Oakley Sound Systems

5U Oakley Modular Series

The SVF issue 5
Voltage Controlled State Variable Filter

Builder's Guide

V5.5

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the issue 5 State Variable 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 board, interconnections and some basic testing methods.

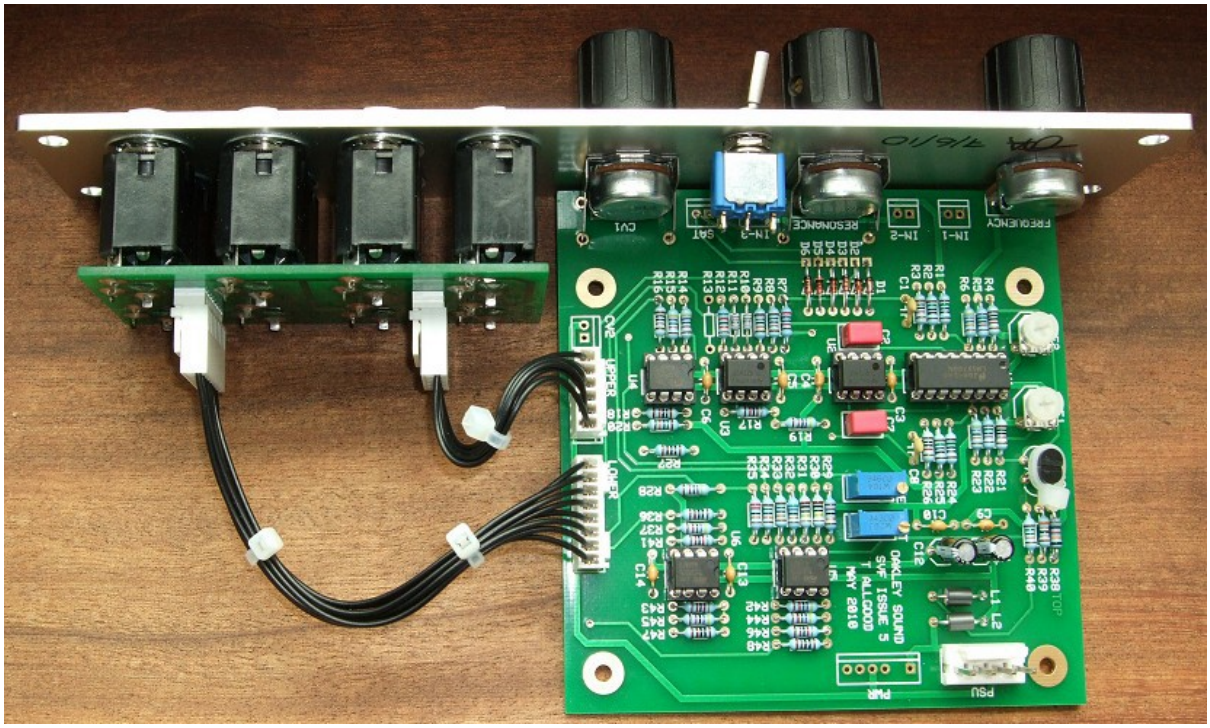
For the User Manual, which contains an overview of the operation of the unit, advice on connecting the unit and calibration procedures, please visit the main project webpage at:

http://www.oakleysound.com/svf.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 SVF issue 5 PCB



This is the prototype issue 5 module with natural finish Schaeffer panel. Note the use of the Sock8 board to mount the jack sockets. Two pot brackets are used to hold the PCB firmly to the front panel.

I have provided space for the three main control pots on the PCB. If you use the specified 16mm Alpha pots and matching brackets, the PCB can be held firmly to the panel without any additional mounting procedures. The pot spacing is 1.625” and is the same as the vertical spacing on the MOTM modular synthesiser and most of our other modules.

The design requires plus and minus 15V supplies. The power supply should be adequately regulated. The current consumption is about 30mA for each rail. Power is routed onto the PCB by 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 PCB has four mounting holes for M3 bolts, one near each corner. These are not required if you are using the two 16mm pot brackets.

The issue 5 board size is 104mm (high) x 104mm (deep), while the issue 5.1 board size is 104mm (high) x 99mm (deep). Electrically both issue 5 and issue 5.1 are the same. Issue 5.2 adds a panel ground connection to the MU power header and changes the size of the two timing capacitors in the filter core.

The board has been laid out to accept connection to our Sock8 socket board. This small board speeds up the wiring of the eight sockets and reduces the chances of mistakes.

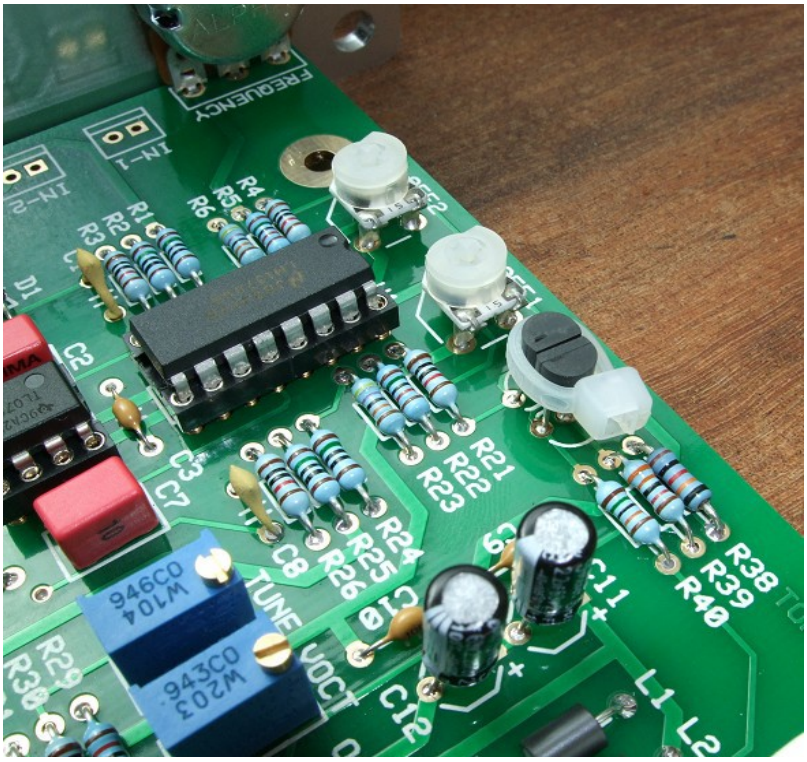
Components

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.

Some special considerations for this project

The op-amps can be any decent medium (audio) speed op-amp, eg. TL072 or 4558. However, U2 should have a low input bias current and a FET input op-amp is to be recommended. The TL072 is perfectly good enough for this, although you could use the LF412 or AD712.

For the 470pF capacitors in positions C2 and C7, you must use good quality capacitors. Failure to use low tolerance parts in these locations will mean your filter will be unlikely to oscillate over the full audio range in the HIGH setting. I have suggested using 2.5% (or better) tolerance 63V polypropylene film types. You can also use 2.5% polystyrene capacitors. These are generally axial types, but radial types do exist. These are little silver coloured cylinders, and offer exceptional quality, but are easy to melt with your soldering iron. Another good quality capacitor one could use is the silvered mica type. These are quite expensive but offer exceptional performance in a small package.



Close up showing the two transistors of the exponential pair held together with a cable tie. The red capacitors you can see are 5mm polypropylene film types.

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

All resistors should be 0.25W 1% metal film (MF) types unless stated.

150R	R25, R2, R24, R1
1K	R27, R47, R48, R36
1K +3000ppm/k PTC	R38
10K	R21, R4, R16, R18, R26, R3
15K	R5, R22
27K	R7
33K	R39
47K	R42, R37, R45, R44, R28, R14, R19, R15
62K	R12
82K	R43, R17, R41, R46
100K	R33, R20, R8, R34, R9
130K	R29
150K	R40
220K	R35
330K	R10, R11, R13*
470K	R23, R6
560K	R32
1M	R31, R30

* R13 fitted only in 2U version

Capacitors

100nF, 63V axial ceramic	C3, C4, C5, C6, C9, C10, C13, C14
4p7 C0G ceramic*	C1, C8
470pF 1% C0G ceramic **	C2, C7
2.2uF, 35V electrolytic	C11, C12

* C1 and C8 have a lead spacing of 5mm for the issue 5 SVF and a lead spacing of 2.5mm for issue 5.1 and 5.2.

** The issue 5 and issue 5.1 PCB board locations for C2 and C7 were originally designed for 5mm 1% polypropylene or silver mica capacitors, and you can still use these if you wish. However, it is easier to get good quality 5mm C0G capacitors which work equally as well. For the most recent issue 5.2 boards it is expected that you will use 2.5mm C0G capacitors.

Discrete Semiconductors

1N4148 signal diode	D1, D2, D3, D4
3V6 500mW zener diode	D6
6V2 500mW zener diode	D5
BC549 NPN transistor	Q2
BC559 PNP transistor	Q1

Integrated Circuits

LM13700 dual OTA	U1
TL072 dual op-amp	U2, U3, U4, U5, U6

Trimmers

100K trimmer (eg. Bourns 3386F)	OFF1, OFF2
20K multiturn trimmer (eg. Bourns 3296W)	V/OCT
100K multiturn trimmer (eg. Bourns 3296W)	TUNE

Potentiometers

50K linear Alpha 16mm	CV1-DEPTH, FREQUENCY
10K linear Alpha 16mm	RESONANCE

Two Alpha pot brackets are also required. The FREQUENCY pot has no bracket.

Switch

SPST or SPDT toggle switch	RESONANCE MODE

The lower two contacts on the switch connect to the PCB pads marked SAT.

Miscellaneous

Leaded Ferrite beads	L1, L2
Knobs to fit 6mm shafts	Three off

Cable tie for holding Q1 and Q2 together

Molex or MTA 4 way header	PSU – Oakley/MOTM power supply
MTA100 6-way header	PWR – Synthesizers.com power supply

Molex/MTA 0.1” header 6-way	UPPER – for connecting to sockets
Molex/MTA 0.1” header 8-way	LOWER – for connecting to sockets
Molex/MTA 0.1” housing 6-way	UPPER – for connecting to sockets
Molex/MTA 0.1” housing 8-way	LOWER – for connecting to sockets

1/4” sockets	Eight off mounted either on the Sock8 board or on panel

Hook up wire (26awg).

Additional components required if using optional Sock8 board

Molex/MTA 0.1” header 6-way	UPR
Molex/MTA 0.1” header 8-way	LWR
Molex/MTA 0.1” housing 6-way	UPR
Molex/MTA 0.1” housing 8-way	LWR

112APC Switchcraft 1/4” socket	SK1, SK2, SK3, SK4, SK5, SK6, SK7, SK8

If using Molex KK you'll also need at least 28 crimp terminals.

Suitable lengths of wire to make up the two interconnects and three cable ties.

You need to fit a wire link in L1 on Sock8 issue 2 boards.

Additional parts required for the 2U version

You won't be needing the Sock8 board since all the sockets will need to be wired up individually.

Make sure that R13 is fitted and is 330K.

Miscellaneous

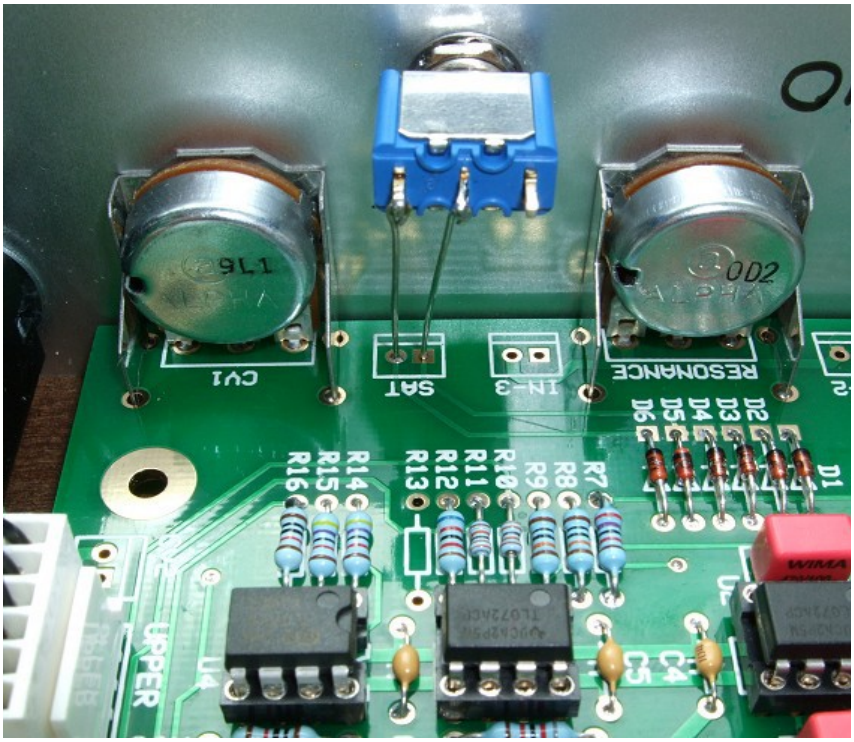
1/4” sockets	IN3, 1V/OCT

Offboard Pots (2U format only)

47K Log	IN1, IN2, IN3
47K Linear	CV2

Mounting the Resonance Mode switch

The switch is connected so that the when the toggle is in the up position the two connections of SAT are shorted together. You will need to connect your switch so that the lower two lugs of the switch are connected to the SAT pads on the board. This can be done with either insulated multistrand wire or solid core wire.



This is an APEM toggle switch wired to the PCB with solid core wire. Note also that R13 is not fitted as this is a 1U wide filter core module.

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 LWR header has been provided to allow the ground tags of the jack sockets to be connected to the power 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
Socket Ground *	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 and using it with a standard MU power distribution system, you will also need to connect together the middle two pads of the PSU header on the main board. This link connects the socket and panel ground with the module ground. Simply solder a solid wire hoop made from a resistor lead clipping, or bit of solid core wire, to join to the two middle pads of PSU.

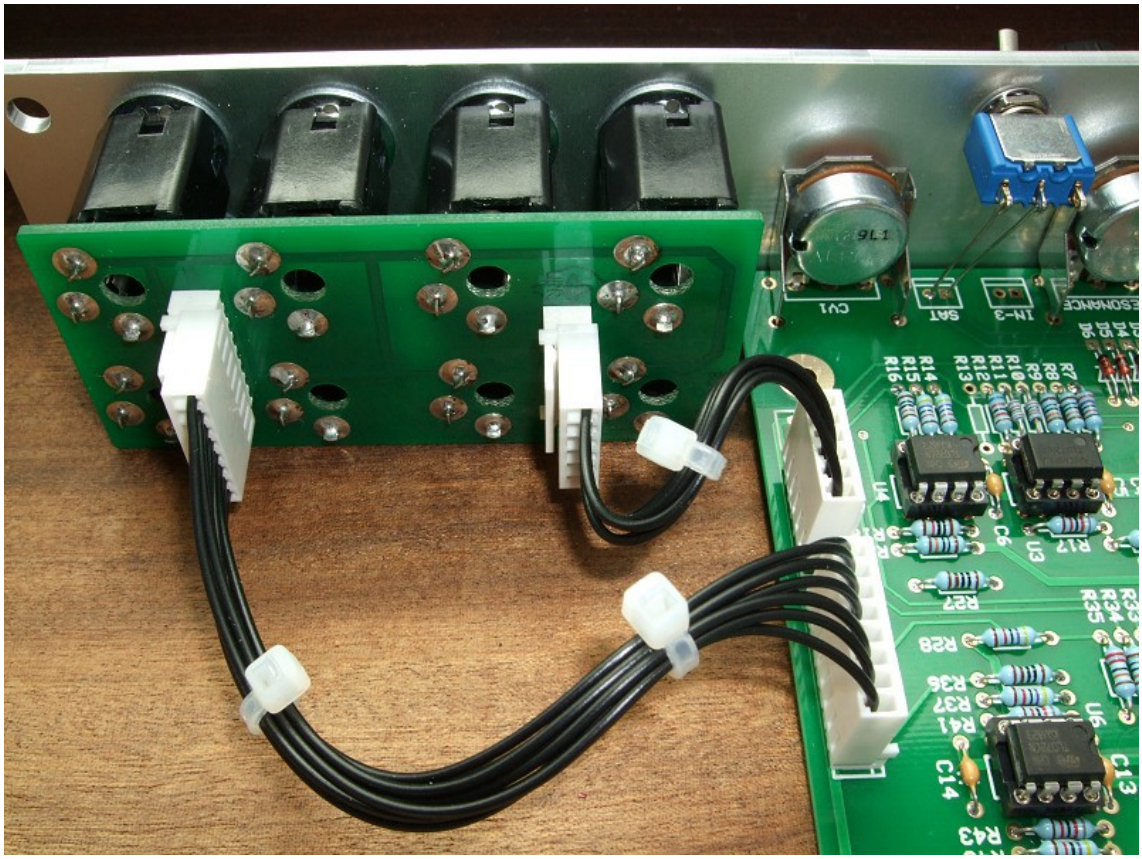
* Issue 5.2 boards connect the normally unused pin 6 of the MU connector to socket ground. With the link on PSU not fitted, and using an Oakley MU Dizzy distribution board with a five way power cable, will allow the socket ground to be kept separate from module ground to prevent ground loops.

Building the 1U Filter Core module using the Sock8 board

This is the simplest way of connecting all the sockets to the main board. The Sock8 board should be populated in the way described in our construction guide found on the project webpage. There are only two headers, UPR (for upper) which is six way, and LWR (for lower) which is eight way. Both headers are fitted to the bottom side of the board.

If you have an issue 2 Sock8 board. Don't forget to fit a wire link in position L1.

You need to make up two interconnects. The six way one should be made so that it is 80mm long. The eight way should be made to be 130mm.



The prototype unit showing the detail of the board to board interconnects. Here I have used the Molex KK 0.1" system to connect the Sock8 to the main PCB.

Building the 1U Filter Core module by wiring the sockets manually

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 LWR. Pin 1 is the square solder pad.

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

UPPER

Pin	*Pad name*	*Socket Connection*	*Lug Type*
Pin 1	BANDPASS	Connect to BP	Signal lug
Pin 2	Not connected		
Pin 3	Not connected		
Pin 4	NOTCH	Connect to NOTCH	Signal lug
Pin 5	module ground	Connect to IN1 & IN2	NC lugs
Pin 6	IN1	Connect to IN1	Signal lug

LOWER

Pin	*Pad name*	*Socket Connection*	*Lug Type*
Pin 1	Panel ground	Connects to all sockets	Ground lugs via wire frame
Pin 2	LOWPASS	Connect to LP	Signal lug
Pin 3	KEY-CV	Connect to 1V/OCT	Signal lug
Pin 4	HIGHPASS	Connect to HP	Signal lug
Pin 5	module ground	Connect to 1V/OCT	NC lug
Pin 6	IN2	Connect to IN2	Signal lug
Pin 7	module ground	Connect to CV1	NC lug
Pin 8	CV1	Connect to CV1	Signal lug

2U SVF full format

I am not going into great detail with this format as the PCB has been primarily 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 ten sockets and four additional pots. You can use any pots you like, but I am rather partial to the 16mm Alpha pots sold by Banzai which have the solder lugs and not the usual PCB mounted ones you have used on the board. You may be tempted to use the larger 24mm Alpha pots, indeed, these are great pots, but the width of them may mean that the top pot will clash with your choice of mounting rail.

Your first job will be to ground the earth lugs on each socket. Do this by joining the earth lugs of each vertical row of sockets together first with stiff single core wire. Then use another piece of solid core wire, going across horizontally, to connect all four vertical wires together. Take a single insulated piece of wire back to on the PCB and connect it to pin 1 of the header LOWER. This now connects all the sockets' ground lugs to earth/pan connection on the PCB. This in turn goes, via pin 3 of the PSU header, back to the power supply.

It is also advisable to ground the NC lugs of the six input sockets too. Do this in the same way you have commoned the earth lugs. Simply connect all three sockets' NC lugs in each of the right hand columns together with two pieces of stiff wire. Then, with two more short lengths of insulated multistrand wire, connect the stiff wire pieces to pins 5 and 7 of UPPER respectively.

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

UPPER

Pin	*Pad name*	*Socket Connection*	*Lug Type*
Pin 1	BANDPASS	Connect to BP	Signal lug
Pin 2	Not connected		
Pin 3	Not connected		
Pin 4	NOTCH	Connect to NOTCH	Signal lug
Pin 5	Module ground		
Pin 6	Not connected		

LOWER

Pin	*Pad name*	*Socket Connection*	*Lug Type*
Pin 1	Panel ground	Connects to all sockets	Ground lugs via wire frame
Pin 2	LOWPASS	Connect to LP	Signal lug
Pin 3	KEY-CV	Connect to 1V/OCT	Signal lug
Pin 4	HIGHPASS	Connect to HP	Signal lug
Pin 5	module ground	See text	NC lugs
Pin 6	Not connected		
Pin 7	module ground	See text	NC lugs
Pin 8	CV1	Connect to CV1	Signal lug

All your other connections will be made via the four two way 0.1" headers that are situated on the board near the pots. These are labelled appropriately to help you connect up your module correctly. They are IN-1, IN-2, IN-3 and CV2.

Pots have three pins. Two of these pins will be connected to PCB, whilst the remaining one will be connected to the appropriate socket's signal lug.

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

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

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

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

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

Testing

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.

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. Middle A, 440Hz is a good note to use.

Moving the FREQUENCY control should produce the usual and distinctive filter effect from the low pass output. From the high pass output, you should hear the sawtooth get brighter and less smooth as the cut-off frequency is increased. The band pass output should produce a sort of wah-wah sound as the Frequency pot is moved back and forth. The notch output will be more subtle, and will appear to sound like a mild phasing effect. Turning the Resonance up will accentuate the 'electronic' nature of the sound on all four outputs.

Remove the audio input and ensure the RESONANCE mode switch is set to HIGH. Check that at maximum resonance the filter output will oscillate across the whole audio band. You'll probably need to turn up the cut-off frequency pot to above the mid-point first to actually get the SVF to start to oscillate – it does not tend to oscillate naturally at frequencies below 1kHz or so. 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.

Now click the MODE switch to NORM and you should find that the loudness of the oscillation drops somewhat. It will also probably cease to oscillate at all at frequencies below 800Hz or so.

Listening to the band pass output with the sawtooth input still connected, patch a LFO or EG output to the CV inputs. The 1V/octave input should produce large sweeps of cut-off. Check also that with the LFO or EG connected to the CV1 input, the CV1 pot allows you to control the depth of the sweep. Fully clockwise the CV1 input should produce very deep sweeps.

Notice that the minimum sweep depth should occur with the CV1 pot at its mid point. Use a sawtooth waveform on your LFO, and see if the CV1 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.

Check that both the audio inputs behave identically.

If all this happens, the chances are that you have a working module. Now before you start to use it in your modular you will need to calibrate it. The full calibration procedure 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. 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 those nice people on the Synth-diy and Analogue Heaven mailing lists and those at Muffwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK