Oakley Sound Systems

Eurorack Modular Series

Croglin Dual Filter

Builder's Guide

V1.1

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

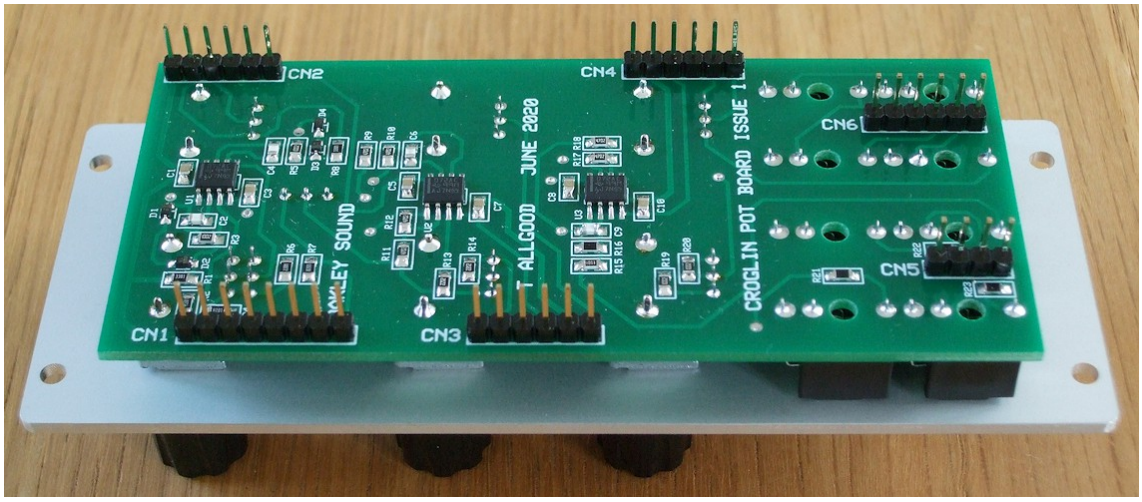
This is the Project Builder's Guide for the Croglin Eurorack module from Oakley Sound. This document contains a basic introduction to the issue 1 board set, a full parts list for the components needed to populate the boards, some basic help on how to purchase parts and to build the project, a simple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

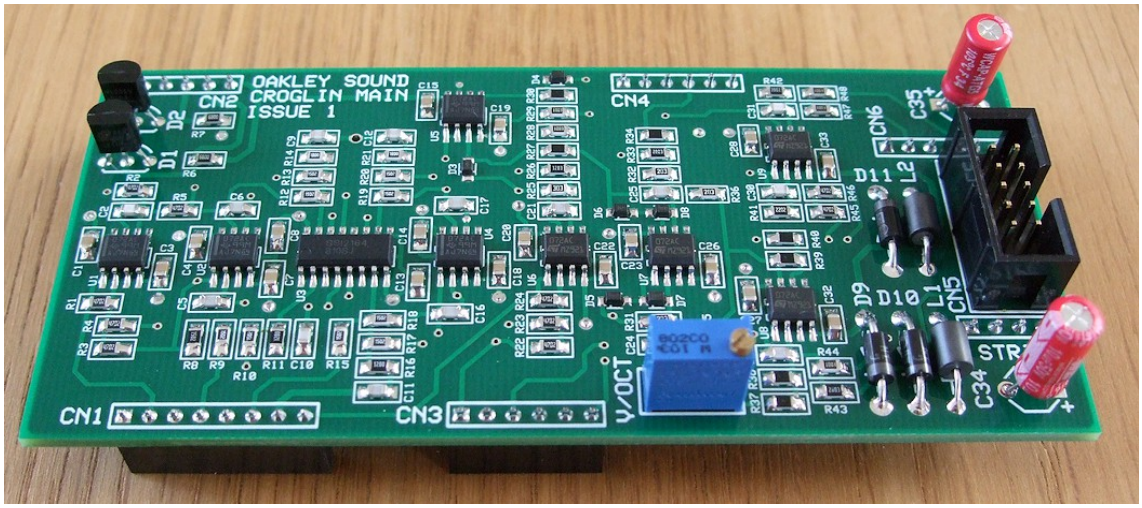
The Croglin PCB Set

The electronics of the Oakley Croglin filter module is built on two printed circuit boards (PCBs). The Pot board holds the six pots, one switch, input and output sockets, the output circuitry for filter B, the resonance circuitry, and the offset CV processing. 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 and sockets are soldered to the underside which faces the inside surface of the front panel.



The top side of the Croglin pot board. The board is already fitted to the front panel. Note that all the surface mount components are fitted to this side. The pots, switch and sockets cannot be seen in full as they are fitted to the underside of the board.

The second board is called the main board. This holds the filter core based around the SSI2164, the output circuitry for filter A, the CV processing circuitry, the power supply conditioning and references, 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 pot board's electronics.



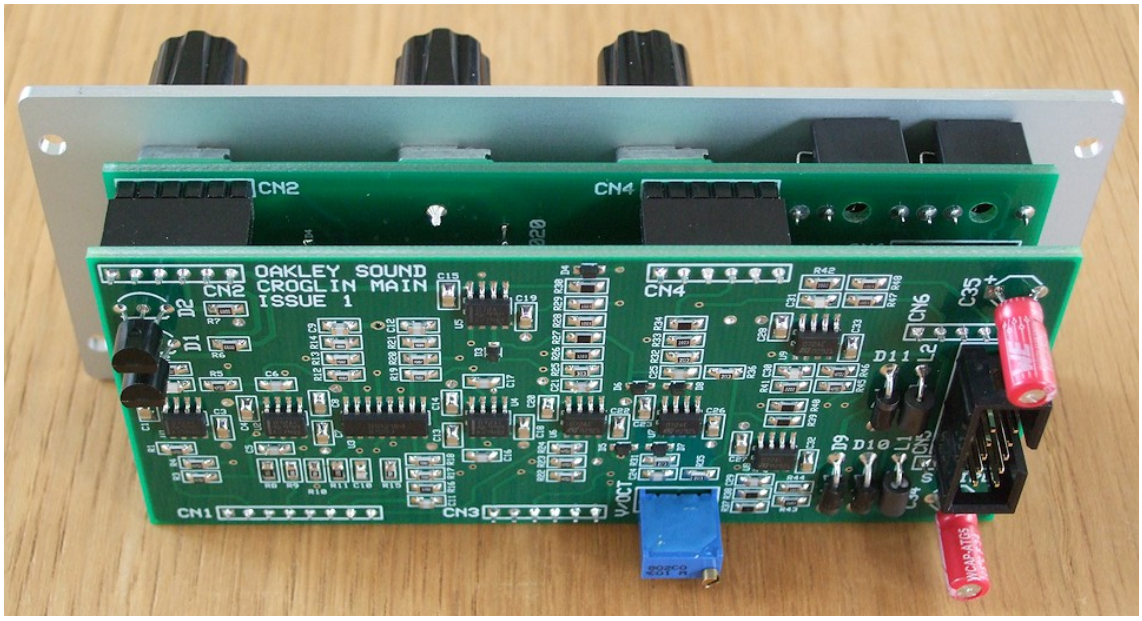
The Croglin Main Board.

The components, again a mixture of through hole and surface mount devices, are 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 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 are narrow body SOIC. The electrolytic capacitors, ferrite beads, power diodes, and all the interconnects are through hole parts.

Both circuit boards are the same size, that is, 49 mm (wide) x 107 mm (high). The boards are designed to go behind a panel that is 10HP wide.



The module comprises of two circuit boards connected together with six 0.1" SIL headers and sockets.

The grip of the multiple SIL interconnects is strong enough to hold the boards together for most uses. Should you require more rigidity then a **small** blob of silicone sealant or heat melt glue across the join between the plastic housings of CN1, CN2, CN3 and CN4 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 around +60mA and -60mA at +/-12V.

Croglin 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.

Pot Board issue 1 Parts List

Resistors

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

1K	R21, R22, R23
2K2	R2, R6
3K3	R1, R8
8K2	R4, R7
10K	R3, R5, R19, R20
22K	R13, R14
47K	R9, R10, R11, R12, R17, R18
82K	R16
100K	R15

Capacitors

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

100nF	C1, C3, C5, C7, C8, C10

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

10pF	C2, C4
47pF	C6, C9

Discrete Semiconductors

The following devices are SOD-323 surface mount parts.

BZX384-C3V9 zener diode	D1, D2, D3, D4

ICs

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

TL072ACD dual op-amp	U1, U2, U3

Potentiometers

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

10K linear (dual gang)	Resonance
50K linear	Frequency, Offset CV, Offset, Frequency CV, Duality

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.

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

Miscellaneous

4-way 0.1” header	CN5
6-way 0.1” header	CN2, CN3, CN4, CN6
8-way 0.1” header	CN1

Special care must to be taken to ensure that connectors CN1 to CN4 are mounted perpendicular to the board surface.

Thonkiconn 3.5mm socket	Input 1, Input 2, Output A, Output B, 1V/Oct, Frequency, Offset, Output
SPDT switch (eg. Thonk DW1)	LP/BP

The eight 3.5mm sockets and switch is to be fitted to the underside of the board and their pins soldered from the topside. See later for more details.

Main Board issue 1 Parts List

Resistors

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

120R	R11, R16
180R	R14, R21
680R	R6, R7
4K7	R27, R30
15K	R8, R9, R10, R12, R13, R15, R17, R18, R19, R20
22K	R41
30K	R25
36K	R38, R39, R40
39K	R42
47K	R1, R2, R3, R4, R5, R22, R23, R24, R45, R46
82K	R34
100K	R28, R29, R44, R47, R48
120K	R26
150K	R37
200K	R33, R43

The following resistors are surface mount, size 0805 (or metric 2012) 0.1% 125mW thin film.

20K 0.1%	R31, R32, R35, R36

Capacitors

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

100nF	C1, C3, C4, C7, C8, C13, C14, C15, C18, C19, C20, C22, C23, C26, C27, C28, C32, C33

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	C2, C30, C31
100pF	C21, C24, C25, C29
220pF	C5, C6, C16, C17
1n5	C9, C12
2n2	C10, C11

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

4u7, 50V	C34, C35

Discrete Semiconductors

The following devices are surface mount parts.

1N4148WS signal diode	D3, D4, D5, D6, D7, D8

The following devices are standard through hole parts.

1N5819 Schottky diode	D9, D10, D11

Integrated Circuits

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

SSI2164 quad VCA	U3
TL072ACD dual op-amp	U1, U2, U4, U5, U6, U7, U8, U9

The following devices are standard through hole parts.

LM4040-10V	D1, D2

Trimmer (Preset) Resistor

10K multiturn top adjust	V/OCT	(eg. Bourns 3296W-1-103LF)

Miscellaneous

Axial ferrite bead	L1, L2
2 x 5 0.1” boxed header	STRIPE
4-way 0.1” socket	CN5
6-way 0.1” socket	CN2, CN3, CN4, CN6
8-way 0.1” 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.

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 10HP wide panel was around 30 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 six 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.

Note that the Resonance pot is a dual gang or stereo pot. All the others are standard single gang pots.

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

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

The six 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 8-way and three 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

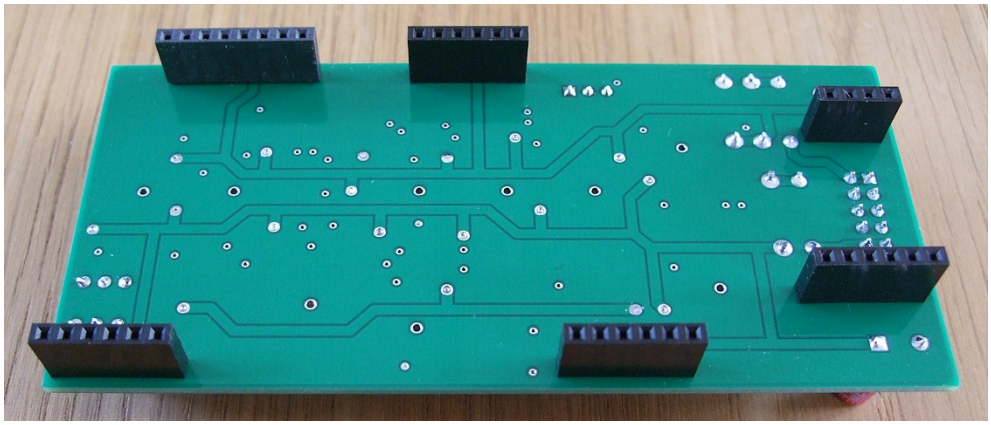
Populating the Circuit Boards

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

Main Board Construction

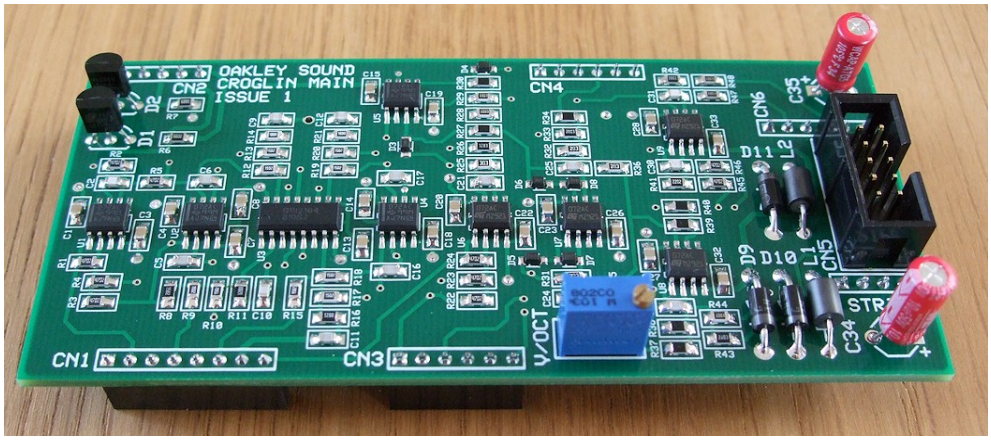
All the surface mount components should be soldered first. Take care to treat all transistors, diodes and ICs as static sensitive devices. I usually solder the resistors first, then the capacitors, then the discrete semiconductors, and then the ICs.

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.

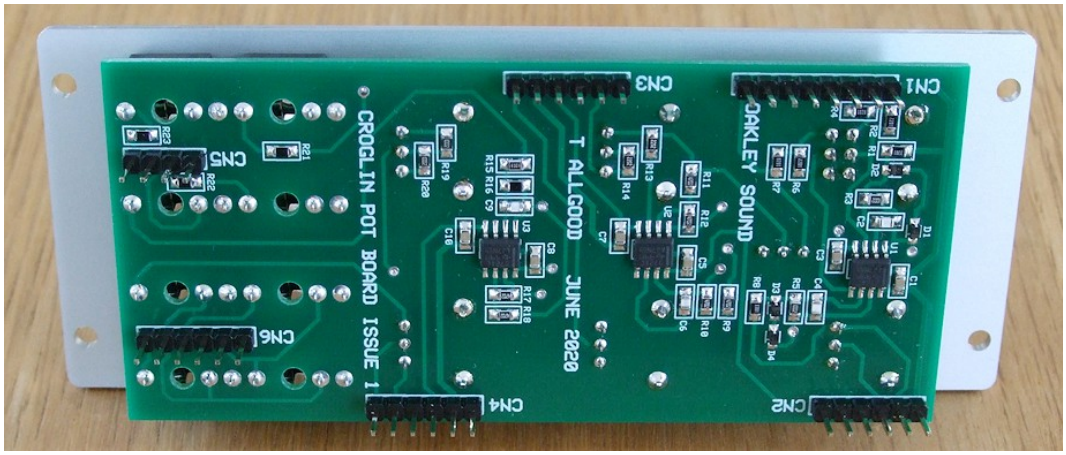
The next items to be soldered will be the through hole components. Remember that diodes and electrolytic capacitors are polarised so they need to be fitted the right way around. You also should especially make sure that the boxed header is correctly orientated. Pin 1 is normally designated with a little 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 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 for these components 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 pot will sit securely in the board even without soldering.

Now fit the eight sockets and switch into their places on the board but again do not solder yet. Place a toothed washer that came with the switch onto the threaded bush of the switch.

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 six pots and the eight 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 switch 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 the switch. You can ignore the other nut that came with the switch, as well as any washers that have locating lugs. Gently tighten the nut taking care not to scratch the panel. The switch will be pulled towards the panel and slightly off the board's top surface when you tighten the nut. The module can be turned over and the switch's leads soldered.

Initial Testing

It is prudent to test the main board on its own before fitting it to the pot board and 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 75mA. 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 -45mA from the negative rail and +45mA from the positive rail. Anything significantly more than this, like 55mA, 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 whole module and, if you can, check the current draw. It should be around +60mA and -60mA. Anything significantly different to this, say over 75mA, will indicate a problem.

Set the switch to the LP mode and listen to the signal coming from the Output A socket. Turn the Resonance up full and make sure you can hear Filter A self-oscillating. The Frequency pot should be able to sweep the oscillation from very low sub-audible frequencies, across the whole audio band, and beyond. If you have an oscilloscope you should see that the signal at Output A is a sine wave of around 4.5V to 5.0V peak to peak.

Now listen or monitor the signal coming from the Output B socket. Again you should hear a sine wave signal whose frequency can be controlled by the Frequency pot. You should also find that the frequency can be changed by the Offset pot.

Now listen or monitor the signal coming from the Output socket on the bottom row. This is the combined signal from both Filter A and Filter B. With the Duality control in the middle, the signal from Filter A should be the dominant signal and the primary tone heard should not be affected by the Offset pot. Moving the Duality pot either side of the centre will increasingly allow Filter B's self-oscillation to be heard. At either extremes of the Duality pot's movement Filter A and Filter B should be roughly the same volume. Check that when the Offset pot is in its middle position the frequency of self-oscillation for both filters is more or less the same. That is, they should be in unison when the Offset pot is central.

Move the resonance control to its minimum value. Patch a sawtooth, of roughly 220Hz, from an oscillator module into the Input 1 socket and listen to the signal from the Output socket. Put the Duality control in the middle and assure yourself that the module is filtering correctly. Opening the filter should allow you to hear the sawtooth clearly, while reducing the cut-off frequency should give you a duller tone. With the Frequency control fully counter clockwise the output of the filter should be very quiet indeed.

Increase the Frequency to the maximum again and insert the sawtooth into Input 2. You should hear the same signal again from the Croglin's Output socket but this time somewhat quieter. Input 2 is 6dB quieter compared to Input 1 when Input 1 is used on its own. A smaller input signal will allow the resonance of the Croglin's filters to have a more pronounced effect.

Calibration

There is one multi turn trimmer on the Croglin.

V/OCT

This adjusts the scaling of the Frequency and 1V/Oct CV inputs. Adjust this so that there is an octave jump in cut-off frequency when the 1V/octave input is raised by one volt.

Plug a 1V/octave source into the 1V/Oct socket. This may be your keyboard or sequencer's pitch CV output, or the CV output of a midi-CV convertor. Set the resonance pot fully clockwise to get the filter cores oscillating. Monitor the audio signal coming from Output A. Play a lowish note on your keyboard or sequencer and adjust the frequency pot on the Croglin's front panel so that the filter is oscillating at around 110Hz. Now play a note three octaves above on your sequencer or keyboard and you should hear the frequency of the oscillation increase significantly. Repeat this again and again and adjust the SCL trimmer to get Filter A's oscillations to jump three octaves. For example, the filter may go from 110Hz to 880Hz. However, don’t worry about the actual pitch the VCF is producing. Just concentrate on getting roughly a three octave difference between the low note and the higher note.

It's worth noting at this point that the Croglin was never designed as a precision VCO, it's first job is as a filter, so even when calibrated it will never be as accurate as a dedicated VCO module. The Croglin makes a reasonable audio oscillator but it is not temperature compensated (so it will drift when the ambient temperature changes) and it goes flat at higher frequencies.

Additional Points

Unlike some filter topologies, the Sallen-Key filters used in the Croglin, will boost the signal level when resonance is increased. The Eurorack modular system typically uses +/-5V (10V peak to peak) audio signals and yet has only a +/-12V power supply. This doesn't allow for much headroom for signal gain in any module without running the risk of clipping the signal. Clipping is where the signal wants to go to a higher level but the circuitry won't allow it. In clipping the audio waveform's peaks are thus removed and the sound changes significantly. Clipping can be a good thing sometimes – it's how many distortion pedals work – but it's not usually welcome and it can sound particularly nasty with complex audio, or taint very pure signals.

To ensure a 10V peak to peak input signal will not clip the Croglin's output stages, even when the resonance is turned up full, there is a small drop in signal volume at low resonance. When using Input 1 alone with Output A and/or Output B, there is a -2dB signal loss. When using Input 2 either alone or with Input 1, there is a -8dB signal loss.

Croglin's main Output combines the signals from both of its filters. To allow this to happen without clipping in the worst case of both Resonance and Duality turned to maximum, the signal gain is half that of the individual outputs.

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 EU, 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
© June 2020