Oakley Sound Systems

Eurorack Modular Series

Journeyman Diode Ring Filter

Builder's Guide

V1.2

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

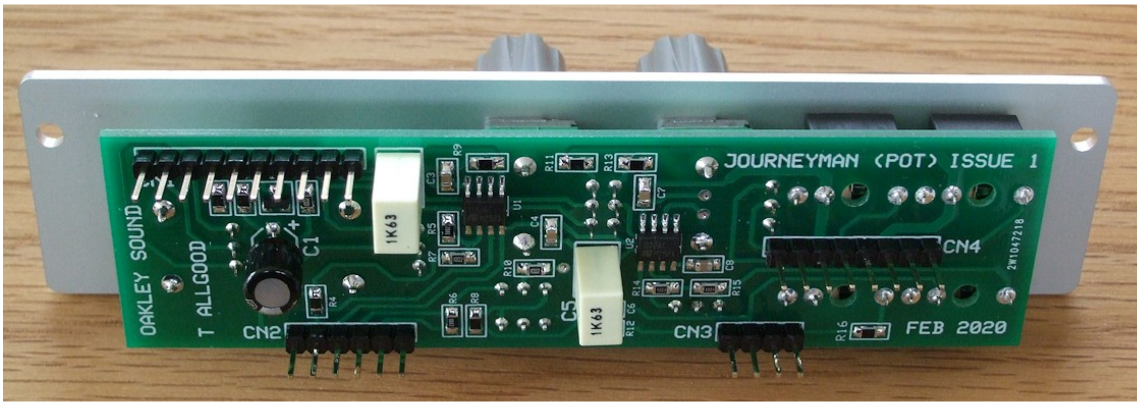
This is the Project Builder's Guide for the Journeyman filter Eurorack module from Oakley Sound. This document contains a basic introduction to the issue 1 board set, a full parts list for the components needed to populate the boards, some basic help on how to purchase parts and to build the project, a simple test procedure, and calibration.

For general information regarding where to get parts and suggested part numbers please see our useful Parts Guide at the project webpage or http://www.oakleysound.com/parts.pdf.

For general information on how to build our modules, including circuit board population, and mounting front panel components please see my generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

The Journeyman PCB Set

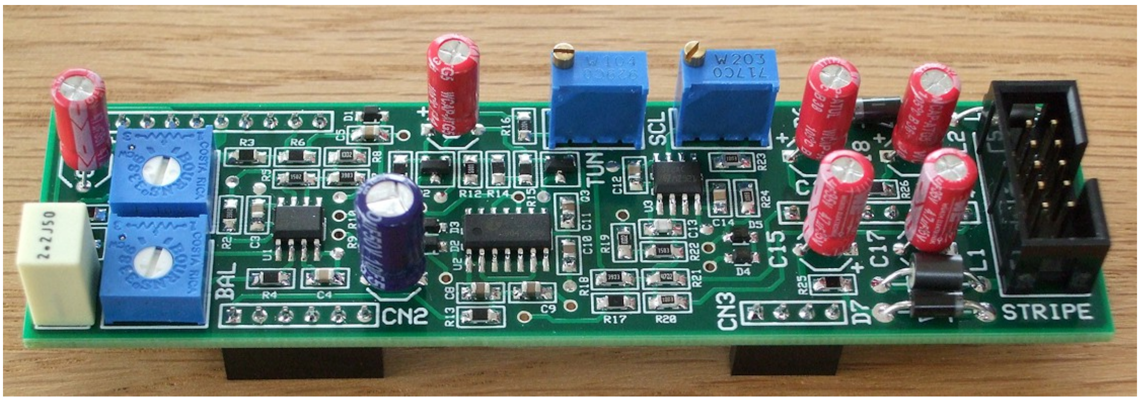
The electronics of the Oakley Journeyman are built on two printed circuit boards (PCBs). The Pot board holds the four pots, two switches, four input and output sockets, and the input and output audio circuitry. All the front panel components are directly soldered to the pot board. The pot board is a double sided design meaning that there are electrically conductive copper tracks on the top and bottom surfaces of the board. The surface mount components are all soldered to the top side of the board while the pots, switches, and sockets are soldered to the underside which faces the inside surface of the front panel.



The pot board fitted to the front panel. Note that all the surface mount components and big capacitors are fitted to this side. The pots, switches, and sockets are fitted to the underside of the board.

The second board is called the main board. This holds the discrete filter core, the control voltage circuitry, the power supply conditioning, and the power inlet. The main board is a four layer design. This means that the board has layers of copper on top and bottom sides, as well as two internal copper layers. The top internal layer is solely reserved for the 0V connections – sometimes called module ground – used for the main and top board's electronics.

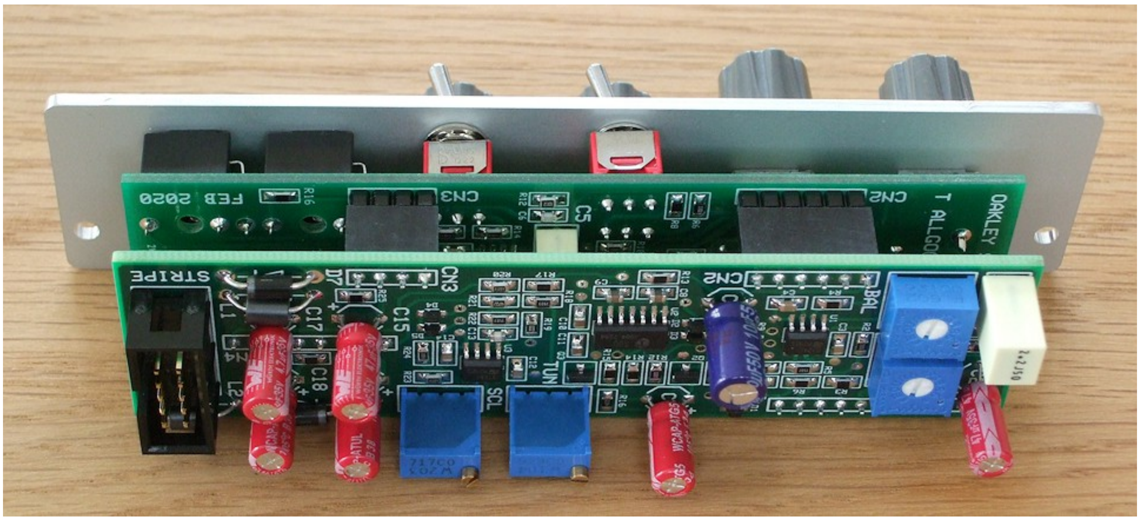
The components, again a mixture of through hole and surface mount devices, are mostly soldered to the top of the board. The main board's components are all accessible from the rear of the module. Although this means that you need to be a little careful when handling the module, it does mean that fault finding is much easier than it would be if the main board was facing the other way and none of the components would be visible.



The four layer main board.

The surface mount components are mostly of relatively large geometries to make the build suitable even for beginners in surface mount soldering. The resistors and capacitors are all 0805, and the ICs including the That Corp. NPN array are narrow body SOIC. The larger capacitors, ferrite beads, power diodes, and all the interconnects are through hole parts.

Both circuit boards are the same size, that is, 29 mm (wide) x 107 mm (high). The boards are designed to go behind a panel that is 6HP wide.



The module comprises of two circuit boards connected together with four different size SIL headers and sockets.

The grip of the four SIL interconnects is strong enough to hold the boards together for most uses. Should you require more rigidity then a **small** blob of silicone (neutrally curing RTV) sealant across the join between the plastic housings of CN1, CN2 and CN3 would hold everything together tightly. However, it is best not to do this until your unit has been thoroughly tested.

The design requires plus and minus 12V supplies. The power supply should be adequately regulated. The current consumption is slightly below +25mA and -20mA at +/-12V.

Journeyman Parts Lists

Many of the parts for this circuit board are surface mount devices but not all of them. Take special care when ordering your parts that you order the correct type of part. This parts list shows the type of part needed whereas the circuit diagram does not.

A quick note on European part descriptions. R is shorthand for ohm. K is shorthand for kilo-ohm. R is shorthand for ohm. So 22R is 22 ohm, 1K5 is 1,500 ohms or 1.5 kilohms. For capacitors: 1uF = one microfarad = 1000nF = one thousand nanofarad. For electrolytic capacitors the maximum working voltage is normally given with the value, eg. 1uF, 63V is a one microfarad capacitor with a working DC voltage of 63V.

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.

Main Board issue 1 Parts List

Resistors

All resistors are surface mount, size 0805 (or metric 2012) 1% 125mW metal film.

33R	R3, R13, R25, R26
100R	R9, R10
1K8	R14
6K8	R12
10K	R8, R19
15K	R5
18K	R6
33K	R1
36K	R4
39K	R2
47K	R21
100K	R20, R23, R24
150K	R22
180K	R11, R15, R17
390K	R7, R18
2M2	R16

Capacitors

The following capacitors are surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric X7R, working voltage 50V, tolerance +/-5%.

100nF	C3, C4, C5, C12, C14

The following capacitors are surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric C0G (or NP0), working voltage 25V or 50V, tolerance +/-5%.

47pF	C13
22nF	C8, C9, C10, C11

The following capacitor is a standard through hole component with 0.2” (5mm) spaced radial leads.

2u2, 50V polyester	C1

The following capacitors are standard through hole electrolytic capacitors with 0.1” (2.5mm) or 0.2” (5mm) spaced radial leads.

4u7, 63V electrolytic	C17, C18
10uF, 35V electrolytic	C7
47uF, 35V electrolytic	C15, C16
220uF, 16V electrolytic	C2

The following capacitor is a non-polar (or bipolar) through hole electrolytic capacitor with 0.1” (2.5mm) or 0.2” (5mm) radial leads.

10uF, 16V NP electrolytic	C6

Discrete Semiconductors

The following devices are surface mount parts.

1N4148WS silicon diode	D1, D2, D3, D4
BZX384-C6V2 zener diode	D5
BC850 NPN transistor	Q1, Q2, Q3

The following devices are standard through hole parts.

1N5819 Schottky diode	D6, D7

Integrated Circuits

The following parts are all surface mount devices. All are small outline (SOIC) narrow body packages.

THAT300S14-U NPN array	U2
TL072ACD dual op-amp	U1, U3

Trimmer (Preset) Resistors

5K 3/8" single turn (eg. Bourns 3386F)	BAL
10K 3/8" single turn (eg. Bourns 3386F)	RES
100K multiturn (eg. Bourns 3296W)	TUN
20K multiturn (eg. Bourns 3296W)	SCL

Miscellaneous

Axial ferrite bead	L1, L2
2 x 5 way boxed IDC 0.1" header	STRIPE
4-way SIL socket	CN3
6-way SIL socket	CN2
8-way SIL socket	CN4
10-way SIL socket	CN1

CN1 to CN 4 are mounted on the underside of the board. Special care must to be taken to ensure that connectors CN1 to CN4 are mounted perpendicular to the board surface.

Pot Board issue 1 Parts List

Resistors

All resistors are surface mount, size 0805 (or metric 2012) 1% 125mW metal film.

150R	R8
330R	R6
1K	R16
4K7	R1, R2, R11, R13
10K	R3, R7, R10, R12
18K	R5
100K	R14, R15
180K	R4, R9

Capacitors

The following capacitor is a surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric C0G (or NP0), working voltage 50V, tolerance +/-5%.

100pF	C6

The following capacitors are surface mount, size 0805 (or metric 2012) multilayer ceramic, dielectric X7R, working voltage 50V, tolerance +/-5%.

100nF	C3, C4, C7, C8

The following capacitors are standard through hole components with 0.2" (5mm) spaced radial leads.

1uF, 50V or 63V polyester	C2, C5

The following capacitor is a low profile (no more than 7.5mm in height) through hole electrolytic capacitor with 0.1” (2.5mm) or 0.2” (5mm) radial leads.

10uF, 35V electrolytic low profile	C1

Discrete Semiconductors

The following device is a surface mount part.

BC850 NPN transistor	Q1

Integrated Circuits

The following parts are all surface mount devices. All are small outline (SOIC) narrow body packages.

TL072ACD dual op-amp	U1, U2

Potentiometers

All pots are Alpha 9mm vertical pots with 6.35mm round shafts.

10K linear	RESONANCE
50K linear	FREQ, CV2_DEPTH
50K dual gang linear	DRIVE

The pots are to be fitted to the underside of the board and their pins soldered from the topside. Note that there is no component identification on the reverse side of the board so use the front panel as guide to make sure the parts go into the correct locations.

Four knobs to suit. Davies 1900H or clones thereof are to be recommended.

Miscellaneous

SPDT switch (eg. Thonk DW1)	INVERT
DPDT switch (eg. Thonk DW3)	LPF/HPF
Thonkiconn 3.5mm socket	IN, OUT, CV1, CV2

The four 3.5mm sockets and two switches are to be fitted to the underside of the board and their pins soldered from the topside. See later for more details.

4-way SIL 0.1” header	CN3
6-way SIL 0.1” header	CN2
8-way SIL 0.1” header	CN4
10-way SIL 0.1” header	CN1

Part Sourcing

For general information regarding where to get parts and suggested part numbers for Oakley projects please see my Parts Guide at the project webpage or direct from:

www.oakleysound.com/parts.pdf.

The front panel is obtained either from Schaeffer in Germany, or Front Panel Express (FPE) in the US. The database for the panel is provided on the project webpage and this file can be opened, edited and ordered using the Frontplatten Designer program available free from Schaeffer or FPE. The cost of the 6HP wide panel was around 27 Euros at the time of writing. The panel is 2.5mm thick and has a natural silvery finish. The black text is printed, although it is possible to edit the database to have the text engraved.

The four pots are Alpha 9mm vertical pots as sold by Thonk and others. I use Thonk's standard ones with 6.35mm (0.25”) round shafts. Other shaft types are available, like splined or D-type, and you should purchase those that are compatible with your choice of control knob. I prefer the round shafts as they require control knobs that have a fixing screw. Although these knobs take more time to place and secure, the height at which the knob sits on the pot shaft is under your control. Push fit knobs can sometimes not sit at the right height which is unsightly if they are too high, or graunch against the pot's securing nut and washer if they are too low.

For control knobs I use Davies 1900H clones available from Thonk and others. I used dark grey ones for my prototype module.

The toggle switches are a standard sub-miniature type that are designed to fit directly into a PCB. The on-on SPDT switch is available from Thonk as their DW1 sub-miniature switch, and the on-on DPDT switch as their DW3. They are made by Dailywell and offer good performance for their price.

The four 3.5mm sockets are available from Thonk known either by their Thonkiconn moniker or as part number PJ398SM.

The fixed 2.54mm (0.1”) interconnects are in two parts, the male header, and the female socket or receptacle. I use one 10-way, one 8-way, one 6-way and one 4-way single in line (SIL) connectors in this project. The ones I use are made by Multicomp, although more expensive types are available which may offer increased longevity. These are the Multicomp part numbers for the parts I used:

4 way socket	2212S-04SG-85
4 way header	2211S-04G
6 way socket	2212S-06SG-85
6 way header	2211S-06G
8 way socket	2212S-08SG-85
8 way header	2211S-08G
10 way socket	2212S-10SG-85
10 way header	2211S-10G

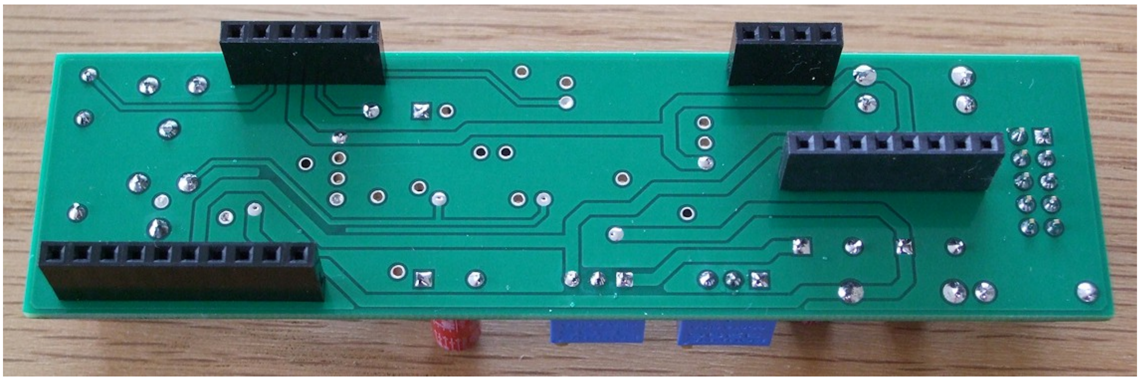
Populating the Circuit Boards

For general information on how to build Oakley modules please see my generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

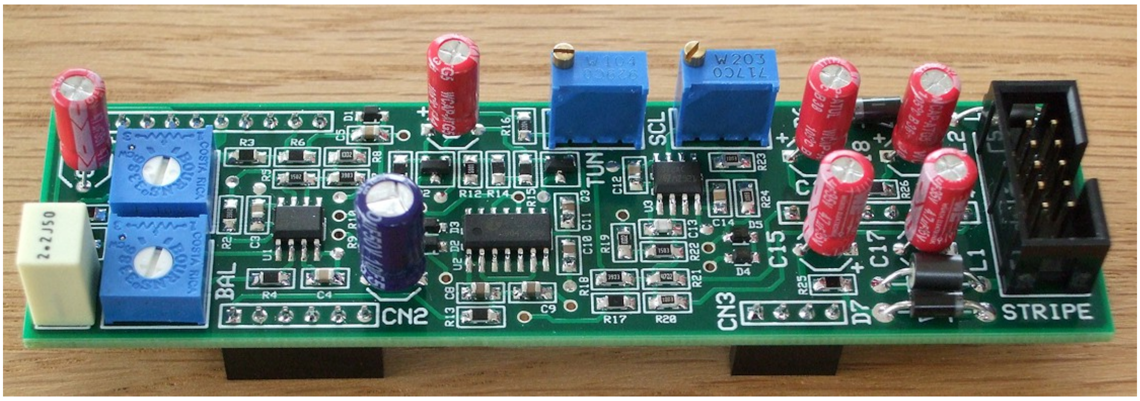
Main Board Construction

All the surface mount components should be soldered first. Take care to treat all transistors, diodes and ICs as static sensitive devices. I usually solder the resistors first, then the capacitors, then the discrete semiconductors, and then the ICs.

The next items to be soldered are the single in line (SIL) sockets. These are to be fitted to the underside of the board and soldered from the top side.



Special care should be taken to ensure that each SIL socket sits at right angles to the board surface. One way to do this is to temporarily fit the SIL headers into the SIL sockets and fit the Main board and Pot board together with the headers and sockets in between. Then solder the headers to the main board. Pulling the two boards apart should reveal all the sockets neatly in place and at the correct angle.

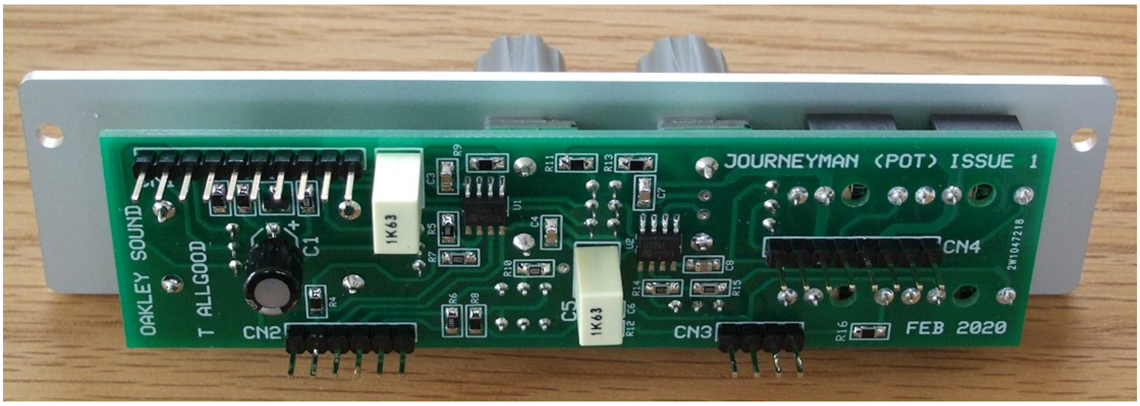


The next items to be soldered will be the through hole components. Remember that diodes and the standard electrolytic capacitors are polarised so they need to be fitted the right way around. C6 being a non polar capacitor can go any way around. You should especially make sure that the boxed header is correctly orientated. Pin 1 is normally designated with a little arrow shape on the plastic housing of the header and this should align with the square pad on the board. Also, the hole in the housing should correspond to the little box shape on the board's printed legend.

Pot Board Construction

Except for the pots, switch, and sockets all the parts are to be fitted to the topside of the board. The surface mount parts should again be soldered first.

Now fit the SIL headers into place and solder from the underside of the board. It may be worth temporarily fitting the Pot board to the Main Board so as to hold the headers exactly at right angles while you solder.



The remaining parts are the front panel components and these will be fitted to the underside of the board and soldered from the top of the board. There are no legends on the underside of the board to indicate where the parts will go but using the front panel it should be clear where they need to be fitted. To ensure the correct alignment of these parts before soldering you should have your front panel ready. The panel will be used as a jig to hold the parts in the correct place while you solder. Failure to use the panel as a jig prior to soldering could mean that the panel will not slide onto the components.

The first parts to place, but not solder, will be the pots. The pots should be fitted so that the three pot pins go into the board first, then ease in the two lugs into their holes and push in firmly so that the pot clicks into place. The pots will sit securely in the board even without soldering. The dual gang pot may need its lugs slightly straightened with a pair of pliers before fitting it to the board.

Now fit the four sockets and switches into their places on the board but again do not solder yet. Place one of the toothed washers that came with the switches onto each switch's mounting bush.

Now ease the panel down onto the pots, switch and sockets making sure that all their threads are sitting snugly in their holes. Place a washer and a nut on each of the pots and sockets, but not the switch. Tighten the nuts but not too tightly. Turn the module over to reveal the topside of the circuit board. Now solder all the pots and sockets, but not the switches yet.

Flip the module over again and fit a single nut, and if you have one, a flat dress washer, to the exposed thread of each switch. You can ignore the other nuts that came with the switches, as well as any washers that have locating lugs. Gently tighten the nuts taking care not to scratch the panel. The switches will be pulled towards the panel and slightly off the board's top surface when you tighten the nuts. The module can be turned over and the leads of both switches soldered.

Initial Testing

It is prudent to test the main board on its own before fitting it to the pot board and its panel. If you have a bench power supply that allows you to select the current limit then set both the +12V and -12V supplies to a current limit of 50mA. This should prevent any serious meltdowns if there is a problem with the build.

Power up the main board on its own. If you can measure power supply current then it should be around -8mA from the negative rail and +10mA from the positive rail. Anything significantly more than this, like 20mA, will indicate a problem. If you can't measure current then check that no devices are getting warm. If all is well, then the main board can be powered down. Wait a minute and then attach the main board to the pot board. Be very careful to ensure all the SIL connections are correctly in place.

Power up the module and, if you can, check the current draw of the module. It should be around +20mA and -20mA. Anything significantly different to this, say over 30mA, will indicate a problem.

Set the mode switch to LPF and invert switch to its up position. Turn the Resonance, Drive and CV2 pots to their minimum settings. Turn the Frequency pot to its maximum. Connect a 10V peak to peak (ie. +/-5V peak) 220Hz (the A below middle C) sawtooth wave to the Input socket. Listen to and, or, monitor on an oscilloscope the signal from the output socket. You should hear the sawtooth waveform and it will be somewhat quieter than the input signal. The output signal is typically around a fifth of the input signal voltage. This is done so that when the resonance is turned up the final output does not exceed the maximum capability of the module at +/-10V peak.

Turn the frequency pot down and you should hear the characteristic behaviour of a low pass filter whereby the sound gets progressively duller.

Turn up the resonance a little. If it starts to self-oscillate, back it off a little, and remember that a self-oscillating diode ring filter can produce a very large output signal. Rotate the frequency control again and assure yourself that the filter is producing the expected behaviour of a high resonance low pass filter. If your module does not self-oscillate at all, then it is likely that the RES trimmer needs adjusting. However, the resonance control should still affect the sound of the filter even if the module cannot yet self-oscillate.

Change the filter mode from LPF to HPF, the high pass filter function. The output signal should now get fizzy and the bottom end of the signal drop out as the frequency pot is turned up.

Go back into LPF mode and turn up the drive control. The output signal should get slightly quieter as the increasing input signal is compressed by the overdriven filter circuitry. If you have the resonance turned up then you'll notice that the resonance appears to reduce as drive is increased. This is because, in a diode ring filter, a large input signal acts to dampen any resonant behaviour.

Patch an LFO into the CV2 input. Check that the CV2 Depth pot controls the amount of modulation of the Journeyman's cut-off frequency. Change the nearby switch to invert and check that the modulation still occurs. If you use a sawtooth LFO signal into CV2, the invert switch should turn the incoming LFO signal into a ramp waveform, so instead of hearing 'dow, dow, dow...' you now hear 'yit, yit, yit...'.

The module is now ready for calibration.

Calibration

The module has four trimmers, two small single turn units and two multiturn ones. You should trim the module in the order given below.

RES

This adjusts the maximum level of resonance obtainable with the resonance front panel control. There is no correct position for this trimmer since it should be set to personal taste. I prefer to have the filter just starting to self-oscillate with the resonance pot set to around 2 o'clock. In this case, simply set the resonance pot to this position and adjust RES until oscillation just starts to break out. You will find that the actual point self-oscillation occurs depends a little bit on the actual frequency of oscillation. Lower cut-off frequencies often need a higher resonance setting to self-oscillate. So have a play with the frequency and resonance pots and the RES trimmer to give you the response you like. You can always change it at a later date if you want your resonance control to be more or less sensitive.

BAL

This is adjusted so that the amount of CV breakthrough is set to a minimum. Turn the Drive and Resonance pots to their minimum positions. Turn the Frequency pot to its maximum setting. Turn the CV2 pot to around 9 o'clock. The Invert switch can be either on or off. Now connect a 220Hz square wave signal to the CV2 input.

Listen to the audio output very carefully – you may need to turn up the volume a bit on your monitoring system. You should hear the 220Hz signal breaking through slightly, this is CV breakthrough and it is unwanted. Adjust BAL to minimise the breakthrough. You won't get it to be completely silent but you should be able to find a point that produces less of a breakthrough than other positions. Note that as you turn the BAL trimmer the output will wheeze and crackle a little. This is to be expected. Remember your aim is to make sure you get the quietest output after you've stopped adjusting the trimmer.

SCL

This adjusts the exponential scaling of the two CV inputs. Adjust this so that there is an octave jump in cut-off frequency when the CV1 input is raised by one volt.

Connect the CV1 input to the 1V/octave output (sometimes called KCV) of your sequencer, midi-CV convertor, or analogue keyboard. This output should produce a +1.000V change in output CV for every octave you go up.

Set the Journeyman to self-oscillate and monitor the output signal.

Play an A on your keyboard or sequencer and adjust the Frequency control on the front panel of the Journeyman so that the filter gives out a sine wave like signal of around 220Hz. Now play the A note two octaves higher than you were pressing. By adjusting SCL you should aim to get the higher A to make the Journeyman oscillate at 880Hz, ie. four times that of 220Hz. However, any change in scaling will also change the lower note as well as the higher note. So you will have to move back and forth between altering SCL and the Frequency control until you get the two octave spread you require.

Remember that the Journeyman will not be able to be made to track perfectly over a very wide range. Nor is the scaling adjustment temperature compensated. As such, even if you have achieved a perfect two octave spread with your two A notes, you won't be able, for example, to hear a perfect octave spread from the two As an octave above those. However, since the purpose of the Journeyman is, for the most part, to filter and not to act as a perfectly tuned oscillator, this lack of accurate scaling should not be a problem.

You should note that scaling is affected by the position of the BAL trimmer so remember to set BAL first before calibrating the scaling.

TUN

This adjusts the filter's cut-off frequency and works in much the same way as the Frequency pot on the front panel.

The maximum useful cut-off frequency of the Journeyman's diode ring core, like the original circuit from which it was inspired, is surprisingly low at around 17kHz. Beyond this frequency the filter core starts to behave like an inverted voltage controlled amplifier (VCA) which reduces the amplitude of the whole input signal with increasing control voltages. This causes an obvious squashing of the audio signal when the filter core is modulated over a wide range and is not, in my opinion, musically useful. Because of this the Oakley Journeyman's internal CV processing circuitry prevents the filter core from being modulated much above 17kHz.

So that the frequency control on the Journeyman's front panel covers the full range available to it, it is necessary to adjust TUN to set the cut-off frequency to be 17kHz when the Frequency pot is set to its maximum setting.

Turn the frequency and resonance controls to their maximum. Set drive and CV2 to their minimum. Monitor the output, either with a frequency counter, 'scope, or spectrum analyser plug-in, and adjust TUN so that the Journeyman is self-oscillating somewhere between 17kHz and 17.5kHz

If you have no way of measuring frequencies, then simply adjust TUN so that the middle pin of the TUN trimmer is around -5V with respect to 0V.

Final Comments

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

If you can't get your project to work and you are in the UK, 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 directly with questions about sourcing components or general fault finding. Honestly, I would love to help but I 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 the great people on the Synth-diy and Analogue Heaven mailing lists and those at Muffwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK