

Oakley Sound Systems

Eurorack Modular Series

Discrete Ladder Filter

Builder's Guide

V1.0

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the Discrete Ladder Filter (DLF) 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.

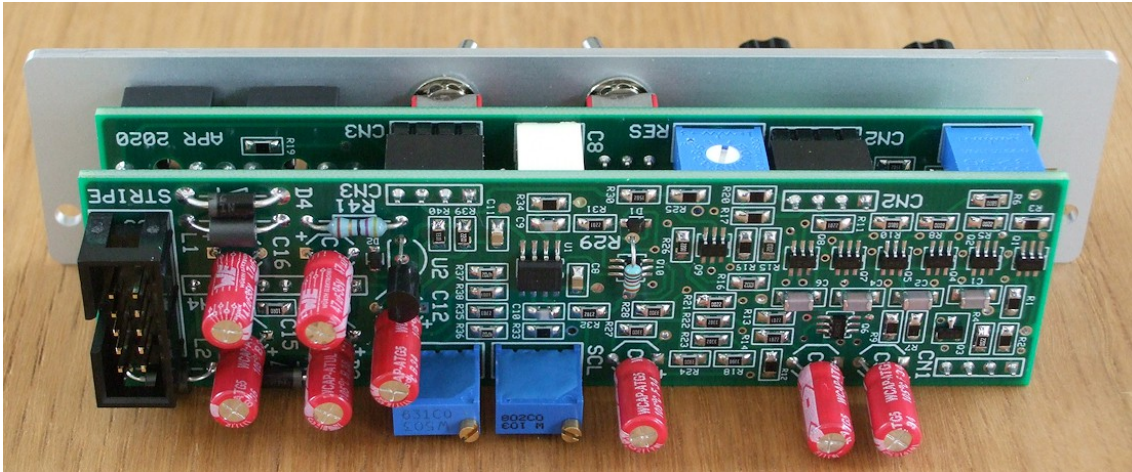


For general information regarding where to get parts and suggested part numbers please see our useful Parts Guide at the project webpage or <http://www.oakleysound.com/parts.pdf>.

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 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 larger capacitors, trimmers, ferrite beads, power diodes, and all the interconnects are through hole parts. The matched pairs are in the very small SOT457 packages but these can still be soldered easily enough with a standard soldering iron.

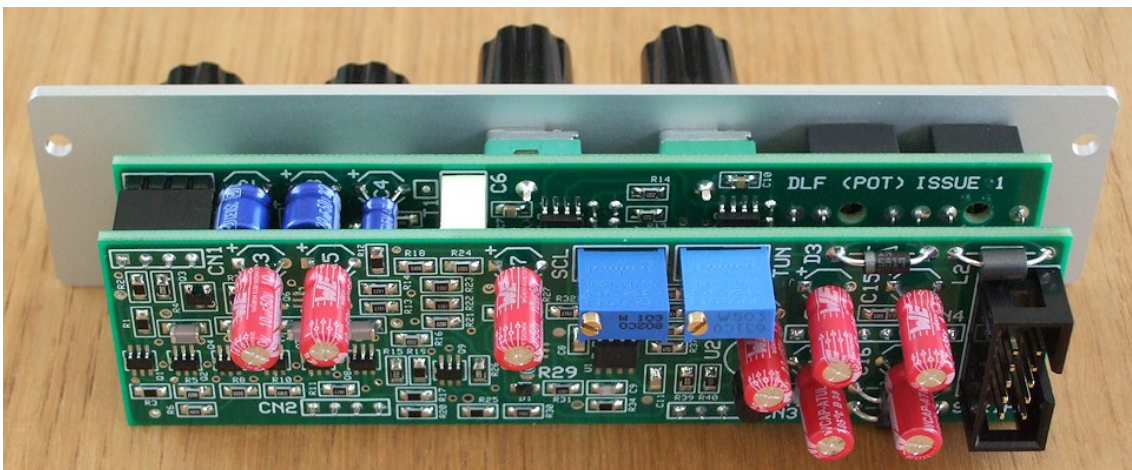
Both circuit boards are the same size, that is, 29 mm (wide) x 107 mm (high). The boards are designed to go behind a panel that is 6HP wide.



The module comprises of two circuit boards connected together with four 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 around +60mA and -50mA at +/-12V.



Discrete Ladder Filter 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. M 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.

| | |
|------|--------------|
| 33R | R17, R18 |
| 220R | R3 |
| 680R | R4, R8 |
| 1K | R19 |
| 6K8 | R7 |
| 10K | R13, R14 |
| 15K | R5, R6 |
| 47K | R1, R10, R11 |
| 68K | R2 |
| 100K | R15, R16 |
| 220K | R9, R12 |

Capacitors

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

| | |
|-------|------------------|
| 100nF | C7, C9, C10, C11 |
|-------|------------------|

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

| | |
|-----|----|
| 3n3 | C1 |
|-----|----|

The following capacitors are standard through hole components with 0.2" (5mm) spaced radial leads. Please note the maximum height cannot be larger than 11mm. Kemet R82 series are suitable.

| | |
|--------------------|--------|
| 1uF, 50V polyester | C6 |
| 2u2, 50V polyester | C5, C8 |

The following capacitors are low profile through hole electrolytic capacitors with 0.1" (2.5mm) or 0.2" (5mm) spaced radial leads. Low profile in this case means that the height of the devices must not be greater than 9mm.

22uF, 25V low profile electrolytic C2, C3
4u7, 35V low profile electrolytic C4

Discrete Semiconductors

The following devices are surface mount parts.

BCM847DS NPN dual transistor Q1
BCM857DS PNP dual transistor Q2

ICs

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

TL072ACD dual op-amp U1, U2

Trimmer (Preset) Resistors

100R side adjust multiturn OFF (eg. Bourns 3296X-101LF)
2K side adjust trimmer RES (eg. Bourns 3386X-1-202LF)

Potentiometers

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

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

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

4 way 0.1" SIL header CN1, CN2, CN3
8 way 0.1" SIL header CN4

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

3.5mm socket Thonkiconn CV1, CV2, IN, OUT

SPDT switch MODE, INVERT (eg. Thonk DW1)

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.

Main Board issue 1 Parts List

Resistors

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

| | |
|------|------------------------|
| 10R | R42 |
| 220R | R4, R5, R21, R26, R39 |
| 330R | R18, R24, R27, R28 |
| 680R | R6, R8, R10 |
| 1K | R2, R17, R19, R20, R25 |
| 1K3 | R40 |
| 2K2 | R11, R13, R14, R31 |
| 4K7 | R7 |
| 10K | R15, R16 |
| 15K | R30 |
| 27K | R32 |
| 33K | R22, R23 |
| 47K | R36, R37 |
| 100K | R38 |
| 180K | R1, R3, R33 |
| 220K | R34 |
| 270K | R35 |
| 330K | R9, R12 |

The following components are through devices.

47R 1/4W 1% metal film R41

1K +3300ppm/K temp. co. R29 (eg. Arkaneohm LT16S102F33)

R29 is placed across Q10 once the transistor pair has been soldered into place. R29 may also be a standard 1K 1/8W 1% metal film resistor if a small amount of temperature drift in the cut-off frequency can be tolerated.

Capacitors

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

100pF C9, C10

The following capacitors are surface mount, size 1206 (or metric 2012) multilayer ceramic, dielectric C0G (or NP0), working voltage 25V or 50V, tolerance +/-5%.

100nF C1, C2, C4, C6 (eg. TDK part CGA5L2C0G1H104J160AA)

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

100nF C8, C11

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

| | |
|------------------------|------------|
| 2u2, 50V electrolytic | C12 |
| 4u7, 50V electrolytic | C15, C16 |
| 10uF, 35V electrolytic | C3, C5, C7 |
| 47uF, 25V electrolytic | C13, C14 |

Discrete Semiconductors

The following devices are surface mount parts.

| | |
|------------------------------|---------------------------------|
| 1N4148WS diode | D1, D2 |
| BC860 PNP transistor | Q3 |
| BCM847DS NPN dual transistor | Q1, Q2, Q4, Q5, Q7, Q8, Q9, Q10 |
| BCM857DS PNP dual transistor | Q6 |

The following devices are standard through hole parts.

1N5819 Schottky diode D3, D4

Integrated Circuits

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

TL072ACD dual op-amp U1

The following device is a standard through hole part.

79L05 -5V regulator U2

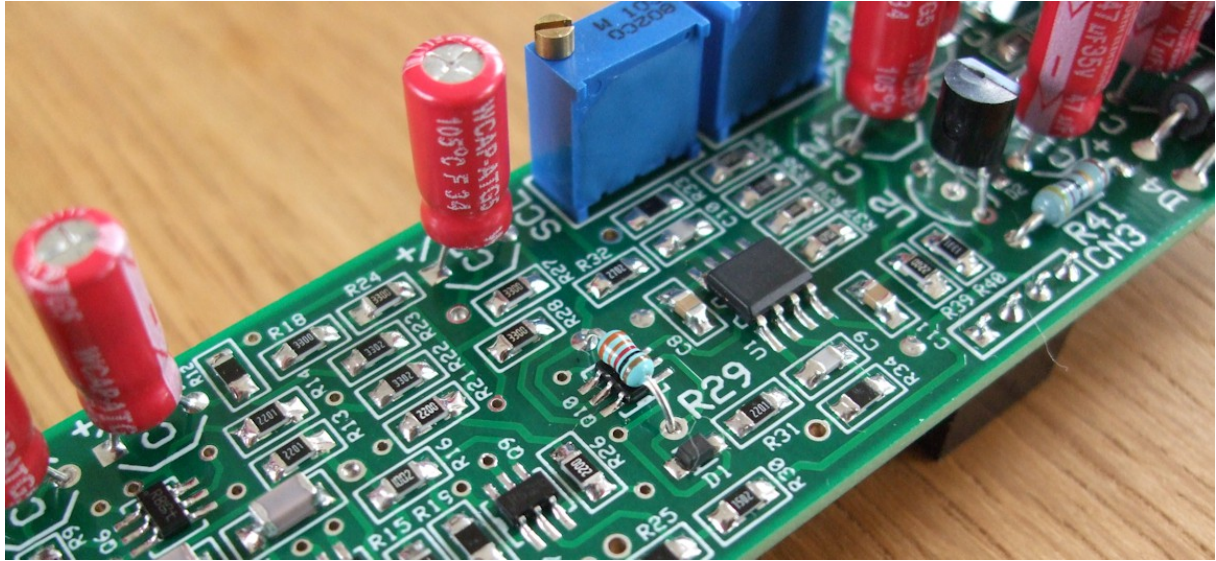
Trimmer (Preset) Resistors

| | | |
|---------------------------|-----|----------------------------|
| 10K top adjust multiturn | SCL | (eg. Bourns 3296W-1-103LF) |
| 100K top adjust multiturn | TUN | (eg. Bourns 3296W-1-104LF) |

Miscellaneous

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

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.



R29 is a positive temperature coefficient resistor and is fitted so that it straddles and makes contact with the top of matched NPN transistor pair Q10.

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.

Note that the Drive 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 switches are a standard sub-miniature type that are 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 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 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 |
| 8 way socket | 2212S-08SG-85 |
| 8 way header | 2211S-08G |

Populating the Circuit Boards

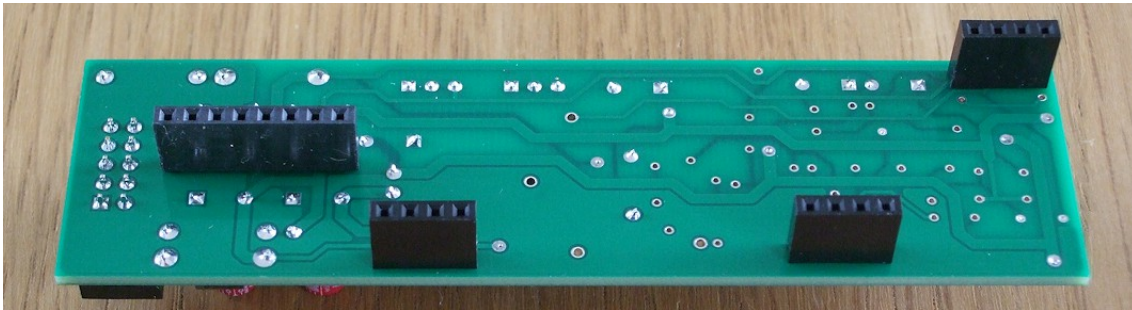
For general information on how to build Oakley modules please see my generic Construction Guide at the project webpage or <http://www.oakleysound.com/construct.pdf>.

Main Board Construction

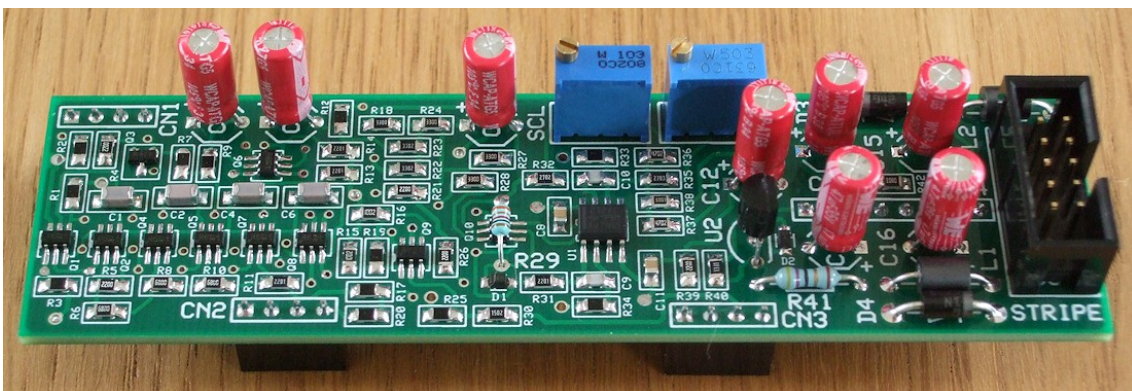
All the surface mount components should be soldered first. Take care to treat all transistors, diodes and ICs as static sensitive devices. I usually solder the resistors first, then the capacitors, then the discrete semiconductors, and then the ICs.

Note that the four ladder filter capacitors, C1, C2, C4, and C6, are of a different size to all the other surface mount capacitors.

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.

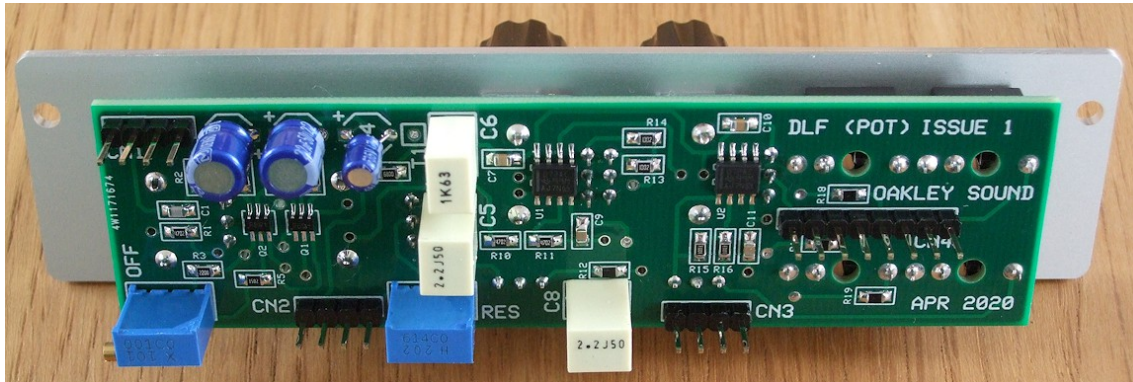


The next items to be soldered will be the through hole components. Remember that diodes and the standard electrolytic capacitors are polarised so they need to be fitted the right way around. C6 being a non polar capacitor can go any way around. You should especially make sure that the boxed header is 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. Make sure that the three electrolytic capacitors do not stick up too high so the main board cannot be fitted safely into place. It is imperative that the leads of C3 and C5 on the underside of the main board do not touch the tops of the three electrolytic capacitors on the pot board.

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 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 -20mA from the negative rail and +30mA from the positive rail. Anything significantly more than this, like 50mA, 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 -50mA. Anything significantly different to this, say over 75mA, will indicate a problem.

Set the mode switch to Mode A 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 should be around the same volume as the input signal. If you can monitor the signal on an oscilloscope then you should see that the top and bottom parts of the sawtooth waveform have been clipped at around +5.5V and -5.5V respectively. This is the action of the discrete input stage.

Change the mode to Mode B. There should be a slight change in the tone heard but not hugely so. If you can monitor the waveform you will see that the signal has now been inverted, and that the negative peak of the signal becomes extended downwards.

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

Turn the frequency back to full and slowly turn up the emphasis (resonance). You will hear the characteristic drop in volume of an uncompensated ladder filter. At high values of emphasis the module may start to self-oscillate, although this may be well above the range of your hearing or loudspeakers. Bring down the frequency and see if you can hear the self-oscillation. 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.

Turn up the drive control. The output signal should get slightly quieter as the increasing input signal is compressed by the overdriven filter circuitry.

Patch an LFO into the CV2 input. Check that the CV2 Depth pot controls the amount of modulation of the DLF'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...!'.
yit...!.

The module is now ready for calibration.

Calibration

OFF (Pot Board)

Measure the voltage at the test point on the pot board with no signal input. The test point is an empty solder pad near the edge of the board in between C4 and C6. The voltage should be measured between the test point and 0V. You can use the metal barrel of one of the sockets on the front panel as a suitable 0V connection for your voltmeter. Adjust OFF so that the voltage on the test point is as close as you can get it to 0.00V.

RES (Pot Board)

This determines where you want the resonance to occur as you turn up the Emphasis pot. Set the frequency control to its middle position and the emphasis control to 3 o'clock. Adjust the trimmer so that the module is just starting to oscillate.

SCL (Main Board)

This adjusts the 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.

Plug a 1V/octave source into the CV1 socket. This may be your keyboard or sequencer's pitch CV output, or the CV output of a midi-CV convertor. Set the emphasis pot fully clockwise to get the filter oscillating. Monitor the audio output of the module. Play a lowish note on your keyboard or sequencer and adjust the frequency pot on the DLF's front panel so that the filter is oscillating at around 220Hz. Now play a note two 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 the filter's oscillations to jump two octaves. For example, the filter may go from 220Hz to 880Hz. However, don't worry about the actual pitch the VCF is producing. Just concentrate on getting roughly a two octave difference between the low note and the higher note.

It's worth noting at this point that the 904A was never designed as a precision VCO, it's first job was as a filter, so even when calibrated it will never be as accurate as a dedicated VCO, or even a more modern VCF design.

You may also notice that after the CV changes quickly the oscillation takes a short while to settle to its new frequency. This appears to be due to the time taken for the two transistors in the exponential convertor to reach a new thermal equilibrium.

TUN (Main Board)

This adjusts the filter's cut-off frequency. Adjust this so that the range of the DLF's frequency pot covers your chosen range. In a polyphonic modular, this can be used to make each voice's VCF behave identically. However, for most users the TUN trimmer can be just left in its default middle position.

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 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

© May 2020