Oakley Sound Systems

‘Control-X’
Voltage Source and Attenuator Modules

Issue 1

User’s Guide

V1.0

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

Introduction

These three modules are simple but very effective control modules. Each Control module is based around the same circuit board, the Oakley Four-pot PCB. This is then simply built up in one of the three ways to match your chosen panel option.

The Control-1 module generates four very stable voltages. The output can be varied from -5V to +5V for each of the four outputs. Each output is completely separate and can control many modules without loosing accuracy. When the control knob is central, ie. pointing directly upwards, the output voltage is zero volts. You can use these voltages to control other modules within a modular synthesiser. For example, you can add offsets to CV inputs to increase the range of existing modules. This way you can drive your Oakley VCO to very low frequencies and use it as an LFO. Or you can deliberately cause audio inputs to go into asymmetrical distortion by forcing the input amplifiers into overdrive.

The Control-2 generates two stable voltages in a similar fashion to the Control-1. However, this module also features a highly stable reversible attenuator too. In this section the control knob can be set so that the gain of the attenuators input is anything from -1 to +1. This means that the input signal, CV or audio, can be inverted and attenuated at the same time. You can make -4V into +4V and vice versa, or turn positive envelope outputs to negatively going ones.

The Control-3 module is the big one. This uses a 2U wide panel to achieve multiple functions. You can use it as four separate voltage sources, where each section has a dedicated switch that can turn the output off and produce an accurate zero volts. Also each section can be used as a reversible attenuator when the input jack socket has a jack plug inserted.

Remember that all three modules are made from the same PCB. You simply build it up as your option dictates. On the board there are four identical reversible attenuators and one precision 5V reference.

Power Supplies

The design requires plus and minus 15V or 12V supplies. These 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. You could, of course, wire up the board by soldering on wires directly. The four pins are +15V, ground, panel, -15V. The panel connection allows you to connect the metal front panel to the power supply’s ground without it sharing the modules’ ground line. More about this later.

Circuit Description

The board consists of four identical attenuator and mixer circuits, and one precision 5V reference source.

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, and one for the earthing of the jack sockets on the front panel.

The +5V reference is simply constructed from a precision zener diode, U1. Technically, its more than just a zener diode. Its a complete integrated circuit (IC), hence its U status rather than D. But one can think of it just like a normal zener but with a superb performance. R1 provides the bias for the internal zener in the chip and supplies the current to keep the rest of the IC working. C1 provides a little high frequency decoupling, and D1 provides protection against overvoltage from external sources. In certain implementations of the Control module, the reference voltage output from U1 goes directly to the NC lugs on the front panel sockets via the REF solder pad. NC lugs don't normally connect to the outside world, since they are an internal connection only. However, during the point of jack insertion, the jack will touch both NC and Signal lugs together. This could damage U1 if unprotected and too much current flows into the socket. D1 protects U1 by shunting any negative or excessive positive voltage to ground.

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 or LK1 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. In the Control-3 module it will be connected to an input socket on the front panel. In Control-1 and Control-2 we don't use it and LK1 is fitted instead. LK1 connects the +5V reference voltage generated by U1 directly to the attenuator circuit.

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. In practice it allows us to set the null point of the reversible attenuator’s pot.

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 not used in the Control Modules. It is used in the ADSR module, so that an external CV can add to the voltage created at the wiper of the pot. R20 and the other three resistors like it are not fitted to this module.

Since the output of this amplifier is inverting, we need to invert once again to get it the right way up. U3 (pins 5,6,7) is configured as yet another inverting amplifier, but this time with an extra twist. 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. In many other modules this stabilisation is simply achieved by a 1K resistor in series with the output load. However, this causes the output voltage to fall with increasing load resistance. Here we are using a more active approach. 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 other many Oakley projects.

Two output pads are laid out on the PCB for each attenuator circuit. This is to keep the wiring down to a minimum when you are making your chosen module. Control-1 and Control-2 modules will use the lower row of pads, that is; O1 to O4. Control-3 uses the OP1 to OP4 pads. Control-3 uses a simple single pole double throw (SPDT) switch to either connect the appropriate output socket to the attenuator's output, or shunt it down to ground. The OP pads are situated on the PCB so that the two wires to the switch are kept as short as possible.

At the output of each attenuator circuit is a small voltage limiting network based around a resistor and a schottky diode. These are not used in the control modules and are omitted from the parts list. They are used in the VC-ADSR to limit the maximum output swing to positive values only.

Each of the four attenuators are identical in circuitry and operation. However, the user is able to configure the PCB to his own needs. Fitting the links LK in place connects the +5V directly to the attenuator. The front panel pot will control the output of the attenuator from -5V to +5V. The Control-1 module has all four attenuators wired like this to provide four independent voltage sources.

The Control-2 module has two of its attenuators fitted with LK links. This gives us two voltage sources. The third attenuator is connected to an input socket, and thus provides a reversible attenuator function for any CV or audio signal. The fourth attenuator is unused and its parts should be completely omitted.

The Control-3 module has no links fitted. The inputs to each attenuator come from its own dedicated input socket. The NC lugs on these sockets are connected to the REF pad. This causes +5V to be fed back into the attenuators if the input socket has no jack inserted.

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 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.

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. You could use a better specified op-amp if you want a little more stability. LF412 devices should work very well at both audio and CV. Also the AD712 is very good too. LT1013 and OPA2277 are not to be recommended because of their slow speed and high input bias current.

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 Control 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.

The multiturn trimmers are the ones that have the adjustment on the top of the box. Spectrol and Bourns make these. Some types are 22 turns, while others are 25 turns. Either will do. They should have three pins that are in a line at 0.1” pitch. Don’t chose the 10-turn ones with the adjustment on the end, they won’t fit on the PCB.

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 that the parts lists are separated out into two. The first one covers the Control-1 and Control-3 modules. Whilst the second deals with the Control-2 only.

Control-1 and Control-3 Parts List

Control-1 boards must have the following four wire links fitted: LK1, 2, 3, 4. For Control-3 boards these links are left unfitted.

Resistors

All 5% carbon 1/4W or better.

10K	R3, 7, 11, 15
100R	R21, 27, 39, 33
27K	R4, 8, 12, 16
2K7	R1
33K	R5, 19, 18, 23, 22, 9, 25, 24, 29, 28, 13, 31, 30, 35, 34, 17, 37, 36, 41, 40
470K	R2, 6, 10, 14

Capacitors

100nF 63V axial ceramic	C1, 2, 3, 4, 5, 6, 7, 8, 9, 14, 15
33pF Low-K ceramic	C10, 11, 12, 13
2u2, 63V electrolytic	C16, 17

Discrete Semiconductors

10V zener diode approx 300mW	D1

Integrated Circuits

TL072 dual bi-fet op-amp	U2, 3, 4, 5, 6, 7, 8, 9
LM136 or LM336 5V reference	U1

Miscellaneous

10K multiturn trimmer	PR1, 2, 3, 4
50K or 25K Linear Spectrol 248 pot	VR1, 2, 3, 4
Pot brackets to suit Spectrol pots	4 off
Single pole, Double throw toggle switches	4 off for Control-3
Leaded ferrite beads	L1, L2

Molex/MTA 0.156” four way header	PWR

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.

Control-2 Parts List

Control-1 boards must have the following wire links fitted: LK1, 2.

LK3 and LK4 are left unfitted.

Resistors

All 5% carbon 1/4W or better.

10K	R3, 7, 11
100R	R21, 27, 33
27K	R4, 8, 12
2K7	R1
33K	R5, 19, 18, 23, 22, 9, 25, 24, 29, 28, 13, 31, 30, 35, 34
470K	R2, 6, 10

Capacitors

100nF 63V axial ceramic	C1, 2, 3, 4, 5, 6, 7, 14, 15
33pF Low-K ceramic	C10, 11, 12
2u2, 63V electrolytic	C16, 17

Discrete Semiconductors

10V zener diode approx 300mW	D1

Integrated Circuits

TL072 dual bi-fet op-amp	U2, 3, 4, 5, 6, 7
LM136 or LM336 5V reference	U1

Miscellaneous

10K multiturn trimmer	PR1, 2, 3
50K or 25K Linear Spectrol 248 pot	VR1, 2, 3
Pot brackets to suit Spectrol pots	3 off
Leaded ferrite beads	L1, L2
Molex/MTA 0.156” four way header	PWR

You may well want to use sockets for the ICs. I would recommend low profile turned pin types as these are the most reliable. You need six 8-pin DIL sockets.

Building the Control Modules

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!

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 zener diode can be treated much like a resistor. 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 ceramic capacitors are strange flat plates made from pot. Be careful with these and make sure you have bought the ones with 0.2” lead spacing. Forcing the smaller 0.1” ones into these larger pads will break them. Another thing to watch out for is the identification markings on these capacitors. For example n33 is actually 330pF and not 33pF.

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.

There are three types of trimmer used in Oakley projects. But here we are just using the Spectrol or Bourns multiturn type. Your multiturn pot should be marked as 103 as it is a 10 000 ohm or 10K pot. These multiturn pots can go in any way around.

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 trimmers and temp co resistor to the board with no-clean or ordinary ‘ersin’ flux solder. 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 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.

Control-1 Connections

This module is the simplest to wire up. 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 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 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.

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

Socket Name	***PCB Solder Pad***
V1	O1
V2	O2
V3	O3
V4	O4

That’s it, your module is complete bar the trimming.

Control-2 Connections

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 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 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.

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

Socket Name	***PCB Solder Pad***
V1	O1
V2	O2
IN	I3
OUT	O3

That’s it, your module is complete bar the trimming.

Control-3 Connections

This module is the most complex to wire up out of the three. 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 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 the 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 top row of NC lugs together. This will allow the +5V reference to be automatically connected whenever a jack is not inserted into a particular socket. Using the same method as you commoned the earth tags, connect all the NC lugs together. 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.

Do not common the NC lugs on the lower row of pots. These should be left floating.

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 to the pad labelled REF on the PCB.

Now its time to wire up the upper row’s signal lugs to the board.

Socket Name	***PCB Solder Pad***
IN1	I1
IN2	I2
IN3	I3
IN4	I4

The signal lugs on the lower row of sockets are not to be connected to the PCB at all. These will connect to the four switches. Fit the switches now, making sure that you do not scratch the panel when you tighten up the nuts. I normally mount the switches so that one nut goes onto the switch bush, then the sprung washer. Then the bush is fed through the panel. One nut then secures the bush to the panel. There normally is another flat washer provided with the switch, but I do not use this.

Each switch will be wired in a similar manner. The top terminal of the switch must be wired to the pad labelled OP. That is, switch 1 will go to OP1, switch 2 will go to OP2, and so on. The bottom terminal will go to the pad labelled 0V. Switch 1 will go to 0V1, switch 2 will go to 0V2, and so on. Keep your wires tidy and short. The board has been designed to allow that only short runs of wire are needed.

The remaining switch lug, the middle terminal, on each switch must go to the signal lug on the appropriate socket. That is, switch 1 goes to OUT1, switch 2 goes to OUT2, and so on.

Power Connections

The power socket is 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.

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 per 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.

Trimmers

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

Each trimmer is set up in the same way. The aim of the trimmer is to make sure that when the pot is set vertical that the output level is zero volts. The trimming method varies depending on which module you have built.

Control-1 and **Control-3** is set up with a voltmeter. Simply set the pot so that it matches the line on the front panel. Then measure the output with a voltmeter. Adjust the trimmer until you get 0V from the output. PR1 adjusts Voltage V1, PR2 adjusts Voltage V2, and so on.

Control-2 is set up with a voltmeter and a pair of ears!! For the first two trimmers, simply set the top two pots so that they match the line on the front panel. Then measure the outputs with a voltmeter. Adjust the trimmer until you get 0V from the output of each one. PR1 adjusts Voltage V1 and PR2 adjusts Voltage V2. For the third pot, connect an audible sine wave (from a VCO) to the input of the attenuator and connect its output to an amplifier or mixing desk. Set the pot to the middle position, and adjust PR3 until any audio breakthrough is minimised.

Final Comments

I hope you enjoy building the Oakley Control 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.

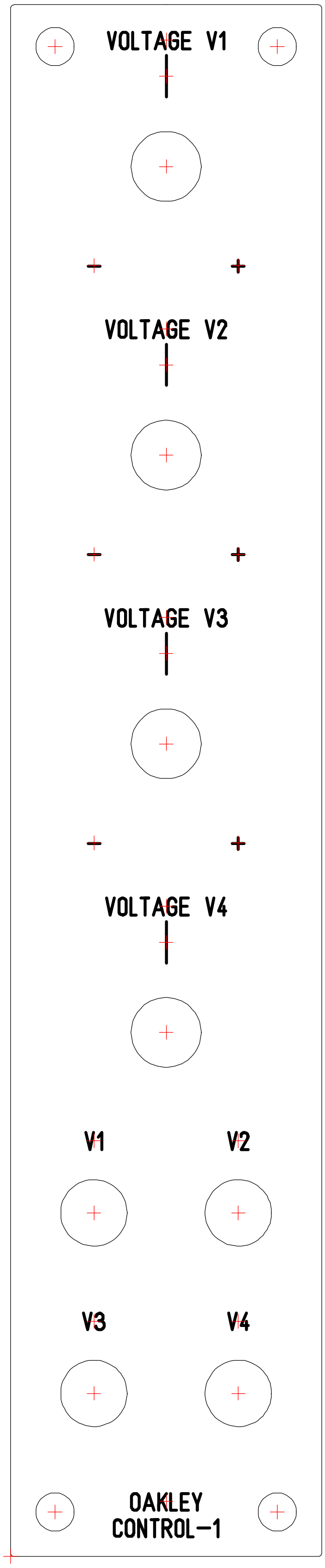
I do offer a get-you-working service. Send your completed non-working module back to me with £20 and I will fix it for you. You will also have to pay for the postage both ways, and for any replacement parts needed. Make sure you wrap it carefully and include a full description of the fault. If you are sending the item from outside the EU, then be sure to say on the customs label ‘item being sent for repair only’.

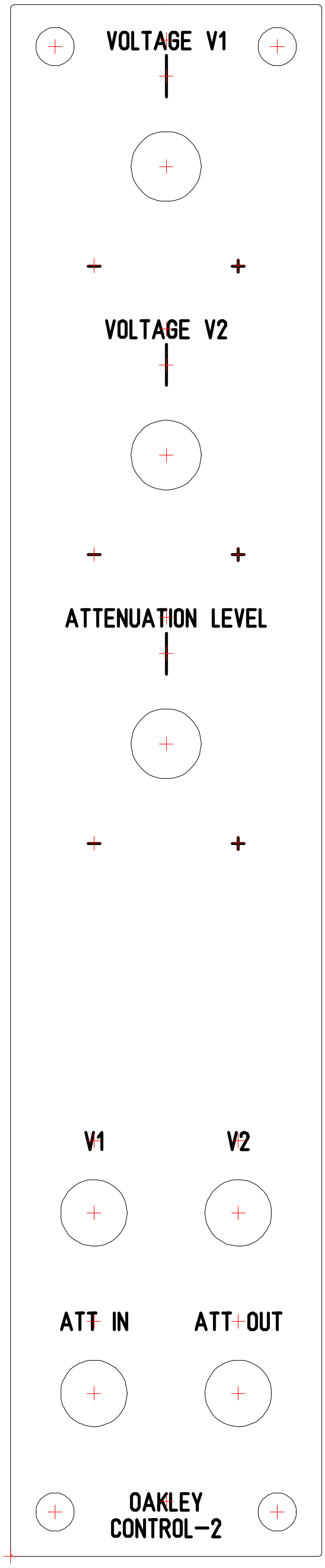
Please further any comments and questions back to me, your suggestions really do count. If you have any suggestions for new projects, feel free to contact me. I am also looking for feedback regarding the usefulness of this User Guide. If you feel that it is lacking in any way, please do get in touch. 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.

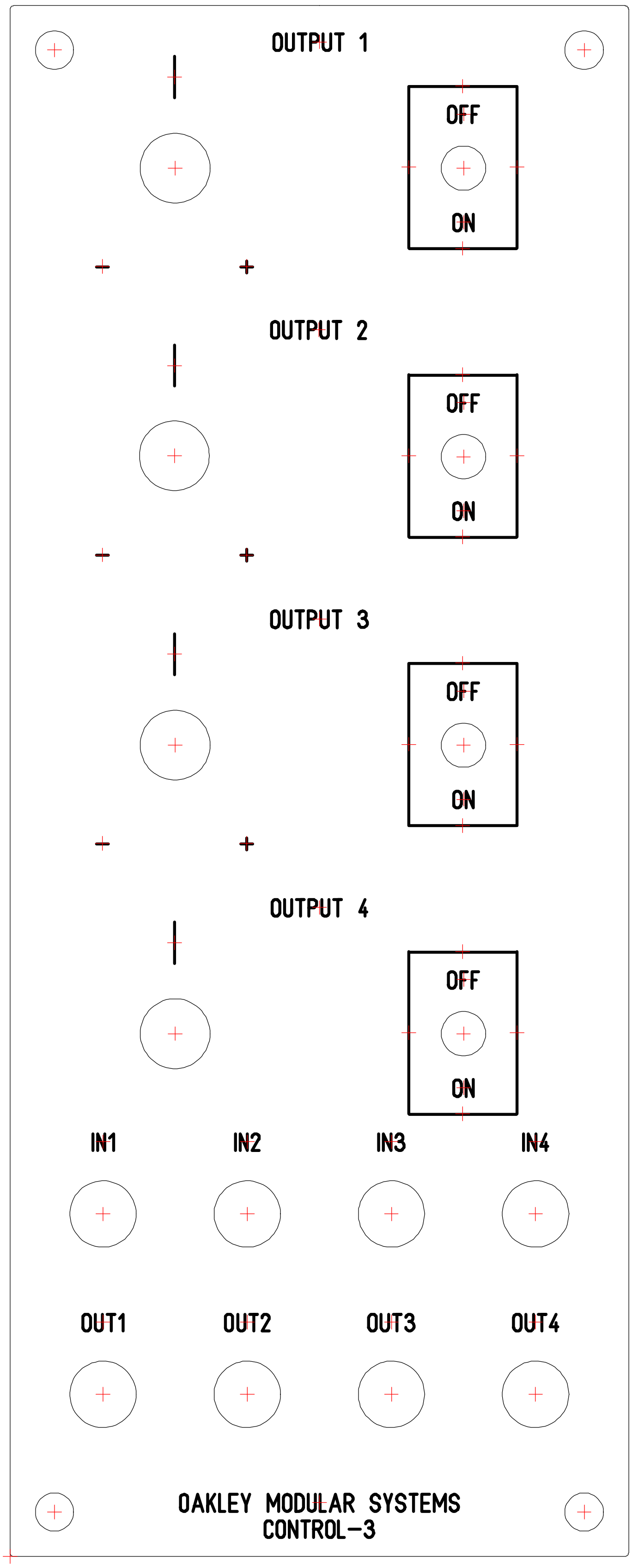
Last but not least, can I say a big thank you to all of you who helped and inspired me.

Tony Allgood. January 2003

Version. 1.0


VOLTAGE V1
VOLTAGE V2
VOLTAGE V3
VOLTAGE V4
V1
V2
V3
V4
OAKLEY
CONTROL-1


VOLTAGE V1
VOLTAGE V2
ATTENUATION LEVEL
V1
V2
ATT IN
ATT OUT
OAKLEY
CONTROL-2


OUTPUT 1
OFF
ON
OUTPUT 2
OFF
ON
OUTPUT 3
OFF
ON
OUTPUT 4
OFF
ON
IN1
IN2
IN3
IN4
OUT1
OUT2
OUT3
OUT4
OAKLEY MODULAR SYSTEMS
CONTROL-3