Oakley Sound Systems

5U Oakley Modular Series

DFL – Discrete Ladder Filter

PCB Issue 1

Builder's Guide

V1.3

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the issue 1 Discrete Ladder Filter 5U module from Oakley Sound. This document contains a basic introduction to the board, a full parts list for the components needed to populate the boards and a list of the various interconnections.

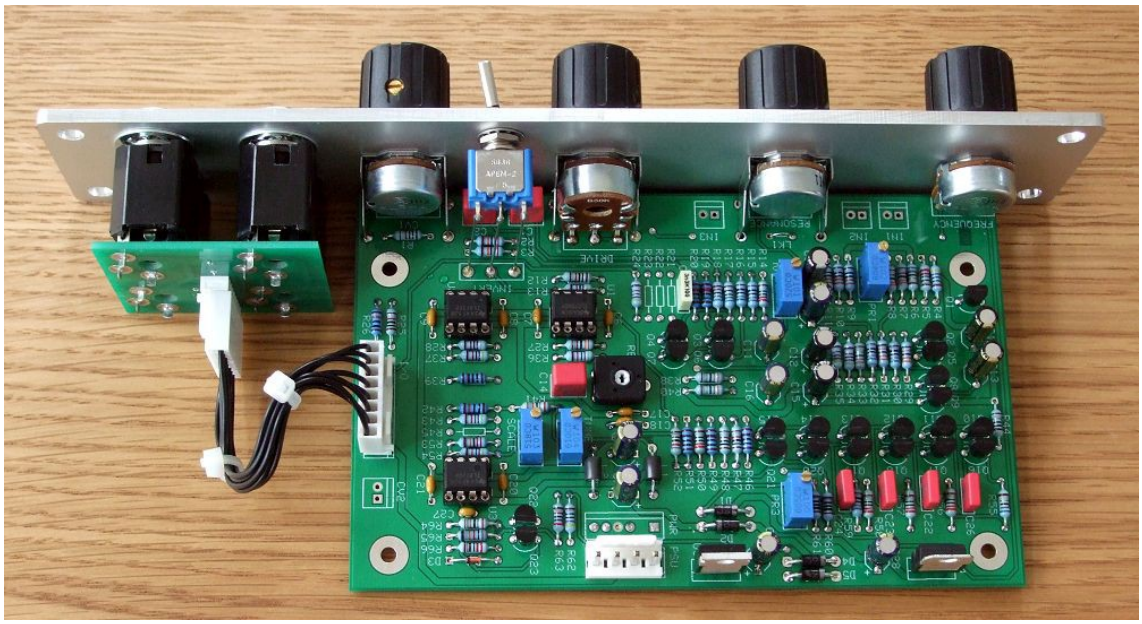
For the User Manual, which contains an overview of the operation of the unit and the calibration procedure, please visit the main project webpage at:

http://www.oakleysound.com/dlf.htm

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, mounting front panel components and making up board interconnects please see our generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

The Discrete Ladder Filter PCB



The Oakley Discrete Ladder Filter as a single width MOTM format module in a natural finish Schaeffer panel. Note also the use of the optional Sock4 socket board to help keep the wiring to the sockets neat and tidy.

I have provided space for the four main control pots on the PCB. If you use the specified pots and brackets, the PCB can be held firmly to the panel without any additional mounting procedures. The pot spacing is 1.625”. The pots are cut off frequency, resonance, drive and a reversible attenuator for a CV input. Please note that this issue 1 has two unwanted errors that make mounting the pots more difficult than normal. Please see the section later in this document that details the recommended method used for mounting the pots.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 80mA for the +15V rail and 55mA for the -15V rail. Power is routed onto the main PCB by either a four way 0.156” MTA156 type connector or the special five way Synthesizers.com MTA100 header. 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/panel 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 main PCB has four mounting holes for M3 bolts, one near each corner. These are not required for panel mounting if you are using the three 16mm pot brackets.

The board size is 89mm (deep) x 143mm (high).

The main board has been laid out to accept connection to our Sock4 socket board. This small board speeds up the wiring of the four sockets and reduces the chances of mistakes.

Discrete Ladder Filter issue 1 Parts List

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.

The components are grouped into values, the order of the component names is of no particular consequence.

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.

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

Resistors

1% 0.25W metal film types are to be recommended.

Components marked with a * are only to be fitted if building the full 2U version. For those building the 1U filter core version you should not fit these parts.

10R	R44
33R	R40, R33, R38
68R	R6, R55
100R	R61, R60
220R	R57, R56, R50, R14, R58
330R	R11, R9, R10
470R	R62, R4
680R	R59, R15, R32, R30, R16
1K	R46, R39, R51, R49
2K2	R5, R66
6K8	R17
8K2	R52
10K	R1, R47, R48, R25, R12
15K	R19, R20
27K	R63
33K	R27, R8, R7
47K	R35, R13, R28, R37, R43, R54, R34, R24, R36, R23*, R22*, R21*
68K	R18
100K	R26, R42, R45*
220K	R53, R3, R2
270K	R64, R41
330K	R29, R31

1K +3000ppm/K TC	R65

Capacitors

100nF axial multilayer ceramic	C21, C17, C18, C8, C20, C9, C6, C7
470pF C0G 2.5mm ceramic	C27
3n3, 100V radial polyester	C4
100nF polyester film box	C24, C23, C22, C26
2u2, 50V radial polyester	C2, C14, C1
2u2, 63V electrolytic	C19, C25
4u7, 63V electrolytic	C28, C29
10uF, 35V electrolytic	C12, C3, C15
22uF, 35V electrolytic	C11, C16
47uF, 25V electrolytic	C10, C13

If the resonance pot is a 50KC (or 47KC) pot then C5 should be 2u2, 63V electrolytic capacitor. If the resonance pot is a 10KB pot then C5 should be a 4u7, 63V electrolytic capacitor.

Discrete Semiconductors

1N4001 diode	D1, D5, D2, D4
1N4148 small signal diode	D3
BC550C NPN transistor	Q8, Q9
BC560C PNP transistor	Q1

BC550C NPN transistors to be matched in the following pairs:

Q10 & Q16
Q11 & Q17
Q12 & Q18
Q13 & Q19
Q14 & Q20
Q15 & Q21
Q4 & Q7
Q22 & Q23

BC560C PNP transistors to be matched in the following pairs:

Q2 & Q5
Q3 & Q6

All matched pairs of transistors have to be matched for Vbe to +/-1mV.

Integrated Circuits

TL072ACP dual JFET op-amp	U1, U2, U3
7812 12V linear regulator	U4
7906 6V linear regulator	U5

Trimmers (preset) resistors

10R cermet multiturn	PR3
100R cermet multiturn	PR2
1K cermet multiturn	PR1
10K cermet multiturn	SCALE
100K cermet multiturn	TUNE
2K or (2K2) horizontal single turn	RES

The multiturn trimmers are Bourns 3296W or similar.

Potentiometers (Pots)

All pots Alpha 16mm PCB mounted types. See section 'Issue 1 PCB errors' for mounting details.

47KB (or 50KB) linear	FREQUENCY, CV1 (labelled CV2 on the front panel)
47KB (or 50KB) dual/stereo linear	DRIVE
10KB linear or 50KC reverse log	RESONANCE

If the resonance pot is 10KB then wire link LK1 must be fitted.

Three 16mm pot brackets.

Switch

A 'single pole double throw' SPDT toggle switch is required for the mode selection.

Miscellaneous

Leaded axial ferrite beads	F1, F2	
MTA156 4 way header	PSU	– Oakley/MOTM power supply
MTA100 6-way header	PWR	– Synthesizers.com power supply
Molex/MTA 0.1” header 8-way	I/O	– for connecting to sockets
Molex/MTA 0.1” housing 8-way	I/O	– for connecting to sockets

Other Parts Required

Switchcraft 112APC 1/4” sockets Four off mounted either on the Sock4 board or on panel

Four knobs

Two cable ties.

Around 2m of insulated multistrand hook up wire for the switch and socket connections.

Offboard Pots (2U format only)

47KA Log	INPUT 1, INPUT 2, INPUT 3
47KB Linear	CV1 DEPTH

Components required if using optional Sock4 board

Molex/MTA 0.1” header 8-way	I/O
Molex/MTA 0.1” housing 8-way	I/O

112APC Switchcraft 1/4” socket SK1, SK2, SK3, SK4

A single wire link is to be fitted to L1 on the Sock4 PCB. L2 is left open.

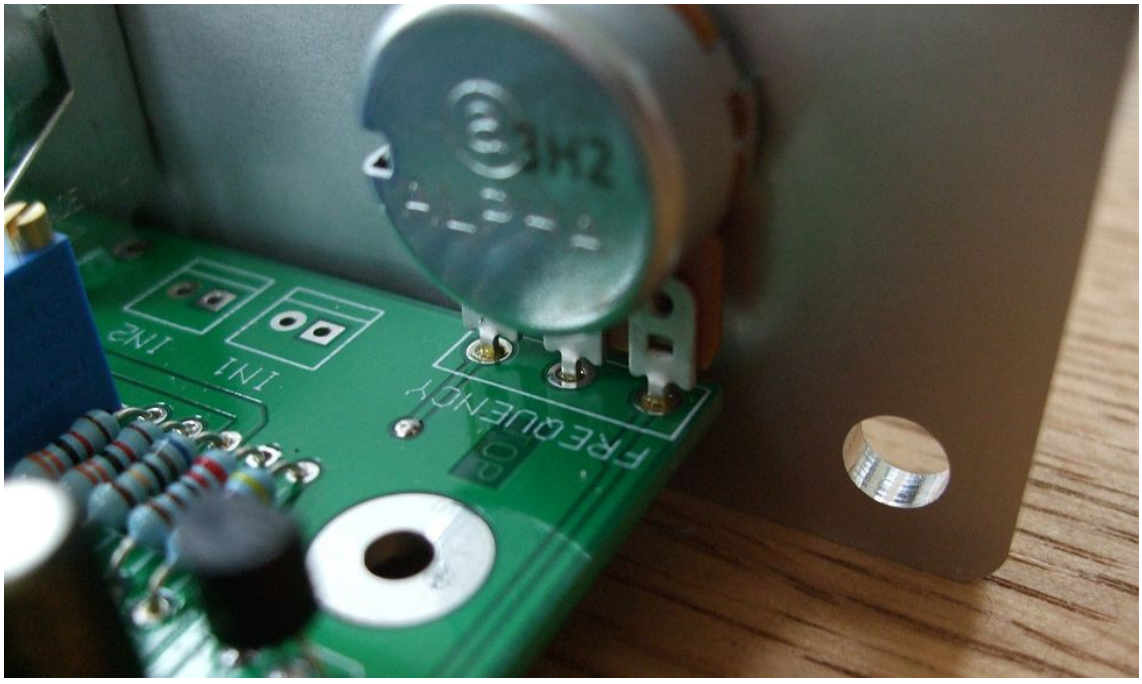
If using Molex KK you'll also need at least 16 crimp terminals.

Suitable lengths of wire to make up the single 100mm interconnect and two cable ties.

Issue 1 PCB Errors

As its name implies the issue 1 PCB is the very first version of the design excepting my breadboarded version I used to test and tweak the circuit. As such a few mistakes can creep in and the DLF is no exception to this. For the first time in a long while I decided to implement a double copper flood area which carries the 0V to all parts of the circuit. This copper flood covers all the areas on both sides of the board not covered by copper traces and is isolated from the traces by a very narrow air gap.

The problem is that some components, namely the 16mm pots and pot brackets I use in Oakley projects actually overlap their solder pads and bridge this narrow air gap on the top surface of the board. This means that if the components are placed into the board as they would be normally the solder tags of the pots and solder brackets are shorted to ground. In practice the generous solder mask, that's the green layer, will probably prevent the two metal surfaces from touching but this is not to be guaranteed. So what we need to do is make sure this does not happen by mounting the pots slightly higher than they would normally be.



The top pot (frequency) mounted so that it sits 1mm above its normal position.

The best way I found to do this is to solder the top pot first. Gently hold the pot in place when you solder the middle pin of this pot. Hold it so that the two parts on either side of the solder tang are at least 1mm above the top surface of the board. Then when the middle pin is secure and in the right place, solder the other two pins making sure the pot stays level and the edges of all the three pins are more or less the same height above the board.

Now repeat this for the bottom pot. You'll have a solder bracket to fit too for this pot but don't solder the bracket just yet.

Right that's those two pots done, now let's try to fix the next error. Wait, there's another mistake? Yep, this time old numpty features really let it slide. The library layout I used for the dual gang pot was incorrect and the distance between the front set of pads and the rear set of pads is too big for a 16mm Alpha pot. Thankfully, because of the cock up with the copper flood and having to mount the pots 1mm off the board, this gives us a neat way of fixing this one too. Carefully bend the rear three tangs of the pot outward so that they stick out backwards at just under ninety degrees from where they were.

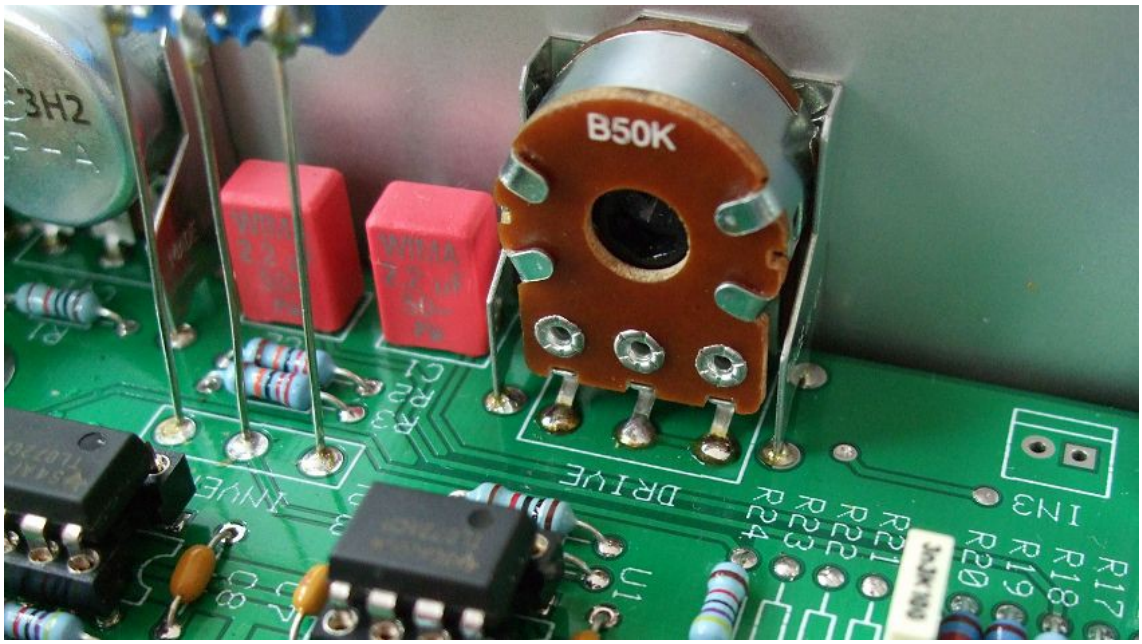
Now fit the drive pot and bracket in their places and you'll notice the rear set of solder tangs of the drive pot sit right above the misplaced solder pads on the board. Don't solder anything just yet though.

Fit the resonance pot and bracket into their places. Again don't solder them just yet.

Carefully fit the PCB with the two loose pots to the front panel. Gently tighten the nuts up on the top and bottom pots and position the PCB so that it sits at right angles to the panel's surface. Solder the pot brackets belonging to the frequency and CV2 pots.

Now you should find that the resonance and drive pots are sitting at exactly the right height above the board – the front panel holding them in the correct place. Solder the resonance pot, the front pins of the drive pot and their pot brackets as you would normally.

Then from the topside of the board allow solder to flow onto each of the rear pins of the drive pot and allow the solder to flow down into the respective solder pads. If the three pins are floating too high above their solder pads then gently push them down with a screwdriver blade so that they touch the pads before you solder them in place.



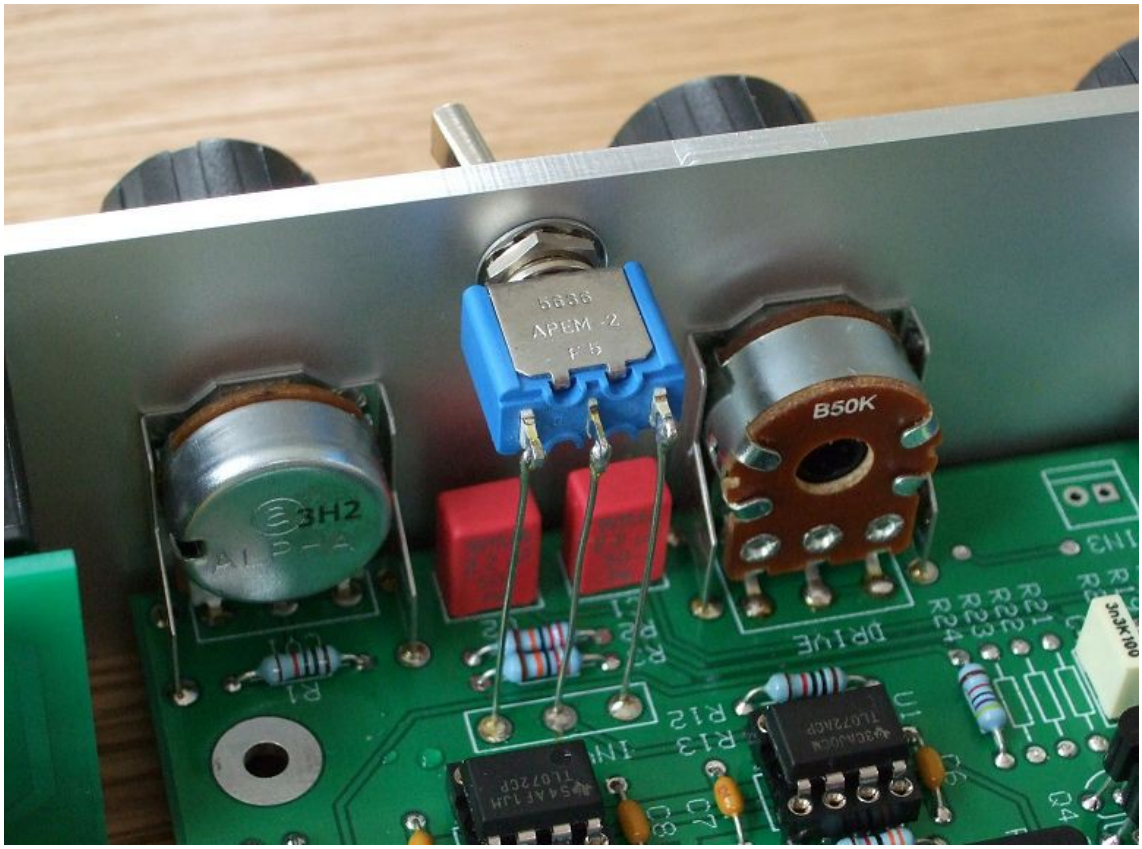
The dual gang drive pot and pot bracket fully soldered into the board. Notice how the pot and the bracket sit just above the board not touching any of the green solder mask layer.

Wiring the Switch

The Discrete Ladder Filter module has one toggle switch to allow use of either the inverting or non inverting output of the CP3 input stage.

You should wire the switch as you would other Oakley modules. I typically use thin solid core wire rather than insulated multi-strand wire. This keeps the connection firmly in place and very neat. I normally bend the wire at one end into a hook and place the straight end into the PCB pad's hole. I then loop the hooked end around the switch tang and squash the hook into place before soldering it. The solder pad on the board can then be soldered from the underside and the excess wire on snipped off.

The SPDT switch should have three solder tags. All three tags need to be connected to the board, each tag simply connecting to the solder pad directly below it.



The two way switch fitted and soldered. Note the use of the crinkle washer on the inside of the panel.

Connections

Power connections – MOTM and Oakley

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

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

Pin 1 on the I/O header has been provided to allow the ground tags of the jack sockets to be connected to the powers supply ground without using the module’s 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.

Power connections – Synthesizers.com

The PWR power socket is to be fitted if you are using the module with a Synthesizers.com system. In this case you should not fit the PSU header. The PWR header is a six way 0.1” MTA, but with the pin that is in location 2 removed. In this way location 3 is actually pin 2 on my schematic, location 4 is actually pin 5 and so on.

Power	***Location number***	***Schematic Pin number***
+15V	1	1
Missing Pin	2	
+5V	3	2
Module GND	4	3
-15V	5	4
Not connected	6	5

+5V is not used on this module, so location 3 (pin 2) is not actually connected to anything on the PCB.

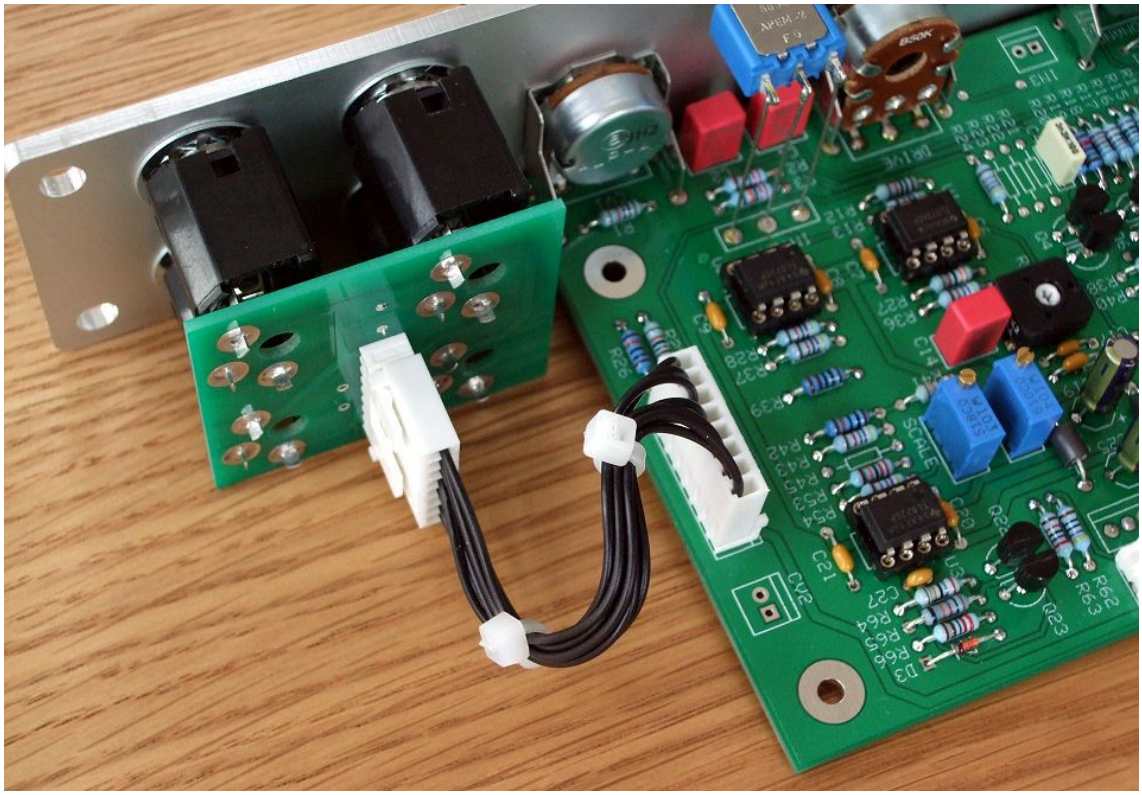
If fitting the PWR header, you will also need to link out pins 2 and 3 of PSU. This connects the panel ground with the module ground. Simply solder a solid wire hoop made from a resistor lead clipping to join the middle two pads of PSU together.

Building the Filter Core 1U wide module using the Sock4 board

This is the simplest way of connecting all the sockets to the main board. The Sock4 board should be populated in the way described in our construction guide found on the project webpage. There is only one eight way header and it is to be fitted to the bottom side of the board.

Do not forget to solder in the wire link L1. Link L2 must be left open.

You need to make up only one eight way interconnect. It should be made so that it is 100mm long.



The issue 1 PCB fitted to a 1U wide 'filter core' module showing the detail of the board to board interconnect. Here I have used the Molex KK 0.1" system to connect the Sock4 to the main PCB.

Hand wiring the sockets for the filter core 1U wide module

If you have bought Switchcraft 112A sockets you will see that they have three connections. One is the earth or ground tag. One 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) tag. The NC tag is internally connected to the signal tag when a jack is not connected. This connection is automatically broken when you insert a jack.

Once fitted to the front panel the ground tags of each socket can be all connected together with solid wire. I use 0.91mm diameter tinned copper wire for this job. It is nice and stiff, so retains its shape. A single piece of insulated wire can then be used to connect those connected earth tags to pin 1 of I/O. Pin 1 is the square solder pad.

All the other connections are connected to the signal or NC lugs of the sockets. The tables below show the connections you need to make:

Pin	*Pad name*	*Socket Connection*	*Lug Type*
Pin 1	PANEL_GND	Connect to all sockets	Earth lugs
Pin 2	CV2	Connect to CV2	Signal lug
Pin 3	GND	Connect to CV2	NC lug
Pin 4	1V/OCT	Connect to CV1	Signal lug
Pin 5	GND	Connect to CV1	NC lug
Pin 6	AUDIO_OUT	Connect to OUTPUT	Signal lug
Pin 7	GND	Connect to INPUT	NC lug
Pin 8	AUDIO_IN	Connect to INPUT	Signal lug

2U wide full format Discrete Ladder Filter

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. As with the 1U module, you need to ground the sockets' earth lugs. Do this by joining together the earth lugs for each row with stiff single core wire. Then with a piece of insulated wire, or a well placed piece of stiff wire, connect together the two horizontal pieces of stiff wire. Now all your socket ground lugs are connected together. Then with a piece of insulated wire connect one of the stiff pieces to pin 1 of the I/O header on the PCB. Pin or pad 1 of I/O is connected to panel ground on the power sockets, ie. pin 3 on the MTA/Molex connectors.

The pads in box labelled I/O at the bottom of the board are mainly set up for the 1U version, but you will need to use some of these pads too.

Pin	*Pad name*	*Socket Connection*	*Lug Type*
Pin 1	PANEL_GND	Connect to all sockets	Earth lugs
Pin 2	CV2	Connect to CV2	Signal lug
Pin 3	GND	Connect to CV2	NC lug
Pin 4	1V/OCT	Connect to 1V/OCT	Signal lug
Pin 5	GND	Connect to 1V/OCT	NC lug
Pin 6	AUDIO_OUT	Connect to OUTPUT	Signal lug
Pin 7	GND	Connect to IN1, IN2, IN3	NC lugs
Pin 8	AUDIO_IN	Not used	

All your other connections will be made via the four two way 0.1" headers, three of which are situated on the board near the pots and the fourth at the bottom of the board. They are labelled and positioned appropriately to help you connect up your module correctly. These are CV2 (which goes to the CV1 pot), IN1 (INPUT1 pot), IN2 (INPUT2 pot) and IN3 (INPUT3 pot).

Pots have three pins. For all four 'off-board' pots two of these pins (the CCW and wiper pins) will be connected to PCB via those two way headers, whilst the remaining pin (the CW pin) will be connected to the appropriate socket's signal lug.

CW is 'clockwise' end of pot's resistive track (from the rear this is the left hand side with the pins facing down). CCW is the 'counter-clockwise' end of the pot. The wiper is the middle pin.

The middle pin of the pots, the wiper, will carry the signal to the appropriate two way 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 it is 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 (CCW) 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.

Now each pot will have one unsoldered pin left, the CW pin. Connect these to the appropriate socket. The wire should go to the signal lug of the socket. The pot labelled INPUT1 goes to the signal lug on the socket labelled IN1, 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. Click the switch into position INV. Set the DRIVE pot to the far left to put the filter into its clean mode. Moving the FREQUENCY control should produce the usual and distinctive filter effect from the output.

Turning the Resonance up will accentuate the ‘electronic’ nature of the sound. Check that at maximum resonance the filter output will oscillate or get very close to it. We can actually set the point at which resonance starts with a trimmer so don't worry at this stage if it doesn't actually oscillate. 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.

Turn up the drive control and the output should change in timbre as well as a slight change in volume. If you have the resonance turned up the resonant peak should be less obvious and the sound will become more distorted. If the resonance is low then the sound will become more hollow sounding.

Listening to the output with the sawtooth input still connected, patch a LFO or EG output to the CV1 input. The 1V/octave input sensitivity of the CV1 input should produce large sweeps of cut-off, so you may want to patch in a Multimix or other attenuating module to have some control over the sweep depth.

Now try the CV2 input. This should have the same affect as the CV1 input but you'll be able control the depth of the modulation. Notice that the minimum sweep depth should occur with the CV2 pot at its mid point. Use a sawtooth waveform on your LFO, and see if the CV2 depth pot allows you to invert the modulation input. You should get a ‘dow-dow-dow...’ from one side and a ‘yit-yit-yit...’ from the other.

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

If all this happens, the chances are that you have a working module and the unit is ready for calibration. The calibration details can be found in the User Manual.

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