Oakley Sound Systems

Noise Generator and Dual Filter

PCB Issue 2

Audio Processor

User's Guide

V2.2

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

Introduction

The Oakley Noise and Dual Filter module is used to generate non pitched sounds such as wind, surf and other ethereal sounds. It can also be used as a sound source for percussive sounds like cymbals and snare drums.

Two types of noise output are provided. White noise which when unfiltered sounds like gas escaping, and pink noise which sounds more like a big waterfall. The module's pink noise circuitry comprises of a complex cascade network designed to replicate the -3dB/octave low pass response that true pink noise requires.

The module also gives a very low frequency output or infra-red signal. This can be heard as a serious of random thumps when listened too, but it is actually a randomly varying output voltage changing all the time, sometimes quickly and sometimes hardly at all. Fascinating to watch when it controls an LED, but it comes into its own when controlling filter cut-off on an otherwise static sound

This module also features two separate voltage controlled one pole filters, one high pass and one low pass. These are internally normalised to the noise generator's output, but may also be used separately to process other sound sources too. One pole filters have a roll-off of -6dB per octave, and do not possess any self resonance.

The two filters may be used in cascade to produce a bandpass response.

One pole filters sound particularly nice with audible noise, and give far more natural wind and surf sounds than the usual 2 or 4 pole filters. The two filters are both voltage controlled so they may be controlled by an external CV signal from, perhaps, an ADSR, LFO or midi-CV convertor. A front panel pot allows you to control the depth of the modulation for each filter. This pot is a 'reversible attenuator, so you are able to control not only the depth of the modulation but also the polarity.

The control range of each of the filter sections is identical, and covers the whole audio range from below 20Hz to above 20KHz. The whole range can be swept by the front panel ‘frequency’ pot. In theory, it is possible to exceed this range by using an external CV in conjunction with the ‘frequency’ pot. However, it should be noted that trying to make the high pass filter work above 40KHz does tend to cause some unwanted audible artefacts. This behaviour will not harm the filter but it is probably best avoided.

The noise source is true analogue, a reverse biased NPN transistor.

Power Supplies

This module is designed to run from plus and minus 15V supplies. These should be adequately regulated. Although quite large perturbations in the supply will not cause any appreciable effect. The current consumption is about 30 mA 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.

The PCB set

The main PCB is 101 (height) x 107 (depth) mm in size. All three boards use double sided copper traces and have through plated holes. The solder pads are large and are easy to solder and de-solder if necessary. They have a high quality solder mask on both sides for easier soldering, and have clear legending on the component side for easier building.

If you are building the standard design there are no components mounted off the boards. All components including sockets and pots are soldered directly to the boards.

Previously, many Oakley modules have had the sockets, switches and extra pots wired to the board by individual wires. This module allows all the socket wiring to be done via the socket PCB and two MTA solderless or Molex connections. If you are building this module in the standard Oakley format this new system will reduce assembly time and possible wiring errors.

Some people will wish to use this Oakley design in a non standard format, such as fitting it to another manufacturer's rack or one of their own invention. This is perfectly easy to do. Simply do not use the socket board and wire the main board to the sockets as per usual.

I have provided space for two of the control pots on the main PCB, whilst the other two pots are fitted to their own board. If you use the specified pots and brackets, the PCBs can be held firmly to the panel without any additional mounting procedures. The pot spacing is 1.625" and is the same as vertical spacing of the MOTM modular synthesiser.

There are detailed instructions later in the document about how to build the boards. The whole project takes around 2.5 hours to build and test.

Circuit Description

The noise and the two filter sections are essentially separate circuits that only share the same power supply. The power supply section is shown on page 2 of the schematics. Power is applied to the board through the PSU 4-way MTA / Molex connector. F1 and F2, small axial ferrite beads, provide some high frequency resistance, and along with C28 and C29 prevent the board from being effected by any noises on the power rails. They also help keep any unwanted noises going the other way too.

Additional decoupling is also provided elsewhere on the board by eight other capacitors shown to the left of the power circuitry. 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 power supplies to each of the op-amp ICs are shown separately from the main schematics to avoid cluttering the diagram.

Still on page 2 we will consider the two filter sections. These are actually identical circuits even though one is used as a low pass filter and the other as a high pass filter. The only difference between the two circuits is from where in the circuit we take the output.

The filters are made from a circuit described as a state variable filter or SVF for short. Unlike the Oakley SVF module this filter is only one pole. Only one active part actually controls the cut-off frequency and not two like in the SVF module. The part that gives us control over the cut-off point is an audio voltage controlled amplifier (VCA) made by the THAT Corporation, part number 2180. These parts are normally used in gain control, but it is a simple matter to allow them to work in filters too.

The precise mathematics of the one pole filter is beyond the scope of this User Guide, but a more detailed account of its operation can be found in the THAT Design Note number 130 found on their website. www.thatcorp.com

Those of you who have built our VC-ADSR may recognise the circuitry involved in the one pole filter as I used a similar circuit in the ADSR core.

Two basic components make up the filter circuitry. One is a differential amplifier, this is U6 (pins 5, 6, 7). The other is a more complex circuit called an integrator, this is based around the VCA and U6 (pins 1, 2, 3).

An integrator is a charge storage device. Think of it like a bucket for electricity. It can be filled up and drained at will, and the voltage output of the integrator is analogous to the amount of water in our bucket. The speed at which the bucket can be drained or filled is controlled by the VCA. To be more specific the VCA actually creates a current output that allows C3 to be charged up by the op-amp U6. The VCA’s current output is dependant on two things; one, the current through R6 and the voltage at the VCA’s control pin, pin 3. In fact, the current through R6 is multiplied by the mathematical logarithm of the voltage at pin 3.

Now pin 3 is directly controlled by the front panel pot and any external CV. This way the speed of the integrator can be manually and automatically adjusted. The voltage at pin 3 varies from +341mV (lowest frequency) to -75mV (highest frequency) when controlled by the front pot alone. The gain of the VCA goes up the more negative the voltage is. Unfortunately, if the gain of the VCA gets too high, then the whole filter will become unstable and an oscillation is seen at the output of U6b (pins 5, 6, 7). However, in this design this will only happen if the frequency pot is turned up high and a large positive external CV is used. As it happens the cut-off frequency at the point of instability is well above the audio range so this should never become a problem in use.

The current through R6 is dependant on the voltage output of op-amp, U6b (pins 5, 6, 7). This op-amp is a differential amplifier and this particular circuit subtracts the input signal with the output of the integrator. This now means that the integrator has to respond to both the input signal and its own output. The upshot of this is that the integrator’s output will try and follow the input signal, but only as fast as the VCA will allow that capacitor to charge and discharge. This is the classic ‘lag circuit’.

This slowness of the integrator’s output to react to the input signal gives the desired low-pass effect. The high pass response is obtained at the output of the differential amplifier. You may want to consider this yourself in more detail. However, a simple way of looking at it is to consider that this output is formed by the difference in the input signal and low pass output. If the low frequencies are taken out of the input signal, then you are left with only the high frequencies, and thus, the high pass response.

It should be noted that a one pole filter creates a roll off of around -6db/octave past its cut off frequency. So the effect of the filter is less pronounced than ones of higher orders like Moog Ladders and so on. It is not possible to make a one pole filter have gain at its cut-off point, so there is no chance of a resonance control or self-oscillation at its cut-off frequency.

Now let us look to the first page of the schematic. This covers the noise generator, the pink noise filter and the unstable chaos of the infra-red output circuity.

The noise generator is Q1 and I will discuss choosing this device in more detail later in this document. All devices produce noise of some sort. This is caused by the random movement of charge carriers within the semiconductor (or electrons in metals) at temperatures above absolute zero. At room temperatures, things are really shaking! Various devices can be utilised for their noisy behaviour. Zener diodes are very noisy particularly around their reverse breakdown voltage. Transistors also behave as zeners too; the base-emitter junction will have a reverse breakdown potential, which we normally try to avoid like the plague. In this project we embrace this point of the junction's V/I curve, and deliberately force the junction to conduct the wrong way a little bit.

The emitter is made more positive than the base via R28 and the resistor thus provides the reverse ‘emitter’ current. The positive end of R28 is a well decoupled (filtered) version of the +15V supply rail. Any extraneous noise on the power supplies is not wanted here and R26, C17 and C18 filter the power supply accordingly.

The base-emitter junction of the transistor will effectively regulate the voltage at the emitter with respect to ground, since it is behaving as a zener diode. However, this voltage is not that stable. Superimposed on this ‘stable’ DC voltage is a tiny amount of random noise. Noise is a wideband AC signal. It contains many different frequencies. In fact, white noise contains all frequencies (within reason) at the same average power.

C20 allows this small AC signal through while blocking the larger DC voltage. The next stage is a huge gain amplifier, with a massive amplification of over a hundred. The noise signal is around 2mV or so, and this first stage of amplification takes it up to around 200mV. It is very possible that DC errors at the input of the op-amp will produce output offsets. Normally these are very small, but any offsets at the input will also be amplified by the op-amp. C27 removes any DC from the output signal before the next gain stage.

The second stage of amplification should bring the noise level up to around +/-5V. However, due to the variable nature of noise generators in general, the gain of this stage is made variable. The LEVEL trimmer allows the user to set the output signal to a correct level and account for any differences between noise making devices. It is important to set this level correctly, otherwise performance of the infra red output will be impaired. The white noise output is available from this second stage of amplification via a 1K resistor, R36.

Now, you maybe thinking, why do we need two stages to amplify? Surely, we could have just used one big amplifier. Although it is possible to make a single op-amp gain stage of 1000 or more, it has one major disadvantage. As an amplifier's gain rises, we find its bandwidth lowers accordingly. The bandwidth of an amplifier is its ability to amplify all input frequencies accordingly. A small bandwidth means a restriction in useable frequency response. For example, it would be not very useful here to have an amplifier with a gain of 1000 but with a maximum frequency of 4KHz. By using two lower gain amplifiers in cascade, each will have a bandwidth exceeding the audio band, so the overall frequency response of both of our amplifiers will not effect the quality of the audible noise. Even so, I do recommend that you use a good quality fast op-amp here. The LF412 is an excellent choice, and also gives less offset error than a TL072.

To make pink noise we need to create a signal that contains all frequencies as before, but this time with equal power per octave. Without going into the mathematics of this, we need to filter our white noise so that frequencies roll off at a rate of 3dB per octave. Now, normal one pole RC filters produce roll offs not exceeding -6dB/octave. So to make -3dB/octave we must get very clever. Fortunately for me, other people have done all the difficult work and have published their findings. This one I built many years ago from an original design in Wireless World. It was used in an audio test set and produces a very good quality pink noise. Some synthesisers use simpler methods, but this one sounds better to my ears so why not use it.

U9b (pins 5, 6, 7) provides some gain for the losses incurred within the RC network, and the pink noise output is available via R35.

The infra red output is heavily inspired by the Polyfusion noise module. I have made only a few changes to their design, since it works so well.

R31 and C19 provide further low pass filtering of the pink noise and prevent any unwanted switching clicks from the next stage from leaking back into our pink noise output. The first half of U8 (pins 1, 2, 3) is configured as a comparator. Pin 1, the output, will either be at around +14V or -14V depending on the voltages present at its two inputs. Without R41, if the '+' or 'non-inverting' input is at a higher voltage than the '-' or 'inverting input', the opamp's output will be positive. If it is lower, then the output is negative. But R41 provides a whole wedge of *hysteresis*. This means the voltage at the inverting input must be substantially higher (or lower) then the non-inverting input to get the op-amp to flip states. This prevents the opamp's output from juddering when the two inputs are very close to each other. The signal we see at the output of U8a is a rectangular one with random flips from positive to negative. We will talk about R29 and the offset pot later.

The next stage is an interesting low pass filter arrangement. Not a traditional low pass filter, it has two distinct cut-off frequencies at 0.2Hz and 0.04Hz. Suffice to say, its output will respond very slowly to any changes at its input. The rapidly changing output of U8a is thus turned into slow sluggish random movements by this 'lag' circuit. Further feedback is provided by R38 which feeds back this slow moving output to the comparator. This increases the randomness of the signal. I found experimenting with this value produced some unexpected results.

The Offset trimmer is provided to keep the overall output centred around 0V. Two things seem to affect the offset compensation. One; the offset of the op-amp. I have chosen to use a 1458 as in the original design. This does not have the best offset specifications, but does have a limited slew rate (the speed the op-amp's output stages can swing from one voltage to another) which will act as a further low pass filter and prevent any fast edges to contaminate our clean analogue design. And two; the pink noise will contain a certain degree of imbalance which this circuit is sensitive to. This seems to be affected not by the DC offsets of the pink noise output amplifier but by the spectra of the noise itself after filtering.

The output of U8b is typically around 5V to 10V peak. The value of which seems to depend on the quality of the noise source. After further low pass filtering with R34 and C21, the random output is fed to an amplifier to buffer the signal before the final output. The gain of this amplifier is set to just one or 0dB. There is some scope to alter this should you need a bigger IR output. Making R39 a 10K resistor will double the output voltage.

Components

All of the parts are easily available from your local parts stockist with the exception of the PCBs and the pot brackets. Rapid Electronics, RS Components and Farnell, here in the UK. 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. However, I would go for 1% 0.25W metal film resistors throughout, since these are very cheap nowadays.

The pots are Spectrol 248 conductive plastic types and are held onto the boards with specially made Oakley pot brackets. Two pot brackets each are required for main board and the pot board. These are provided with the 'pot bracket kit' which also contains the four extra nuts required to correctly fit the panel to the boards. Farnell, CPC, and Rapid sell the pots but not the brackets.

The six electrolytic capacitors are radial types. These are the types that are cylindrical in shape with their legs sticking out of the bottom. I use bog standard 2.2uF, 63V and 22uF, 25V or 35V types for this non critical situation.

The PCB is yet another Oakley board to allow you to incorporate axial multilayer ceramics for the power supply decoupling. These are good components with an excellent performance. Various types exist but I tend to use the better quality C0G types from Farnell. They look like small yellow or orange beads with wires that stick out on either side like a resistor. They normally come delivered on tape. You can use 63V or 100V types.

The polyester capacitors are actually metallised polyester box types. These have a lead pitch of 5mm (0.2”) and come in little plastic boxes in a variety of colours. Typical working voltages will be 63V, 100V or even 400V. They key thing to look out for is physical size and lead pitch. Most of Rapid’s are either 63V and 100V and all will fit on the board.

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

L1 and L2 are radially mounted ferrite beads. These look a little like black resistors. They are usually in the EMC or Filtering section of your components catalogue. Farnell sell them as part number: 9526820 (latest RoHS compliant component). Rapid sell them as part number: 26-4860.

Q1 is the noise transistor. I tried many different types of transistor. I found BC548s and BC547s to be best. I have no idea why they should sound better then the BC549s or any other I tried, but they did. I didn’t like the sound from all the BC550s I tried at all. I suggest buying five BC547s and try each one in turn. I’ll talk more about this later in the document.

All ICs are dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP or a CN in their part numbers. For example; TL072CN and LF412CN. Do not use SMD, SM or surface mount packages. They do not fit at all.

The LM1458 is a real oldie, and some places don’t stock it anymore. In the UK, Farnell and RS sell the little chap for peanuts.

The THAT2180LC is available from Rapid Electronics or www.profusionplc.com. You can also use THAT2180LA and THAT2180LB. These are the same part but with lower offset voltages and a higher price.

The two trimmers are a single turn cermet type. This type of trimmer has a different PCB footprint than other types you may have used. Lead spacing is 0.2” for the track ends, and the wiper is 0.2” away above a line joining the other two pads.

The six way jumper interconnect is a one piece assembly bought ready made from several places. I buy mine as an eight way 55 mm or 80mm long piece which I then cut down to six way with a sharp knife or decent pair of scissors. These come as pre-stripped and even tinned with solder too. Make sure you get the 0.1” (2.54mm) pitch variety. The Rapid part number is 22-1650 for the 55mm and 22-1655 for the 85mm one.

For the 0.1” interconnections I use the 26 awg MTA parts. These are made by Amp, now part of the massive Tyco empire. To use these effectively you need a special tool to poke the wires into the special ‘housings’ The housing contains specially shaped contacts that cut through the insulation of the wire so you don’t need to do any stripping. Just simply push down on the wire with the tool to lock it into place in the housing.

You can also use Molex strip and crimp systems. More about this one later on.

Name	*Farnell Part number*	*Amp's part number*
Five way housing	1098713	640442-5
Five way header	588-600	640456-5
Six way housing	1098714	640442-6
Six way header	588-611	640456-6
Handtool	589-494	59803-1

The special handtool you need is rather expensive if you are only doing one module. But following the success of our trials we will be using the MTA system for all new boards and new issues of older boards.

Input and output sockets are now board mounted in the standard module assembly. You could of course use any types if you would like to wire your sockets up with individual wires.

Switchcraft 112A 1/4" sockets are excellent parts, also used on the Moog and MOTM modulars. The version you need to fit in the socket board is the 112APC. This part is stocked by most suppliers. Both Rapid and Farnell sell is at a reasonable price. Rapid also sell a Far Eastern clone of the 112APC part that is considerably cheaper, but doesn't look as nice. Please note that the standard 112A will not fit easily into the boards as it has solder tags.

UK builders should know that there is now a 'Oakley Preferred Parts List' online. This gives the part codes of our most used parts. It can greatly speed up ordering times. This can be found at **www.oakleysound.com/parts.pdf**.

North American builders should visit **www.wiseguysynth.net** for more information about buying parts in the US.

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

Parts List

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

Resistors

Resistors 1/4W, 5% or better.

22R	R40, R53
100R	R26, R57

330R	R56
620R	R34
1K	R32, R36, R55, R35, R43, R14, R11, R12, R2, R27
1K2	R33
1K5	R50
3K3	R54
4K7	R49
5K1	R4, R3
6K8	R52
10K	R29, R42
15K	R18, R16
18K	R41, R6, R5
36K	R22, R20
47K	R46, R23, R15, R19, R10, R8, R13, R17, R25, R1, R21, R24, R7, R9
51K	R38
100K	R28, R37, R31
150K	R47
470K	R30, R44, R51
1M	R58, R48, R45

R39 is not fitted and the space on the PCB should be left empty.

Capacitors

100n axial multilayer ceramic	C18, C16, C5, C23, C25, C4, C24, C26, C21, C9, C8, C7, C6, C10, C11
4p7 low-K ceramic plate	C15, C13
470pF low-K ceramic plate	C37
2n2, 63V polyester	C3, C2
10nF, 63V polyester	C14, C12
33nF, 100V polyester	C34
100nF, 63V polyester	C36, C19
330nF, 63V polyester	C33
1uF, 63V polyester	C27, C30, C20, C35
2u2, 50V polyester	C31
2u2, 63V electrolytic	C1, C28, C29
22uF, 25V electrolytic	C17, C22, C32

Semiconductors

TL072CP dual FET op-amp	U3, U4, U5, U6, 9
THAT2180LC voltage controlled amp	U2, U1
LF412CN dual precision FET op-amp	U7
1458 dual op-amp	U8
BC547 *see text*	Q1

Trimmers

20K cermet	ADJ
50K cermet	LEVEL

Other

50K linear single gang pot	Frequency (HPF and LPF), CV depth (HPF and LPF)	
Oakley/Spectrol pot brackets	4 off for the above	
Spectrol nuts	4 off for the above	
4-way 0.156" MTA header	PWR	1 off
5-way 0.1" MTA header	NSE (Main PCB and socket PCB)	2 off
5-way 0.1" MTA housing	NSE cable	2 off
6-way 0.1" MTA header	LP/HP (Main PCB and socket PCB)	2 off
6-way 0.1" MTA housing	LP/HP cable	2 off
8-way (cut down to 6) jumper lead	AUX interconnect	1 off
Leaded Ferrite beads	L1, 2	2 off
Sockets	Switchcraft APC112	9 off

Around 2 m of insulated multistrand wire (26awg)

Power lead MTA to MTA connector

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

Populating the Main Circuit Board

Warning:

Oakley Modular PCBs are now supplied with the RoHS compliant Lead Free HASL finish. This is a high quality finish but does possess slightly different soldering characteristics to the traditional lead based HASL finish. Handle the boards with care, and avoid touching the tracks that can cause premature tarnishing of the finish. Shelf life is hard to predict but we recommend soldering in all the components less than one year from when you receive your board.

These boards can be soldered with either leaded or lead free solder. However, you should be aware that lead solder is toxic. Always wash your hands after handling solder and never put solder, or any products containing solder, in your mouth

We are not responsible for any accidents caused whilst working on these boards. It is up to you to use your board responsibly and sensibly.

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

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

For construction of the PCB I use water washable flux in leaded solder. The quality of results is remarkable, although you should remember that boards made this way are not RoHS compliant and would fall foul of the law should you decide to sell your unit on a commercial basis. In Europe, Farnell sell Multicore's Hydro-X, a very good value water based product. You must wash the PCB at least once an hour while building. Wash the board in warm water on both sides, and use a soft nail brush or washing up brush to make sure all of the flux is removed. Make sure the board is dry before you continue to work on it or power it up. It sounds like a bit of a hassle, but the end result is worth it. You will end up with bright sparkling PCBs with no mess, and no fear of moisture build up which afflicts rosin based flux. Most components can be washed in water, but **do not** wash a board with any trimmers, switches or pots on it. These can be soldered in after the final wash with conventional solder or the better new type of 'no-clean' solder.

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

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

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

The polyester film capacitors are like little coloured boxes. Push the part into place up to the board's surface. Little lugs on the underside of the capacitor will leave enough of an air gap for the water wash to work. Cinch and solder the leads as you would resistors.

The 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 ICs have pin 1 towards the top of the board. This includes the narrow single in line (SIL) THAT2180 parts. Pin 1 is depicted by a notch in top surface of the IC’s packaging.

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.

Do not fit the transistor at this stage.

The 0.1” headers are fitted on the main board so that pin 1 is towards the right hand side of the board. The clip on the side of the header will match with the picture on the PCB legend and should be towards the top of the board.

The 0.156” header should go into the board so that the plastic lock is on the left hand side.

I would make the main circuit board in the following order: resistors, IC sockets, small non-polar capacitors, diodes, transistors, electrolytic capacitors, and connectors. Then the final water wash.

You can now fit the two trimmers.

Do not fit the pots at this stage. The mounting of the pots requires special attention. This will be covered later in this User Guide.

Populating the Socket Board

You have one socket board to populate and the method is a little unusual.

On the board the first things to solder are the headers. These are fitted to the **BOTTOM** of the board and are soldered from the top side. This is obviously opposite to what you are normally used to. The legending is on the top of the board, and the bottom of the board is marked as such in copper on the underside.

Fit both the headers so that pin 1 is the square pin. The friction lock on the header should correspond to the legend on the top, ie. the opposite side, of the board.

The sockets will be fitted on the top of the board, and therefore be soldered on the bottom of the board. You may well find your own way of soldering the sockets, but the way I do it is as follows:

Fit all your sockets into one of the boards. The bevel edge should align with the picture on the board legending. Do not solder them at this stage. Take your front panel and align this over the sockets.

Now carefully place your front panel with PCB and sockets upside down onto your bench [or kitchen table!]. The holes where the sockets will be should hang over the edge of the bench so that the sockets aren't forced back up through the holes. You'll also probably need a small counter weight to stop the panel from falling over the edge. Now allowing the PCB to rest flat on top of the sockets, you can begin to solder all the pins to the board.

Once all the sockets are soldered, you can then flip the panel over and fit the socket's washers and nuts.

Those of you who have built older Oakley modules will probably be stunned how easy this was compared with the making of wire frames done previously.

Mounting the Pots

NOTE: This procedure is rather different to that of the Omeg/Piher pots you may have used on older Oakley boards.

The first thing to do is to check your pot values. Spectrol do not make it that easy to spot pot values. Your pot kit should contain:

Value	**Marked as**	**Quantity**	**Location**
50K linear	M248 50K M	2 off	Main PCB
50K linear	M248 50K M	2 off	Pot board

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. Leave off the washer for later. 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 PCBs. 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 both pots and snip off any excess wire from the pot's pins at this point.

The Spectrol pots are lubricated with a light clear grease. This sometimes is visible along the top of the mounting bush of the pot body. Try not to touch the grease as it consequently gets onto your panel and PCB. It can be difficult to get off, although it can be removed with a little isopropyl alcohol on a cotton wool bud.

You can now present the front panel up to the completed boards to check that they fit. If it does, I then mount the boards up proper. 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.

Note that the pot shafts of the four pots will not need cutting to size. They are already at the correct length.

Connections

The first thing that will need doing is the interconnection between the pot and main board. This is done with a 55mm or 85mm long 6-way jumper connection.

If you have bought the recommended part from Rapid you will find it is an eight way jumper. You'll need to chop of the last two conductors with a sharp knife or a pair of scissors. Then simply thread all six wires into six holes of the AUX pads from the underside of the pot board. The other end of the jumper can then be gently coerced into the AUX pads of the main board. Solder each end of the jumper. You must solder the pot board from the top of that board, and the main board from the bottom of that board.

That's it that the two boards joined up. Wow, that was easy!

If you are using the recommended MTA interconnections this section will be very easy indeed. All the wiring between the sockets and the main board is done with one 5-way jumper and one 6-way jumper. Here you will be using either the MTA system or the slower, but cheaper, Molex system.

Make up the 5-way interconnect first. This should be made from wires 110 mm long. Make sure you get pin 1 going to pin 1 on the other housing, pin 2 to pin 2, etc. This cable will connect to the headers called NSE on each board.

The second lead is a 6-way interconnect. This is made up to be 110 mm long. This should connect the LP/HP headers on the socket board and the main board.

The Molex Alternative to MTA

For those of you who want to use the cheaper Molex system, the following information may be useful:

A quick note on the 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 the correct length. 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.

Power Connections

The power socket is 0.156" Molex/MTA 4-way header. Friction lock types are recommended. This system is compatible with MOTM systems.

Power	***Pin number***
+15V	1
Module GND	2
Earth/PAN	3
-15V	4

The PAN pad on the PCB has been provided to allow the ground tags of the jack sockets to be connected to the powers supply ground without using the 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.

Front Panel

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 of Berlin, Germany, or Front Panel Express in the US. The cost is about £20 per panel. All you need to do is e-mail the appropriate fpd file that is found on the Noise web page on Oakley Sound site to Schaeffer or FPE, and they do the rest.

The database 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 or FPE web site.

The pot spacing on the PCB is equivalent to the vertical spacing of the MOTM series of modules, but the socket and horizontal spacings are very different.

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

Apply power to the unit making sure you are applying the power correctly. Reversal of the power's polarity will usually destroy all the op-amps in an instant.

If you can monitor current, check that it doesn't exceed 25mA or so from each rail. If you can't check current directly, check with your finger that none of the op-amps or VCAs are getting warm.

You will now need a audio VCO and an LFO of some sort. Some form of audio monitoring will be needed too so you can hear the output of the Noise/Filter module.

Connect the audio output of the VCO to the IN socket of the LOW PASS filter. A sawtooth wave in the bass registers is best. Listen to the audio output from the OUT socket. Adjusting the FREQUENCY pot you should notice the sound getting duller as you turn the pot anti-clockwise. This should sound like simple filtering, which is exactly what it is. At the maximum point the output of the filter should sound more or less the same as the input signal. The minimum point should just let through a dull rumble.

Set up an LFO so that its frequency is at around 1Hz. Now attach the LFO's sawtooth output to the CV IN socket. Set the FREQUENCY pot to its middle position and rotate the CV DEPTH pot. The cut-off frequency should now be being modulated with the LFO. Check that the minimum modulation occurs in the middle position of the CD-DEPTH pot. You should have a 'dow, dow, dow' sound with the pot in the '+' position, and 'yit, yit, yit' in the '-' position.

Now we need to test the high pass filter. Repeat the above section to check whether the high pass filter is working. This time as you increase the FREQUENCY pot the output should get 'fizzier' and lose all of its bass end. Check also the CV input for the high pass filter. Again, check that the modulation is at its minimum when the CV DEPTH pot is in the middle position.

If this is all correct you almost certainly have a working module. However, we can't check the noise section yet because we haven't inserted the noisy transistor.

If you have found a fault, then the usual course of action is to try and find the wrong component. It'll usually be a wrong value resistor, like swapping a 100K resistor for a 100R.

Other things to check are the connections between the socket and main boards. Make sure too that your headers have been fitted the right way.

Selecting your transistor and final setting up

As soon as you know the filters are working the next thing you need to do once you have built up your module is to fit the noise transistor. You should have a few BC547, BC548 or BC549 NPN transistors ready. You can try each one in turn into the Q1 position on the PCB and see which one sounds best.

Connect your power supply up to the module, and attach your monitoring amplifier to the white noise socket. Keep the volume levels fairly low, because as you remove and add the transistors it will thump a bit.

For the each transistor 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. For now there is no need to solder it in place.

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.

Once the transistor is inserted the circuit will take time to settle. After a few seconds you should hear the tell tale hiss of a reverse biased transistor junction.

You are listening out for the best quality noise sound. Don't get too concerned with overall loudness as some of the quieter ones actually sound better. You may need to alter the LEVEL trimmer so that you get an even tone. If the LEVEL trimmer is set to high then the noise pre-amps may clip and that won't sound very nice either. There's no hard and fast rules here, just enjoy the process. If you have a 'scope then you can monitor what's going on by looking at the white noise output. You'll find that the best noise sources are the ones with an even voltage swing about 0V. Bad sounding devices tend to have more positive voltage spikes. Once you have found a nice sounding transistor, switch off and solder the little chap in.

There are just two trimmers to be set before you are finished. As we have seen the LEVEL trimmer allows you to trim the output of the white noise and also the pink noise. Monitor the white noise with an oscilloscope. Adjust LEVEL until the peak signal is around +/-6V. If you don't have a 'scope, then just adjust it until you get the roughly the same volume as one as your modular's VCOs.

The OFFSET control can be set in a variety of ways. What we need to do is to set the average value of the IR output to zero. One way is to use a 'scope and try and get equal swings either side of zero volts. Another way, is to use your MultiMix output LEDs. Adjust until you get equal 'green' and 'red' times. Beware, the infra red output is very random and very slow. Be prepared to spend around a minute in waiting for the output to settle to the new OFFSET adjustment.

If you find you cannot get the IR output to get away from either supply rail, then please re-adjust the LEVEL trimmer to give a higher level of noise to the IR circuitry. Too little pink noise will mean you haven't got enough signal to flip the comparator U8a.

Final Comments

I hope you enjoy building and using the Oakley Noise and Dual Filter module.

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

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

If you still can't get your project to work, then Oakley Sound Systems are able to offer a “get you working” service. If you wish to take up this service please email me, Tony Allgood, at my contact e-mail address found on the website. I can service either fully populated PCBs or whole modules. You will be charged for all postal costs, any parts used and my time at £20 per hour. Most faults can be found and fixed within one hour. The minimum charge is £20 plus return postage costs.

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

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

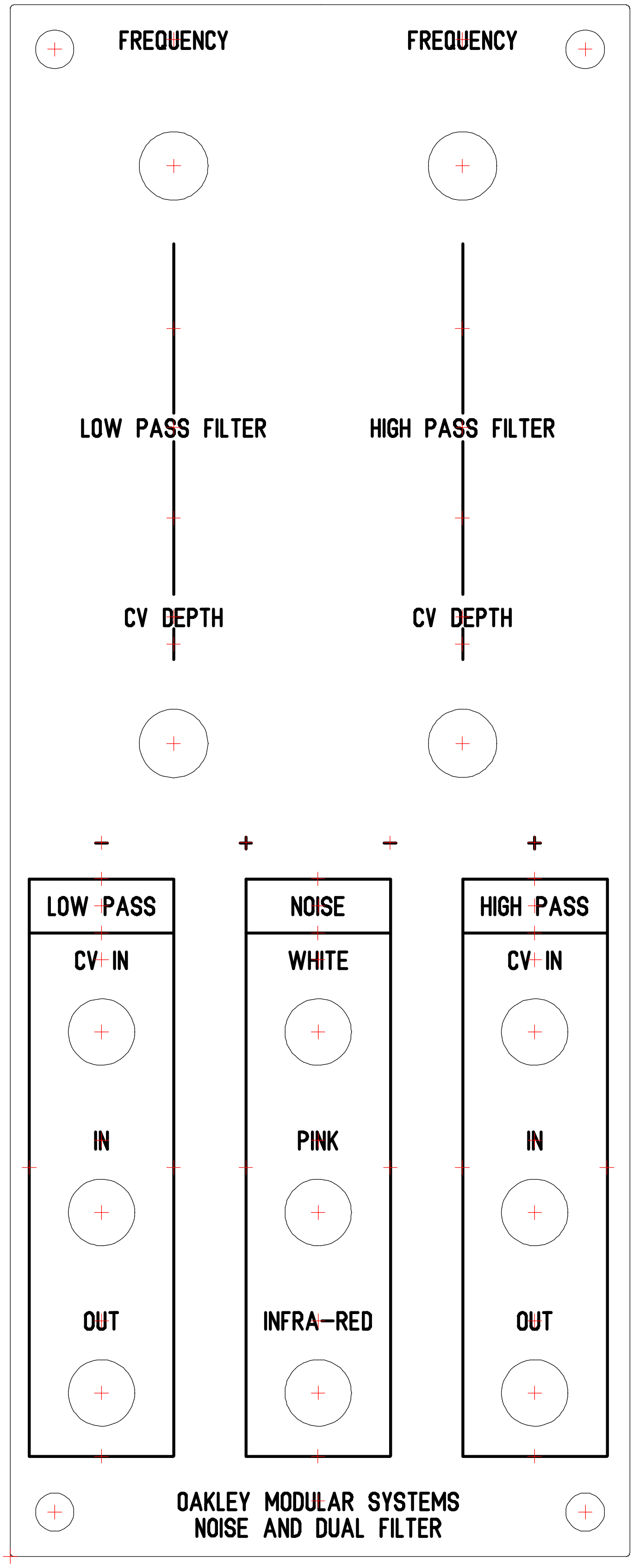
Tony Allgood

Cumbria, UK
July 2007

Version 2.2

Formatted on Open Office 2.0

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


FREQUENCY
FREQUENCY
LOW PASS FILTER
HIGH PASS FILTER
CV DEPTH
CV DEPTH
LOW PASS
CV IN
IN
OUT
NOISE
WHITE
PINK
INFRA-RED
HIGH PASS
CV IN
IN
OUT
OAKLEY MODULAR SYSTEMS
NOISE AND DUAL FILTER