor, R5. However, this isn't really necessary in the VC-ADSR or Upgrade modules. This level of precision is only needed in the Control-X modules. Because of this, we are going to link out PR1 and make R4 and R5 both 33K resistors.

As far as the connected input goes, it does not see what then happens to these two copies of itself. It only 'sees' the resistance of R2. Hang on, what's that R2 doing, why don't we get rid of it altogether and let the input be really high impedance? After all, a high impedance input is a good thing, isn't it? Well, yes and no. R2 is only there to make sure that the input is not just floating in thin air with nothing attached. This is useful for when the circuit is not plugged in. Any stray static electricity cannot build up on this input and destroy the op-amp. So too much input resistance is a bad thing. R2 is chosen to have a high enough value so as not to affect the circuit sensitivity that much. R3 offers some sort of protection for U2 (pins 1,2,3) in any overvoltage situation. A typical example of this is trying to use the inputs when the module is not powered up.

Lets go back to our two copies of the input signal. Remember that one is an exact copy, and one is an inverted version. These two voltages are placed across the two ends of the control pot. As the pot wiper moves from one side to the other, it will tap off a proportion of each signal. Consider what will happen with a +5V input signal at I1. This will give +5V at the output of U2 and -5V at the output of U3. The wiper of the pot can thus move from +5V to -5V. In the middle position, both voltages take equal precedence and the voltage at the wiper is zero.

The voltage at the wiper is connected to another inverting amplifier based around U2 (pins 5,6,7). The op-amp's gain is also minus one. So a voltage of +5V at the wiper of the pot, will give -5V at the op-amp's output. The amplifier has another resistor connected to its input pin, R20. This resistor is very important to us building the VC-ADSR and Upgrade modules. The current flowing in R20 will sum with that flowing in the other input resistor. In other words this circuit will mix the wiper voltage with the voltage present at the point labelled ATTACK-IN. This point goes back to one of the pins on a four way header called A-D. The decay channel also uses the A-D header, whilst the sustain and release channels use another header called S-R.

The headers are shown at the bottom right of the circuit diagram. You can see each one has two inputs and two outputs. This is because the A-D and S-R headers serve two purposes. One, they allow the Four-pot board's output voltages to get to the ADSR board. Two, they allow for the ADSR's pot settings to be sent to the Four-pot board. The Four-pot board not only processes the input CVs, but also adds them to the ADSR board's own settings. The summed CVs are then passed back to the ADSR. This might seem a little odd, but it does mean that the ADSR board does not need to have summing circuitry included. The summing circuitry is only needed with the full VC-ADSR module, so there was little point in putting it on the ADSR board.

As the output of the mixer is inverting, we need to invert it once again to get it the right way up. U3 (pins 5,6,7) is configured as yet another inverting amplifier. C10 and R21 form part of the stabilising network designed to keep the op-amp from oscillating under capacitive loads. Capacitive loads are easily created in a patch cord synthesiser. Each patch cord acts as a low value capacitor to ground. This can make an op-amp very unhappy and its output will oscillate, often at very high frequencies. However, we don't really need to protect for high frequency instabilities in the VC-ADSR or Upgrade modules since we are only connecting to a short length of wire to the ADSR board. You can therefore replace R21 with a wire link and omit C10 entirely. If you do fit them for some reason, they won't affect the circuit operation in a bad way.

At the output of the final inverting stage is a small voltage limiting network based around a resistor, R42, and a schottky diode, D2. These are used to limit the maximum output swing to positive values only. A schottky diode is used because its resistance drops dramatically when it has around -0.2V across it. A normal junction diode needs at least -0.6V before this happens. When D2's resistance drops it shunts any excess negative voltage to ground, thus preventing the output voltage to drop below -0.2V or so. This negative clamping is done to protect the 4052 analogue switch on the

ADSR board. Applying too much negative current to the inputs of this chip will cause it misbehave or possibly die.

The voltage across the diode is fed to the header A-D. Here it will travel to the ADSR board to control the rise time of the ADSR's integrator.

The other three channels are identical in operation to the attack channel.

The wire links, LK1-4, are used in the control modules only and are not used for the Upgrade or VC-ADSR modules.

Components

All of the parts are easily available from your local parts stockist with the possible exception of the pot brackets. I use Rapid Electronics, RS Components, Maplin and Farnell, here in the UK. The Control module was designed to be built from parts obtainable from Rapid Electronics and myself only. Rapid's telephone number is 01206 751166. They offer a traditional 'paper' catalogue as well as an on-line ordering service.

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

The resistors can be 5% carbon 0.25W types except where stated. However, I would go for 1% 0.25W metal film resistors throughout, since these are very cheap nowadays.

The aluminium electrolytics should be 50V or 63V. These should be radially mounted, ie. they are cylindrical with the two legs sticking out from underneath.

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.

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

The board mounted pots are Spectrol 248 series with Oakley pot brackets. You could use any type you want, but not all pots have the same pin spacing. Not a problem, of course, if you are not fitting them to the board.

I carry a stock of all the pots you need as the Upgrade pot kits. One thing to note if you are buying knobs for these pots. The Spectrol ones have a 6.35mm diameter shaft. This will probably not make a difference to your choice of knob unless you are using Collett style knobs which can be quite fussy about diameter.

L1 to L2 are leaded ferrite beads. These are little axial components that look like little blackened resistors. They are available from most of the mail order suppliers. Find them in the EMC or Inductor section of the catalogues. Farnell sell them as part number: 108-267.

0.1" headers come in two basic types: MTA or 'Strip and Crimp'. Although I prefer the former for power leads, I exclusively use the Molex crimp system for the smaller 0.1" interconnects. You'll need to buy vertical friction lock headers to fit onto the boards. The plugs are actually made from separate housings and crimp terminals. Crimp terminals are usually available in bags of one hundred, but are quite cheap. You can crimp these with a special expensive tool, or solder them.

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

Finally, if you make a circuit change that makes the circuit better, do tell me so I can pass it on to others. Any updates are added to the current user guide, and are posted on the 'Oakley-Synths' list.

Parts List

A quick note on European part descriptions. To prevent loss of the small '.'as the decimal point, a convention of inserting the unit in its place is used. eg. 4R7 is a 4.7 ohm, 4K7 is a 4700 ohm resistor, 6n8 is a 6.8 nF capacitor.

Note: Some of the resistor values shown on the circuit diagram are different to those in the part list. The schematic shows the generic circuit for the Four-pot board and covers all versions of the board including, but not only, the Control-X and Upgrade modules.

Links

The Four-pot boards used in the VC-ADSR and Upgrade modules have nine wire links fitted. These can be made from solid core tinned copper wire, or resistor lead clippings.

You must link out the multiturn presets PR1-4. Do this by connecting a small wire hoop between pins 1 and 2. Pin 1 is the square one, and pin 2 is the middle one. Pin 3 is left open.

You must link out R21, R27, R33 and R39.

You must link out C1.

Resistors

All 5% carbon 1/4W or better.

10K	R3, 7, 11, 15
33K	R4, 5, 23, 22, 8, 9, 29, 28, 12, 13, 35, 34, 16, 17, 41, 40
100K	R18, 19, 20, 24, 25, 26, 30, 31, 32, 36, 37, 38
470K	R2, 6, 10, 14
4K7	R42, 43, 44, 45

R1 is omitted.

Capacitors

100nF 63V axial ceramic	C2, 3, 4, 5, 6, 7, 8, 9, 14, 15
2u2, 63V electrolytic	C16, 17

C10, 11, 12 and 13 are omitted.

Discrete Semiconductors

BAT-42 Schottky diode	D2, 3, 4, 5

D1 is omitted.

Integrated Circuits

TL072 dual bi-fet op-amp	U2, 3, 4, 5, 6, 7, 8, 9

U1 is omitted.

Miscellaneous

100K, 50K or 25K Linear Spectrol 248 pot	VR1, 2, 3, 4
Pot brackets to suit Spectrol pots	4 off
Leaded ferrite beads	L1, L2
Molex/MTA 0.156" four way header	PWR
Molex 0.1" vertical friction-lock header	A-D, S-R
Four Molex 0.1" housings	4 off
0.1" terminal crimps	16 off (or one pack of 100)

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 eight 8-pin DIL sockets.

Building the VC-ADSR or Upgrade 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. The most common error with many of these was the wrong parts inserted into the board. 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!

The 'Four-pot' PCB is flashed with solder around the pads. This helps the soldering process. Please note: this flashing is with solder that contains lead. You should therefore wash your hands after handling the board. It is also recommended that for best results this board is soldered with lead-tin solder.

I always using water washable flux in solder these days for my board manufacture. In Europe, Farnell sell Multicore's Hydro-X, a very good value water based product. You must wash the PCB at least once an hour while building. Wash the board in warm water on both sides, and use a soft nail brush or washing up brush to make sure all of the flux is removed. Make sure the board is dry before you continue to work on it or power it up. 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' solder. This is not a good idea as it can create bubbles in the solder. If you prefer to have a fixed temperature iron, then it is best to get a 18W one for this purpose. I use an ordinary Antex 25W iron with a Variac power supply running at 205V. This seems to work well for me.

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

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

The BAT-42 diodes can be treated much like resistors. However, they must go in the right way. The cathode is marked with a band on the body of the device. This must align with the vertical band on the board. In other words the point of the triangular bit points *towards* the cathode of the diode.

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

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

IC sockets are to be recommended, especially if this is your first electronics project. Make sure, if you need to wash your board, that you get water in and around these sockets. And that any water is thoroughly dried out before you power up. Sockets harbour little pools of water in their pins which can lead to some very odd effects. Several sharp taps face down onto a towel will loosen off any remaining globules of liquid.

I would make the board in the following order: resistors, IC sockets, capacitors, diodes, electrolytic capacitors. Then the final water wash. Do not fit the pots at this stage. The mounting of the pots requires special attention. See the next section for more details.

Mounting the Board Mounted Pots

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	4 off	**OR**
25K linear	M248 25K M	4 off	

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

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

You can now present the front panel up to the completed board. Although, I usually fit the sockets at this point, and wire up the ground tags first. After this is done, I then mount the PCB to the front panel. You need to add the washer between the panel and the nut. Again, do not over tighten and be careful not to scratch your panel.

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

The Spectrol 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.

Connections

This is the bit that causes the most confusion. I would recommend that you use coloured wire to help track down any problems if they occur. I use thin multistrand 'hook up' wire for all my connections, except for the grounding frame on the sockets.

If you have bought 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.

How you connect the sockets to the panel will depend a little on whether you are building the VC-ADSR or Upgrade module. I have made two sections for each option, but if you are building the VC-ADSR, you will need to refer to the ADSR User Guide as well.

Building the 1U Upgrade Module

The first thing we are going to do is 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 four sockets in total.

The 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 both 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 we need to common both rows of NC lugs together. This will allow the inputs to be shorted to ground whenever a jack is not inserted into a particular socket. This is done via the REF pad on the PCB. The REF pad is tied to ground by the wire link in position C1. Now using the same method as you commoned the earth tags, connect the NC lugs together on the top row and then the bottom row. One thing to note is that if you are using Switchcraft 112APC sockets: Make sure that any wire does not touch the tip of any inserted jack socket.

Fit the Four-Pot 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.

The commoned NC tags on the top row of sockets must now be connected to the board. Again using a bit of insulated wire, connect one of the NC lugs on the top row to the pad labelled REF on the PCB. Do the same for the bottom row. The REF pad should be big enough to take both pieces of wire.

Now its time to wire up the signal lugs to the board.

Socket Name	***PCB Solder Pad***
A CV	I1
D CV	I2
S CV	I3
R-CV	I4

That's it, your module is complete bar the testing.

Building the 2U VC-ADSR Module

In this module we are going to be using two populated PCBs against one panel. Getting access to the Four-pot PCB once the ADSR board is in place is a little tricky. It is best, therefore, to connect the Four-pot board up first, then the Four-pot board.

The ADSR User Guide will give you the details required to mount the LED in place correctly.

The first thing we need to do is 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 eight sockets in total.

The 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 all four 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 we need to common the four NC lugs that belong to the four A, D, S and R CV inputs together. This will allow the inputs to be shorted to ground whenever a jack is not inserted into a particular socket. This will be done via the REF pad on the Four-pot PCB. The REF pad is tied to ground by the wire link in position C1. Now using the same method as you commoned the earth tags, connect the two NC lugs together for A CV and D CV, and then S CV and R CV.

One thing to note is that if you are using Switchcraft 112APC sockets: Make sure that any wire used will not touch the tip of any inserted jack socket.

Its now time to fit the Four-Pot PCB against the front panel. The commoned NC tags on the top row of CV input sockets must now be connected to the board. Again using a bit of insulated wire, connect one of the NC lugs on the top row to the pad labelled REF on the PCB. Do the same for the bottom row. The REF pad should be big enough to take both pieces of wire.

Now its time to wire up the four input CV lugs to the Four pot board.

Socket Name	***PCB Solder Pad***
A CV	I1
D CV	I2
S CV	I3
R-CV	I4

Now fit the ADSR/VCA PCB against the front panel again. Make sure the LED is fitted correctly. 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. All eight earth tags are now commoned together.

Connect, with five 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, NC, 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

Again 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.

That’s it, you have now pretty much finished your module.

Board to Board Interconnects

You need four 4-way 0.1" female plugs. These are sometimes called housings, since they aren't plugs themselves but merely housings for the individual crimp terminals.

Terminals have to be bought in packs of one hundreds, but this is OK, because they are not expensive. These are normally designed to be crimped but they can be easily soldered with care.

Make each wire to be 25cm long, you'll need 16 of them. I normally strip back the wire by just 2 to 3mm. Place all the bare wire into the crimp on a heatproof surface. I use 12mm MDF board to protect my bench top, which although not at all burn proof will take plenty of heat from a soldering iron without major damage. Rest a pair of pliers on top of the wire to hold it in place. Slip the crimp under the wire, so that the wire's insulation butts up to the edge of the terminal. Then solder in place. Sometimes I find I need to gently squash the crimp part of the terminal so that it will fit into the housing. This is easier to do before you solder it, although it can be done after with care.

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

I like to use as many different colour wires as possible. It makes wiring easier and looks great. And it also helps to make sure that pin 1 on one header is connected to pin 1 on the other.

Power Connections

The power socket on both the ADSR and Four-pot boards is a 0.156" MTA 4-way header. Friction lock types are recommended. This system is compatible with MOTM systems.

Power	***Pin number***
+15V	1
Module 0V	2
Panel	3
-15V	4

Remember, 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 modules 0V supply. Earth loops cannot occur through patch leads this way, although screening is maintained. Of course, this can only work if all your modules follow this principle.

Both modules must be powered up for the VC-ADSR to work.

Ready made power leads are available from me for a reasonable price.

At the rear of this user guide I have included 1:1 drawings of the suggested front panel layouts for each of the Control modules.

Actual panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £15 to £20 depending on the size of the panel. All you need to do is e-mail the fpd file that is found on the Control Module 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.

Final Comments

I hope you enjoy building the Oakley Upgrade and VC-ADSR Modules. 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 directly or the Oakley Synths list on Yahoo groups. 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 and I will fix it for you. You will have to pay for my time at £20 per hour and also have to pay for the postage both ways, and for any replacement parts needed. 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'.

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. I am also looking for feedback regarding the usefulness of this User Guide. If you feel that it is lacking in any way, please do get in touch. 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 5 pm, British time.

Last but not least, can I say a big thank you to all of you who helped and inspired me.

Tony Allgood

Cumbria, February 2005