Oakley Sound Systems

Oakley Modular Series

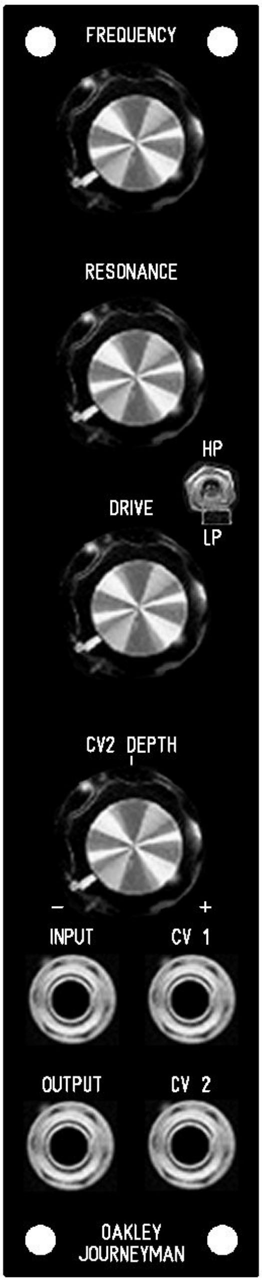
Journeyman issue 1

Vintage Voltage Controlled Filter Module

User's Guide

V1.07

Tony Allgood B.Eng
Oakley Sound Systems
CARLISLE
United Kingdom


FREQUENCY
RESONANCE
HP
DRIVE
LP
CV2 DEPTH
–
+
INPUT
CV 1
OUTPUT
CV 2
OAKLEY
JOURNEYMAN

Introduction

If our Superladder and SVF are well behaved perfect sounding filters, then this one is the rude boy of filter design. Its completely non-linear response and discrete circuitry gives it a unique sound of its own. Modelled after a classic Japanese filter design of the mid to late 1970s, the Journeyman can be configured in either high pass or low pass modes by the flicking of a switch. When two of these VCFs are used together, one in HP and the other in LP, then the effect is astounding.

The filter heart is made entirely from a single ended discrete design and uses four matching diodes as the control elements. The filter behaves differently depending on the input signal level hence the 'drive' pot on the front panel. Turning this up allows the filter to be overdriven. At full resonance the filter will oscillate across most of the audio band.

The design is intended to fit into a 1U wide 'filter-core' module or a more fully featured 2U wide panel with seven control pots.

The 1U 'Filter Core' format is our way of handling filter modules. Although the 1U module can be used as a filter module on its own, it is expected that users will make use of external mixers to control CV and audio levels going into the filter. In this way, you will be able to have a collection of space saving 1U filter cores that can be used with any generic mixer module. The Oakley Multimix is an ideal choice for a handy mixer module.

For the 2U design three audio inputs are provided each with its own attenuator. Three CVs can control the cut-off of the filter. One is fixed at approximately 1V/octave, the other two have input attenuators. CV2 features a reversible type attenuator with inverting/non-inverting properties.

The Filter Core Idea

As you have read this module can be made into either a standard 2U wide module, or as a compact 1U filter core module. The filter core series of modules is our new way of presenting filter designs and it is the suggested way of making your module. This is especially true if you have not built any of our modules before since the filter core version is a lot easier to make.

The Filter Core idea has come from the fact that many of our customers were buying different filter types, eg. they may have an MS-20 clone, a Moog ladder filter and an SVF. Each filter type gives a different sound so its worthwhile having a few in your modular set up. However, each filter module also has its own input mixer for audio and an input mixer for CVs. This adds to panel real estate and soon your modular is filling up very quickly. While this does look very impressive, it does mean that, in many patches, you have a lot of redundant electronics in your modular.

Step forward the 'filter core'. This is quite simply a 1U module that contains only the filter and a few important front panel pots. All the audio and CV mixing is done externally with a dedicated mixer module, like the Multimix. The good thing about this is that any unused filter module is only 'wasting' 1U of panel space. So you can afford to have many different flavours of filter without the additional cost and panel space of mixers.

However, as with all things, there are disadvantages too. The lack of inbuilt mixers mean that you will need to get more dedicated mixer modules. But remember that these relatively cheap mixer modules can be used for **any** mixing or level controlling within your modular. Thus, you have more flexibility, at the expense of a little more patching.

The great thing about the new Oakley Filter Core modules is that they will all be designed so that they can still be used in the full format design. All the Filter Core modules will have input summing amplifiers built onto the PCB. You won't be using these circuits in the 1U format, but they are there if you want to go for the larger 2U or 3U designs.

The Filter Core panel design is a lot easier to make of course. It only has four sockets and all the wiring is done by using some big solder pads at the bottom of the PCB. For the 2U format, you will need to use these pads and some additional 0.1" headers which are placed near the pots. In a possible future addition to our range, we may provide special pot PCBs to directly attach to these headers to make building the larger modules a lot easier. These pot PCBs have proved very successful in the VCO and other modules where we use them.

The Journeyman issue 1 PCB

I have provided space for the four main control pots on the PCB. If you use the specified Spectrol pots and matching brackets, the PCB can be held firmly to the panel without any additional mounting procedures. The pot spacing is 1.625" and is the same as the vertical spacing on the MOTM modular synthesiser.

The design requires plus and minus 15V supplies and the power supply should be adequately regulated. The current consumption is about 35mA for each rail. Power is routed onto the PCB by a four way 0.156" Molex type connector. You could, of course, wire up the board by soldering on wires directly. The four pins are +15V, ground, earth/panel ground, -15V. The earth 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.

The PCB has four mounting holes for M3 bolts, one near each corner. These are not required if you are using our specially made pot brackets.

Circuit Description

The schematic is split up in little sections. The power supply input circuitry is shown in lower left corner. The module is powered in the conventional way from a split rail of +/-15V. This comes in to the module via the 0.156" MTA connector PWR. Low value resistors F1 and F2 in conjunction with C13/15 and C14/16 provide high frequency filtering and decoupling. They essentially act to keep the power supply as free from noise as possible.

The power supply to the three dual op-amps are shown separately so as to avoid to additional wiring on the main part of the schematic. C4 and C5 provide local decoupling to U2, the main audio op-amp.

The main filter core is shown in one large lump of a schematic at the top of the page. Its a very unusual design when compared with more modern IC based designs. But it can be simply looked at in three basic stages, CV processing, the filter core and the input stage.

The input stage is built around one half of U2 (pins 1, 2, 3), a high quality op-amp. Its a standard inverting summing circuit. This particular circuit will sum up to three audio signals together and present them directly into the filter core via SW1. In the 1U layout, we only use one of the inputs since any signal mixing will take place in an external signal mixer module such as the Multimix. However, in the 2U version, an additional two inputs can be utilised to provide on board mixing. The overall gain of the mixer is controlled via the 'Drive' pot, this can vary the gain from -0.22 to -0.72. The minus sign here indicates that the summing process is inverting, that is, positive going signals are turned into negative going ones.

SW1 is a two pole change-over switch and is configured to allow the summed input signal to go to into the filter core via either C10 & C11 or C20. If the signal enters via C10 and C11, then the overall response of the filter is low pass. While with the latter, the signal is high pass filtered. Note how the larger value electrolytic capacitor, C11, is in parallel with the polyester capacitor, C10. This supposedly reduces the high frequency distortion due to the non ideal behaviour of the electrolytic capacitor. However, in this circuit it is probably completely superfluous thanks to the large amount of other linearities in the circuit. I thought I would try it though – but feel free to omit C10 (and C18) if you wish. Don't omit C11 instead though, it needs to be that big so as to allow low frequencies through.

The stage based around Q1, 2 & 3 is the CV processing side of the filter core. This takes a control voltage, splits it into two phases via Q1, and then level shifts and buffers the two outputs with complementary circuits based around Q2 and Q3.

A CV signal arrives at Q1 via base resistor R12. R12 sets the overall sensitivity of the filter. R22 and R14 bias the base of the transistor to half the supply voltage. In our case, this is 7.5V. It does this because for the most part the filter core works off a +15V single rail supply, it doesn't use the -15V rail. This means that all the signals swing about a positive bias voltage – there are no negative signal levels, merely ones that are less positive.

Q1 is a phase splitter. In other words, the signal is split and each output is completely out of phase with the other. That means, if the voltage on the collector, pin 1 of Q1, is going in a positive direction, the voltage on the emitter, pin 3 of Q1, will be going negative. We call this a differential signal. The BAL trimmer sets the operating current of the phase splitter and hence will control the signal level on the collector. This is trimmed so that the two voltages control each filter section equally. Any imbalance in the two control signals results in more control feedthrough. This is where the controlling CV imposes a fraction of itself on the audio output. To much of this feedthrough means that fast sweeps of filter frequency end up with unwanted small clicks at the output.

Q2 and Q3 are complementary voltage followers that provide buffering and level shifting for the next stage. However, it should be noted that changes in temperature will affect the accuracy of these stages. The gain doesn't change, but the average output voltage does a little, thanks to a change in Vbe, the base-emitter voltage. This means the whole filter circuit has a certain amount of temperature 'instability' with regards to the actual cut off frequency. I would say that this lends the design a certain degree of 'vintageness'. In truth I could have replaced the whole Q1, 2 & 3 section with a dual op-amp and some resistors, but I was keen to replicate the behaviour of the original circuit as much as possible. Indeed, the non zero and non symmetrical output impedance of these stages must surely have an affect on the sound of the filter. Interestingly the original Japanese manufacturer did update their filter design to use op-amps in the CV control path in the later MS-50 synthesiser expander module.

The phase split signals now go to two diodes in the form of U4 (pins 5, 6, 7) and U4 (pins 8, 9, 10). These diodes are each made from one junction of an NPN transistor, itself part of a quad transistor array. Both these diodes are matched to each other since they exist on the same die within the THAT300 array. These diodes serve to turn the linearally changing voltage output of the phase splitter to an exponentially changing current. It is this current that is used to drive the next part of the circuit.

The main part of the filter core consists of what is sometimes called a diode ring. In my schematic this is less clear since I have used transistors wired as diodes. However, it is indeed a standard diode ring with four terminals in use, the differential control current being applied to the top and bottom of the ring. Each of the diodes in the ring acts as a variable resistor. The more current that goes through it, the less its 'resistance'. These diode 'resistances' form two resistor-capacitor (RC) networks in conjunction with C8, 9 and C20. This seems initially confusing since we have four diodes and three capacitors. The first network in one built from U5 (pins 1, 2, 3) and C8, along with U5 (pins 5, 6, 7) and C9. These two sets of diode-capacitors act as one RC filter since the current driving the diodes is differential and the audio signal comes in through both of them in parallel. The other RC filter is made from U5 (pins 12, 13, 14), U5 (pins 8, 9, 10) and the capacitor C20. C19 is not part of the RC network, its job is to simply pass the audio signal to the next stage.

The filter elements are actually part of a larger circuit involving the remaining parts of the NPN array and Q4 and Q5. These transistors and their associating circuitry form an amplifier. They are wired to the filter network to form a Sallen and Key filter. You may have seen S-K filters before, but they usually consist of two resistors, two capacitors and an op-amp. Sometimes, the op-amp will have some additional gain setting resistors around it. In this vintage circuit, the op-amp is replaced by an all discrete amplifier, but the basic action is much the same. I will not go into detail about this here, as the maths is way too complex for this simple overview, but there are plenty of online resources available to look at if you want to know more.

The key thing to understand is that the diodes that make up the diode ring will alter their resistance as the current through them changes. This change in resistance causes the two RC networks to change the way they pass audio signals. Now an RC network can act as a low pass filter or high pass filter depending on how it is configured. Changing the resistance will directly control the cut-off frequency of the low or high pass filter.

In a S-K filter, the cut-off frequency is usually controlled by varying the resistance directly. Each of the two resistors must change similarly if one is to maintain an even filter response, that is, a certain filter slope. In a manually controlled S-K filter, this would done with a dual gang pot. In our diode ring each 'resistor' will change equally since all four of them have the same magnitude of current running through them and they are perfectly matched thanks to the characteristics of the THAT300 array.

Part of the S-K topology is that the output signal is passed back into the filter network. The amount of signal passed back controls the resonance of the filter. An increase in the resonance of a filter circuit leads to a pronounced hump or peak in the response of the filter at the cut-off frequency. This peak leads to the characteristic synthesiser swept filter sound. The RES pot controls the level of the feedback signal. It does this in a very crude way. It simply shorts out the signal being fed into a simple voltage follower circuit based around Q5. A voltage follower is a circuit that replicates the signal at its input to its output. The benefit being that the output signal is isolated from the input, thus current can be drawn from it with no effect on the input signal.

The audio output of the filter's amplifier section is fed via C17 and C18 to the module's main output amplifier, built around U2. U2 is a standard op-amp inverting circuit that provides sufficient gain and low output impedance to drive any connected modules.

The remaining parts of the Journeyman to discuss are the CV processors. These allow the various CV inputs to be summed and controlled before being passed on to the phase splitter circuit already discussed.

The circuitry based around U3 is a reversible attenuator. This is quite a common little circuit block in my designs. It allows the gain to be varied from -1 to +1. In other words, a CV input may be controlled in level from inverting, to off, to straight through. Its easy to see how it works. One end of the pot is driven from a voltage follower based around U3 (pins 5, 6, 7). This is a simple circuit that presents at its output a copy of what it 'sees' at its input. The other end of the pot is connected to an inverting amplifier based around U3 (pins 1, 2, 3). This simple circuit inverts the input voltage, eg, 1V becomes -1V and -2V becomes 2V. The wiper on 'depth' pot can be moved from inverting at one end, to non inverting at the other. In any position in between, the voltages from each op-amp are combined in different degrees. At the centre the wiper receives signals from both op-amps in equal proportions and the two cancel out.

The main CV summing amplifier is based around U1. The first section sums the various CV inputs together, including the output from the reversible attenuator. In addition, two control voltages are created from two pots; one, a trimmer that allows fine tuning of the cut-off frequency, the other, a standard front panel pot that allows for manual control over the cut-off frequency. The CV and CV1 inputs are configured to give approximately 1V/octave sensitivity. The output from the summing circuit is actually upside down to how we actually want it, so we invert it with a simple inverting amplifier. The actual gain of this amplifier can be trimmed with the 'scale' trimmer. This allows some adjustment in the sensitivity of the CV inputs so as to allow a rough 1V/octave response. It should be noted that, because of the 'vintage' nature of the design, it is unlikely that you will be able to train your filter to this degree of accuracy over the whole of the main audio band. Indeed, you could just link out the 'scale' trimmer and make R30 a 110K resistor and be done with it. I put in the trimmer so if you do have two Journeyman filters in your modular, you can at least tweak them both so that they behave in a mostly similar fashion.

Lastly, since I get occasionally asked this question, I should explain the four little circles marked H1 to H4 in the bottom left of the schematic. These are schematic representations of the four mounting holes on the board and they don't actually have a part to play in the operation of the circuit.

Components

Most of the parts are easily available from your local parts stockist. I use Rapid Electronics, RS Components and Farnell, here in the UK. The Journeyman module was designed to be built mostly from parts obtainable from Rapid Electronics.

The BC550 and BC560 devices are discrete low noise transistors. The former is NPN, while the latter is PNP. You can replace the NPN with BC549, and the PNP with BC559. Quite often you see an A, B or C suffix used, eg. BC550C. This letter depicts the gain or grade of the transistor (actually hfe of the device). The Journeyman is designed to work with any grade device although I have used BC550C and BC560C throughout in my prototype.

The NPN transistor array is the THAT300P, this excellent part is made by That Corporation. It features four highly matched NPN transistors in one 14-pin DIL package. It is available from ourselves, or from www.profusionplc.com in the UK, or www.thatcorp.com in the US.

The U2 op amp is any decent dual audio op-amp. The design has been tested with OP275G, 4558, TL072 and OPA2134A. All these parts work very well. U1 and U3 are used in the CV processing side of the circuit and as such should be TL072.

The three ICs and two transistor arrays are both dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP or a CN in their part numbers. For example; TL072CN. Do not use SMD, SM or surface mount packages.

The board mounted pots are Spectrol 248 conductive plastic types or the newer BI TT equivalents. Either type is held onto the board with specially made Oakley pot brackets. Four pot brackets and an extra set of nuts are required and these are provided with the ‘pot bracket kit’. You could use any pot type you want, but not all pots have the same pin spacing. Not a problem, of course, if you are not fitting them to the board.

In the UK, Farnell, CPC and Rapid Electronics sell the Spectrol pots.

For the resistors 5% 0.25W carbon types can be used for all values except where stated. In these positions I recommend the use of 1% 0.25W metal film types. Bear in mind that 5% types use a different colour code to the newer 1% types. Therefore mixing the two types sometimes causes confusion. It may be prudent to stick to 1% for all values except the 2M2 which is difficult to get in metal film.

All the electrolytic capacitors should be radially mounted. The working voltage (WV) of the capacitor is not critical, but chose one that will fit on the board. I would use 63V for both the 2u2 and 10uF capacitors, and 35V for the 22uF ones.

The pitch spacing of the polyester capacitors is 5mm (0.2”). These types come in little plastic boxes with legs that stick out of the bottom. Try to get ones with operating voltages of 50V, 63V or 100V. They may be called polyester film or metallised polyester capacitors.

The PCB is another Oakley board to feature axial ceramics for the power supply decoupling. These are good components with an excellent performance. The PCB legend for these devices features a lead spacing of 0.3”. Various types of axial ceramics exist. There are the more expensive C0G types from Farnell, but the other types like Y5V and X7R are perfectly good in this application too. I use Rapid part number: 08-0240.

Note the two components labelled as F1 and F2 would normally be ferrite beads on other Oakley modules. However, we have found that better performance can be obtained when you use 10R resistors in place of the beads for this particular project.

The single ‘horizontal’ trimmer is a standard sealed carbon unit. These are adjusted from the top and, as such, are called horizontally mounted types. Piher and other companies make suitable types. Lead spacing is 0.2” for the track ends, and the wiper is 0.4” away.

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.

You will also need a double pole double throw switch. These are sometimes called DPDT or 2PCO (two pole changeover). This type of switch has two sets of contacts inside, each one has a wiper that can move between two other contacts. These sorts of switches have six solder tags. Watch out for sizes, there are many different types of toggle switches and some of them can be very big and will clash with the PCB if using the suggested panel design. I use miniature flat toggle types made by Apem and sold by Farnell.

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.

Finally, if you make a change that makes the circuit better, do tell the ‘Oakley-synths’ mailing list or myself directly. Any updates are added to the current user guide as quick as possible.

UK builders should know that there is a ‘Oakley Preferred Parts List’ online which is updated periodically by myself. This can be found at www.oakleysound.com/parts.pdf.

Parts List

The components are grouped into values, the order of the component names is of no particular consequence. Please read the above section for more details about the parts used in this module.

A quick note on European part descriptions. R is shorthand for ohm. K is shorthand for kilo-ohm. For capacitors: 1uF = 1000nF. To prevent loss of the small '.'as the decimal point, a convention of inserting the unit in its place is used. eg. 4R7 is a 4.7 ohm, 4K7 is a 4700 ohm resistor, 4n7 is a 4.7 nF capacitor.

Resistors

5% 0.25W or better except where marked.

10R	F1, F2
33R	R31
330R	R19, R13
1K	R34
1K5	R45
3K9, 1% metal film	R25, R29
4K7, 1% metal film	R42, R41, R40, R24, R22, R39, R43, R44
5K6	R21
10K	R47, R48, R28, R27, R6
18K	R14, R7
22K	R10
39K	R20
47K, 1% metal film	R35, R36, R18, R11
62K	R49
82K	R12
100K	R23, R17, R5, R2, R33, R16, R30
120K	R26
150K	R4
180K, 1% metal film	R38, R46
220K	R1, R9, R15, R32
470K	R3, R8
2M2	R37

Capacitors

47pF low-K ceramic	C6
4n7, 100V polyester	C2
10nF, 100V polyester	C10, C18, C12
22nF, 100V polyester	C8, C19, C20, C9
100nF axial ceramic	C16, C15, C5, C4, C21, C3
2u2, 63V electrolytic	C22
10uF, 35V electrolytic	C11, C1, C17, C7
22uF, 35V electrolytic	C13, C14

Discrete Semiconductors

BC550 NPN transistor	Q2, Q4, Q5
BC560 PNP transistor	Q1, Q3

Integrated Circuits

OP275GP audio op-amp	U2
THAT300P NPN array	U4, U5
TL072 FET op-amp	U1, U3

Trimmers

20K multiturn	SCALE
100K multiturn	TUNE
500R or 470R horizontal	BAL

Onboard Pots

All pots Spectrol 248 series or BI TT series

10K linear	RES
50K linear	FREQ, DRIVE, DEPTH

Miscellaneous

Double pole changeover switch	SW1
Molex or MTA 4 way header	PSU
1/4" sockets	IN, OUTPUT, CV1, CV2

You'll also need solder, two lots of about 75cm of insulated multistrand hook up wire, each of a different colour, and a couple of cable ties.

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 three 8-pin DIL sockets and two 14-pin DIL sockets.

Additional parts required for the 2U version

50K linear pots	IN1, IN2, IN3, CV1
1/4" sockets	IN2, IN3, KEY CV

Populating the Oakley Journeyman PCB

Warning:

Oakley PCBs are supplied with a RoHS compliant 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 solder plating since this 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.

Neither I nor Paul Darlow are 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. The most common error with most of these was parts inserted into the wrong holes. Please double check every part before you solder any part into place. Desoldering parts on a double sided board is a skill that takes a while to master properly.

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

Sometimes people like to substitute parts in place of my own recommendations. Feel free to do this, but remember that there is normally a good reason why I have selected that particular part. If you do find that, say changing an op-amp with another one, makes an improvement, please do let me know either via the Oakley-Synths list or directly to 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.

IC sockets are to be recommended, especially if this is your first electronics project. Make sure, if you need to wash your board, that you get water in and around these sockets.

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

The smaller electrolytic capacitors are very often supplied with 0.1" lead spacing. My boards have a hole spacing of 0.2". This means that the underside of these radial capacitors will not go flat onto the board. This is deliberate to allow the water wash to work, 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. The PCB legend marks the positive side with a '+', and its the square solder pad. Most capacitors have the '-' marked with a stripe. Obviously, the side marked with a '-' must go in the opposite hole to the one marked with the '+' sign. Most capacitors usually have a long lead to depict the positive end as well.

The discrete transistors are all in the same type of packaging and therefore look the same. Make sure you get the NPN and PNP types in the correct places. Only the numbers on the side will allow you to tell them apart. Match the flat side of the device with that shown on the PCB legend. Push the transistor into place but don't push too far. Leave about 0.2" (5mm) of the leads visible underneath the body of transistor. Turn the board over and cinch the two outer leads on the flip side, you can leave the middle one alone. Now solder the middle pin first, then the other two once the middle one has cooled solid.

Sometimes transistors come with the middle leg preformed away from the other two. This is all right, the part will still fit into the board. However, if I get these parts, I tend to 'straighten' the legs out by squashing gently all the three of them flat with a pair of pliers. The flat surface of the pliers is parallel to the flat side of the transistor.

I would make the board in the following order: resistors, IC sockets, small non-polar capacitors, transistors, electrolytic capacitors. Then the final water wash if you are using water washable solder. You can then solder the trimmers in place, but do not mount the pots just yet. The mounting of the pots requires special attention. See the next section for more details.

The Front Panel

On the website I have included a 1:1 FPD database of the suggested 1U and 2U front panel layouts. Actual panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £17 for the 1U panel, and about £30 for the 2U panel. VAT and the postage is extra, so it usually helps to order a few panels at the same time.

All you need to do is e-mail the fpd file to Schaeffer in Germany, or Frontpanel Express in the US, and they do the rest. You can also use the Frontplatten Designer program's own online ordering procedure which also works very well.

The panel is black with white **engraved** legending. The panel itself is made from 3mm thick black anodised aluminium. The fpd panel can be edited, including changing the colour, with the Frontplatten Designer. The program available on the Schaeffer web site but it should be noted that the program is for Windows only.

Mounting the PCB mounted Pots

NOTE: This procedure is rather different to that of the Omeg/Piher pots you may have used on the 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. You only need three pots. These are mounted onto the PCB directly, and are held in place by our specially made pot brackets.

Value	**Marked as**	**Quantity**	**Location**
10K linear	M248 10K M	1 off	RES
50K linear	M248 50K M	3 off	FREQ, DRIVE, DEPTH

Fit a pot bracket to each of the four pots by the nuts supplied with the pot bracket kit. You should have two nuts and one washer per pot, including the one you got with the pot itself. 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 because if you do the pot shaft will not rotate smoothly.

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 both 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 to check that it fits. However, I usually fit the sockets before I do this, and wire up the ground tags first. Then I mount the board 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.

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

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 cotton wool bud.

Connections

The power socket is 0.156” 4-way header in common with rest of the Oakley and MOTM modules. Friction lock types are recommended.

Power	***Pin number***
+15V	1
Module 0V	2
Earth/Screen	3
-15V	4

The P1 and P2 pads on the PCB has been provided to allow the ground tags of the jack sockets to be connected to the power 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.

Whether you have chosen to make your module in a 1U or 2U format will determine what you do next. The 1U format is considerably easier to build and test. The 2U is more daunting, but if you take your time, you should not find it excessively difficult. I shall deal primarily with the 1U format first since this is the recommended approach.

1U Filter Core module

You have just four sockets to wire up. If you have used Switchcraft 112 sockets you will see that they have three connections. One is the earth lug or ground tag, this is indicated by a bevel in the socket’s housing. The second 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) lug. The NC lug is internally connected to the signal tag when a jack plug is not inserted. This connection is automatically broken when you insert a jack.

In this module we are going to ‘common’ the sockets’ ground lugs. This means that the sockets’ earth lugs are going to be joined together. I normally do this part of the wiring without the PCB or pots in place.

Fit the four sockets onto the panel so that the bevel on the side of the socket is facing top left as you look at the rear of the panel.

The first lugs we are connecting together will be the ground or earth tags on the two upper sockets and then on the two lower ones. 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 two. What you have now done is common each row’s earth tags together, but each row is separate for now.

Fit the Journeyman PCB against the front panel if you haven't done so already. Solder a piece of ordinary insulated multistrand 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 P1. 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 P2. Your earth tags are now commoned together since P1 and P2 are electrically connected together on the circuit board.

Now its time to wire up the four signal lugs to the board. Use multistrand hook up wire to connect each socket's signal lug to the relevant pad on the PCB. Keep your wires short but not too short and you can use as many different colour wires as you can. There is absolutely no need to use screened cable for such short runs.

The connections of the signal lugs of the CV and audio output sockets that go directly to the PCB are summarised below:

Socket name	***PCB pad name***
INPUT	I/P
CV1	CV
OUTPUT	O/P
CV2	CV2

That completes the connections to the sockets. Now we are ready to move onto the switch. This is actually quite tricky, so follow the instructions carefully.

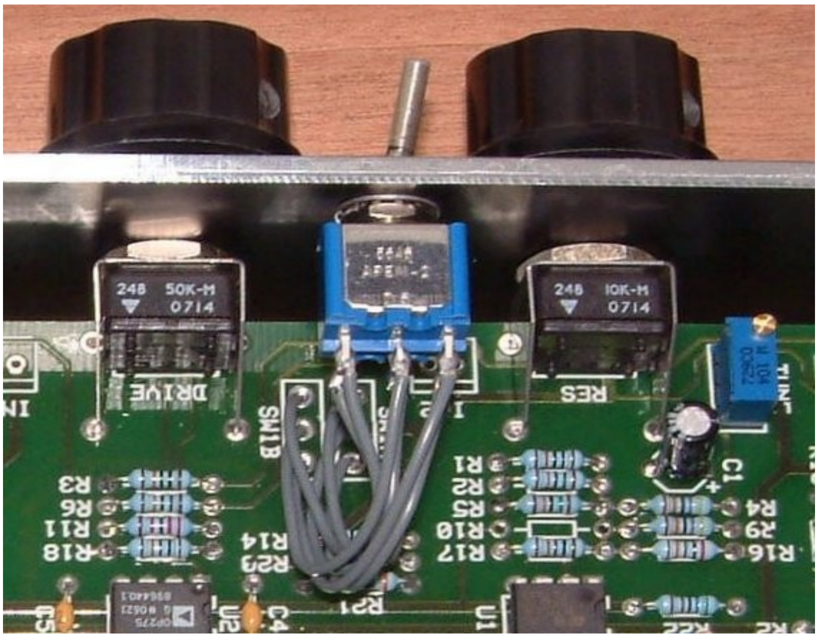
If you are using the suggested front panel, then you will be fitting your switch so that the toggle moves up and down. Lay the module flat so that the front panel is facing away from you and the board is facing upwards. Your fitted switch will have two sets of three contacts, each part of one gang, and these will now be lying horizontally. The wiper contacts of each gang are the two middle pins, one above the other.

Let us wire the bottom set of contacts first. These are to be connected to the pads on the board named SW1A. There are three solder pads in the little box marked SW1A, so each wire is to be connected to its own solder pad. The solder pad nearest the front panel should be soldered to the left hand solder tag, the middle pad goes to the middle tag, and the pad furthest away from the front panel connects to the right hand tag.

Now let us wire the top set of contacts. These are to be connected to the pads on the board named SW1B. Again there are three solder pads in the little box marked SW1A, so each wire is to be connected to its own solder pad. Just like the other gang, the solder pad nearest the front panel should be soldered to the left hand solder tag, the middle pad goes to the middle tag, and the pad furthest away from the front panel connects to the right hand tag.

There's a picture overleaf that shows how I wired the switch up on the prototype board. Notice how I used only grey wires – you may find using different colours would make checking a little easier. I should also add that the component numbering on the prototype is different to your board since the layout on the early board was designed for more experimentation.

That's it, the Journeyman Filter Core module is now ready to test and calibrate.



2U Journeyman full format

I am not going into great detail with this format as the PCB has been designed with the 1U filter core module in mind. However, I will mention a few things that may be useful to you if you do decide to build the larger format design.

The 2U format contains seven sockets and four additional pots. You can use any pots you like, but I am rather partial to the 16mm Alpha pots sold by Banzai. These have nice solderable lugs on them and are therefore easier to wire up than the Spectrol 248 types used elsewhere in the module. One thing to bear in mind though, is that the Schaeffer database I have provided on the site is set up for using Spectrol 248 pots. As such, it has a 10mm diameter hole for the bush. You'll need to make this smaller, to around 8mm, if you are using the Alpha pots. You may be tempted to use the larger 24mm Alpha pots, indeed, these are great pots, but the width of them may mean that the top pot will clash with your choice of mounting rail.

As with the 1U module, you need to ground the sockets' earth lugs. Do this by joining the earth lugs of each horizontal row of sockets together first with stiff single core wire. Then connect each solid wire back to the P1 or P2 pads on the PCB with thin insulated multistrand wire. The P1 and P2 pads are connected to panel ground on the power socket, pin 3 on the MTA/Molex connector.

The connections of the signal lugs of the CV and audio output sockets that go directly to the PCB are summarised below:

Socket name	***PCB pad name***
KEY CV	CV
OUTPUT	O/P
CV2	CV2

All your other connections will be made via the four two way 0.1" headers that are situated on the board near the pots. These are labelled appropriately to help you connect up your module correctly.

The pots have three pins. Two of these pins will be connected to the board, the other one will be connected to the appropriate socket's signal lug.

The middle pin of the pot, the wiper, will carry the signal to the appropriate header on the PCB. The pots' wires will attach to the underside of the board at each header, and thus be soldered from the topside of the board. For each header, pin 1 is connected to the wiper of the pot. Pin 1 is the square pin so its easily spotted even from the underside of the board.

The pot has two other pins, one will be connected to ground, the other to the signal lug on the socket it controls.

With pins facing down and looking at the back of the pot, the right hand pin should go to the ground connection of the header, that is pad 2 on each of the headers. Take a wire from the right hand pin to the round pad on the PCB next to the one that the associating wiper connects to.

Now each pot will have one unsoldered pin left. Connect these to the appropriate socket. The wire should go to the signal lug of the socket. IN 1 goes to the signal lug on the socket labelled IN 1, and so on.

There are a quite lot of wires here, but it should be quite neat once it is all done.

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

Apply power to the unit making sure you are applying the power correctly. Check that no device is running hot. Any sign of smoke or strange smells turn off the power immediately and recheck the polarity of the power supply, and the direction of the ICs in their sockets and the polarity of the electrolytic capacitors.

Assuming everything is OK so far, it is time to apply an audio input. Use a bright signal like a sawtooth output from a VCO. The A below middle C, 220Hz, is a good note to use.

Connect your amplifier or mixing desk input to the output socket. Set the Drive control to its minimum value and switch to LP mode. Moving the Frequency control should produce the usual and distinctive filter effect from the output.

Turn up the Drive the control and notice that the sound should become louder. Click the switch into the HP position and the output should now become fizzy. Sweep the frequency again with the Frequency pot. At the top end, the sound should be very thin, or be gone altogether. At the low end, you should hear the sawtooth as normal.

Turning the Resonance up will accentuate the ‘electronic’ nature of the sound in both the LP and HP modes. Check that at maximum resonance the filter output will oscillate. It'll probably start to squeal around 70% of the way around at some settings of the frequency pot. Beware, it is quite possible to get this filter to oscillate above the range of hearing. So be careful so as not to damage your studio monitor’s tweeters.

It has been noted by one person who had built the Journeyman that at least one example of the THAT300 in position U4 produced a less than satisfactory resonance. This is probably due to a lower value of Hfe of the transistors in that particular specimen. If this happens, the easiest thing to do is simply swap your two THAT300P devices around. If this doesn't work, then both your devices are similarly rated. In this case, you can increase R49 from 62K to 68K.

Listening to the low pass output with the sawtooth input still connected, patch an LFO output to the CV 2 input. Set the CV 2 depth pot to its maximum position. You should now hear the frequency of the filter being swept with the LFO. Reduce the CV 2 depth with the pot and hear how the modulation depth decreases to nothing with the pot in the middle and then increases as the pot moves around to the inverting position.

If all this happens, the chances are that you have a working module.

Calibration

The module has three trimmers, two multiturn and one single turn.

TUNE: This adjusts the filter’s cut-off frequency. Set this so that the filter’s FREQ pot covers your chosen range. I would normally place this in the middle position for now, that is 10 turns or so, from one of the end points.

SCALE: This adjusts the scaling of the exponential inputs. Adjust this so that there is an octave jump in cut-off frequency when the CV 1, or KEY CV on the 2U version, input is raised by one volt. The best way to set this is to turn the resonance full up and let the filter oscillate around 440Hz, which is the A above middle C. Then connect the filter module to the 'keyboard CV' out of your midi-CV convertor or analogue keyboard. Your keyboard should produce a 1V change in output CV for every octave on the keyboard.

Now this is a fiddly adjustment and it takes a while to get it right. Changing the V/OCT will also change the range of the filter as well, so you may need to alter the TUNE trimmer afterwards. You don't need to worry about getting this that accurate since the VCF core itself isn't perfectly exponential and tends to drift a bit with temperature.

BAL: Turn the frequency and resonance pot to their middle position. The drive and CV2 pots can be left anywhere. Now connect a 440Hz triangle wave signal to the CV 1 input socket, or KEY CV input socket on the 2U version. Listen to the audio output very carefully. You should hear the 440Hz breaking through slightly. Adjust BAL to minimise the breakthrough. But, bear in mind that each time you move the BAL trimmer the 440Hz will break through for a short while and then die off a bit. Adjust the BAL trimmer so that once the output does stabilise it falls to its minimum level. You won't get it to be completely silent though.

Final Comments

If you have any problems with the module, an excellent source of support is the Oakley Sound Forum at Muffwiggler.com. Paul Darlow and I are on this group, as well as many other users and builders of Oakley modules.

If you 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 e-mail 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 postage costs, any parts used and my time at 25GBP per hour. Most faults can be found and fixed within one hour, and I normally return modules within a week. The minimum charge is 25GBP plus return postage costs.

If you have a comment about this builder's guide, or have a found a mistake in it, then please do let me know. But please do not contact me or Paul Darlow directly with questions about sourcing components or general fault finding. Honestly, we would love to help but we do not have the time to help everyone individually by e-mail.

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 and Analogue Heaven mailing lists and those at Muffwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK
Last update July 2010