Oakley Sound Systems

5U Oakley Modular Series

TSL - Transistor Superladder Voltage Controlled Filter

PCB Issue 3 & 3.1

Builder's Guide

V3.15

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the issue 3 and 3.1 Transistor Superladder 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, a list of the various interconnections and a 'how it works' section.

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

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

Please note that this is not the Builder's Guide for the original issue 3 Superladder. The SL-3 and the TSL-3 are two very different projects.

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 Transistor Superladder PCB
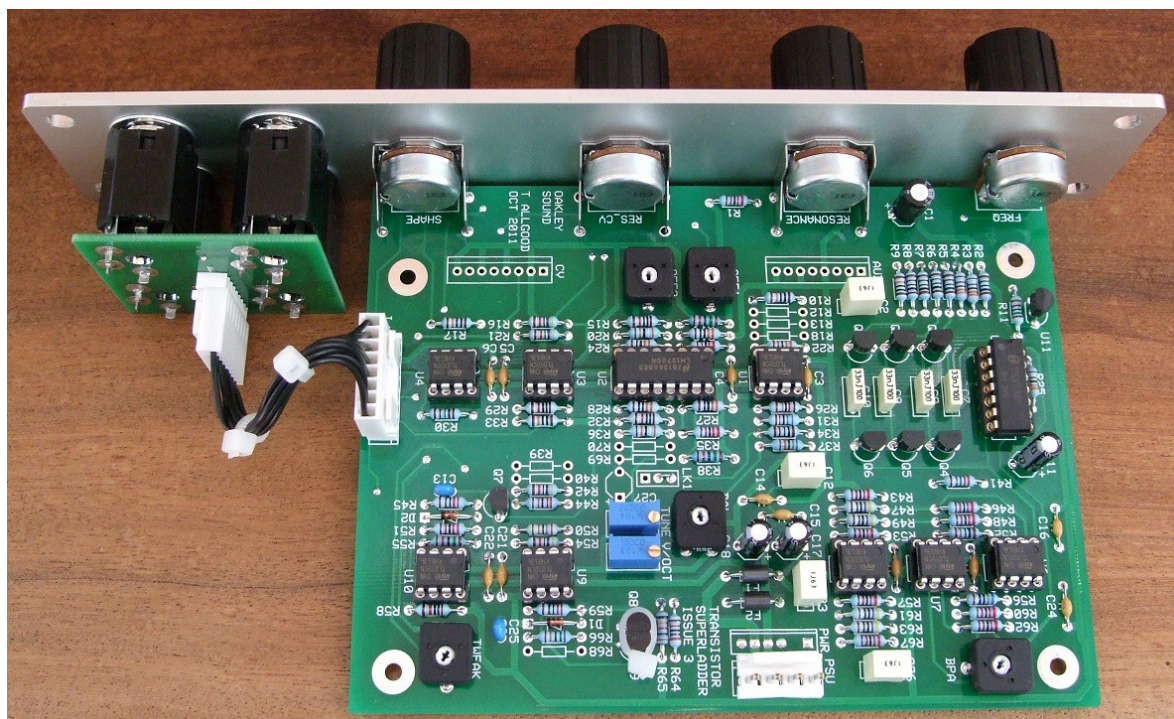
The earlier issue 3 Transistor Superladder behind a natural finish Schaeffer panel. Note the optional Sock4 board which makes wiring the four sockets simple.

The printed circuit board (PCB) is a two layer design with copper tracks on the top and bottom sides of the glass fibre board. The size of the issue 3.1 board is 102 mm by 143 mm.

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" and is the same as the vertical spacing on the MOTM modular synthesiser. The pots are; cut off frequency, resonance, resonance CV depth and filter 'shape'.

The design requires plus and minus 15V supplies. The power supply should be adequately regulated. The current consumption is about 40mA for each 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.

The PCB has four mounting holes for M3 bolts, one near each corner. These are not required if you are using the pot brackets.

Issue 3.1 is the same circuit as issue 3 but the PCB is slightly less deep and the single turn trimmers are now the more popular Bourns 3386F types.

Buffered or Unbuffered Option

The issue 3 and 3.1 boards allow the choice to send the feedback signal through a Darlington buffer (or voltage follower) circuit before re-entering the ladder at the bottom. The PCB can be built to either use this buffer and drive the ladder with a voltage, or not use the buffer and to drive the ladder directly via the OTA's current output.

The sonic differences are fairly minimal and I'll leave it up to you as to which one you should build but you may wish to try both and see which one you like the best. I prefer the sound of the unbuffered one.

It is not easily possible to change modes on the fly so the board will probably built in one mode or the other. The mode is selected by placement of a wire link in LK1 and a choice of four resistors and one capacitor.

For the recommended unbuffered version:

A wire link is fitted to pins 2 and 3 of LK1

R38, R69, R70 and C27 are not fitted.

R35 is 82K

For the buffered version:

A wire link is fitted to pins 1 and 2 of LK1

R38, R69, R70 and C27 are fitted.

R35 is 22K

Transistor Superladder issue 3 and 3.1 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.

The 'Buffered' and 'Unbuffered' Option

A small wire link should be fitted in LK1.

Link pin 1, the square pad, with the middle pin, for the buffered version.

Link pin 3, the end round pad, with the middle pin, for the unbuffered version.

Resistors

1% 0.25W metal film types are to be recommended. However, 5% ones can be used in all places except those marked '1%' which have to be 1% or better.

Components marked with a * are only to be fitted if building the full 3U version. For those building the 1U filter core version you should not fit these parts.

Value	Components
330R	R27, R28, R24, R19
680R	R56
1K	R3, R4, R5, R6, R7, R8, R30
1K2	R11, R52
1K5	R2, R25, R41
2K2	R59, R45
4K7 1%	R61, R63, R47, R49
7K5	R55
10K	R15, R23, R29, R32, R34, R51, R58
15K	R14
22K 1%	R1, R46, R48, R43, R53, R60, R62, R57, R67
27K	R64
33K	R22
39K	R9
47K 1%	R16, R21, R54

68K	R31, R42
100K	R10, R33, R17, R68*, R39*, R18*, R12*, R13*
100K 1%	R40*
120K	R26
220K	R44
270K	R50
390K	R36, R20
750K 1%	R37, R66

1K +3000ppm/K PTC R65

And if building the unbuffered option:

82K	R35

And if building the buffered option:

10K	R69
15K	R38
22K	R35
33K	R70

Capacitors

100nF multilayer ceramic	C3, C4, C5, C6, C16, C19, C20, C21, C22, C24, C14 (issue 3 only), C15 (issue 3 only)
100pF C0G 2.5mm ceramic	C13, C25
33nF, 100V, 5% polyester	C7, C8, C9, C10
1uF, 63V polyester	C23, C26, C12
2u2, 50V polyester	C2
2u2, 63V electrolytic	C11, C17, C18
220uF, 16V electrolytic	C1

And if building the buffered option:

22uF, 35V electrolytic	C27

Discrete Semiconductors

BC549 NPN transistor   Q1, Q2, Q3, Q4, Q5, Q6, Q8, Q9
BC559 PNP transistor   Q7
1N4148 silicon diode   D1, D2

Integrated Circuits

TL072CN dual FET op-amp   U1, U3, U6, U7, U8, U9, U10
OPA2134 dual FET op-amp   U4
LM13700N dual OTA   U2
THAT 300P NPN array   U5
LM4040DIZ-10.0 10V reference   U11

Notes:

For U11 the LM4040CIZ-10.0 is also suitable.

For U2 the LM13600 can also be used if you are building the 'unbuffered' option. However, the LM13600 cannot be used in place of an LM13700 if you are building the buffered version.

Trimmers (preset) resistors

Multiturn types are Bourns 3296W or equivalent.

10K cermet multiturn   V/OCT
100K cermet multiturn   TUNE

Single turns are Bourns 3386F for issue 3.1, or Bourns 3386G for issue 3.

100K single turn   BAL, OFF2, OFF1
1K single turn   BPA
10K single turn   TWEAK

Potentiometers (Pots)

All pots Alpha 16mm PCB mounted types

47K or 50K linear   FREQUENCY, RESONANCE, RES_CV, SHAPE

Three 16mm pot brackets.

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

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

Offboard Pots (3U format only)

47K Log   IN1, IN2, IN3
47K Linear   KEY SCALING, EXP CV1, EXP CV2, LIN CV

**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.

Other Notes
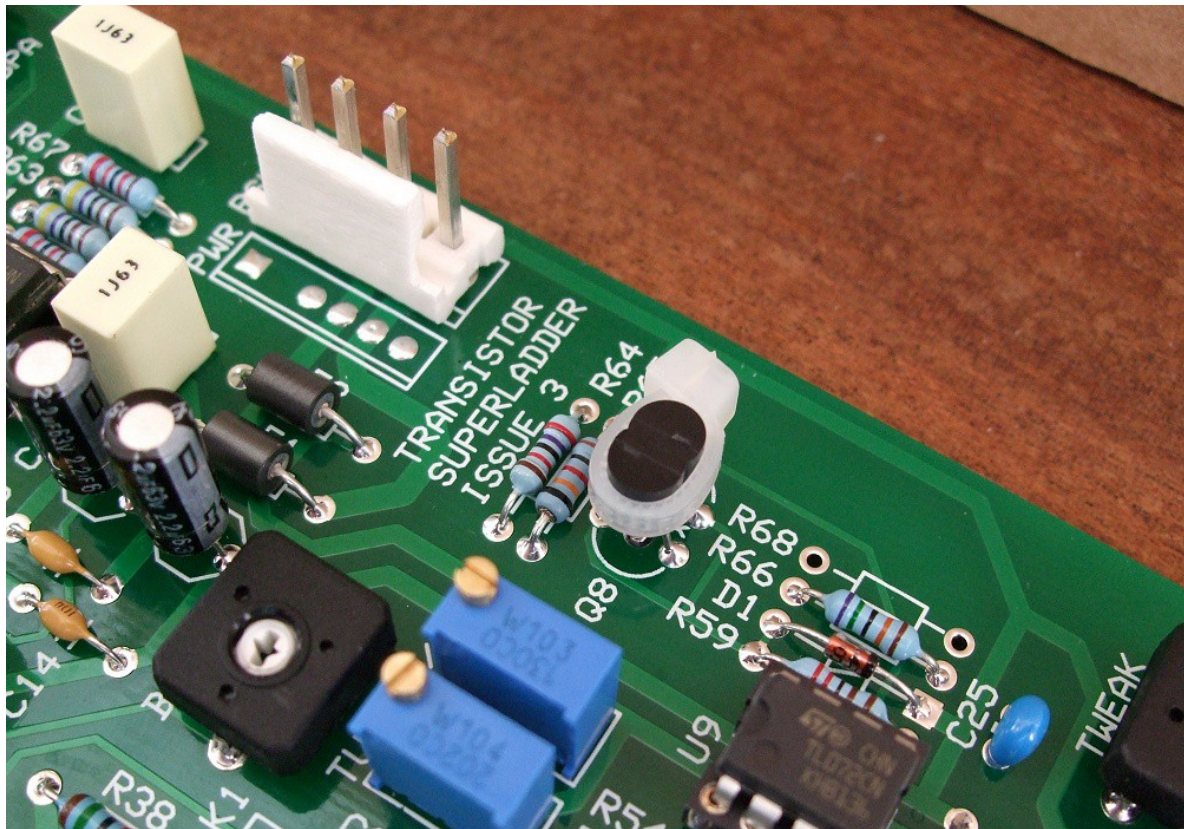
Q8 and Q9 are thermally locked together with cable tie.

Q8 and Q9 require special mention. These should be placed into position on the board, but before soldering you should wrap a small cable-tie around the bodies of the devices. Tighten the cable-tie carefully so that the flat face of each transistor is touching the other. Snip of any excess plastic from the cable tie. This will hold them together to ensure good thermal contact. There is no need for any thermal compound between them, but if you have some, you can add a small bit to ensure good thermal transfer. Now solder all six leads as you would normally.

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 ground (0V)	2
Socket ground	3
-15V	4

Pin 1 of the I/O header is connected to pin 3 of the PSU header and 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.

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 ground (0V)	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, 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.

* Issue 3.1 boards now 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 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 prototype TSL-3 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

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	RES_CV	Connect to RESONANCE	Signal lug
Pin 3	GND	Connect to RESONANCE	NC lug
Pin 4	EXP_CV1	Connect to FREQUENCY	Signal lug
Pin 5	GND	Connect to FREQUENCY	NC lug
Pin 6	OUTPUT	Connect to OUTPUT	Signal lug
Pin 7	GND	Connect to INPUT	NC lug
Pin 8	INPUT	Connect to INPUT	Signal lug

3U Superladder full format

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 3U format contains ten sockets and seven additional pots. As with the 1U module, you need to ground the sockets' earth lugs. Do this by joining the earth lugs together first with stiff single core wire. Since the suggested 3U layout has the sockets arranged in two groups of four and a middle group of two, I would recommend that you use five lengths of stiff wire; each one placed horizontally across each pair of adjacent sockets. Then, with four lengths of thin insulated multistrand wire, connect each piece of solid wire together. Then with one more 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 provided for the 1U version, but you will need to use five of these pads too. These will be pads 3, 5 and 7 which connect to the module ground. Also pad 6 which is the main audio output, and pad 2 which connects to the Resonance CV input socket. Pad 1 is depicted by the square solder pad.

All your other connections will be made via the two 0.1" headers to the left of the board near the pots. These are labelled AUD and CV, the former handling all the audio inputs, and the latter the control voltage inputs.

On the headers, each odd numbered pin is the signal and all the even numbered pins are the module ground, or 0V. The schematics show this clearly, with the AUD header shown on page 1 on the far left, and the CV header shown in the middle of page 2. Note that the audio input header actually has four inputs available, three which will come into the PCB via pots, but the fourth coming direct from the fourth input socket's signal lug.

The pots have three pins. The middle pin, the wiper, will carry the signal to the appropriate header pin on the PCB. The audio pots' wires will attach to the underside of the board at the AUD header, and thus be soldered from the topside of the board. The CV pots' wires will attach to the topside at the CV header and be soldered on the underside. The schematic will show you which pad should connect to which pot's wiper lug.

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 ground connection should go to the right hand pin. Take a wire from this pin to the pad on the PCB below the one that the wiper connects to.

Examples:

Pin 1 of AUD goes to the wiper of the audio input LEVEL 1 pot. Pin 2 of AUD will go to the right hand side pin of that pot.

Pin 1 of CV goes to the wiper of the KEY-SCALE pot. Pin 2 of CV will go to the right hand side pin of that pot.

Wire up all the pots like this until all the pots are connected.

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. That is IN 1 goes to the signal lug on the socket labelled IN 1. Remember too, that the socket labelled IN 4 will go direct to pin 7 of the AUD header and not need an associated pot.

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

Circuit Description

The audio inputs are fed into the filter via a simple voltage mixer based around U1 (pins 1,2,3). Four audio inputs are given on the 3U version, but for the standard 'filter core' modular panel we are only using one. The input levels are expected to be generally around a maximum of 10Vp-p each. The gain of the input amplifier is such that any three inputs on the 3U version can be ran at full volume without any clipping effects being caused by the input op-amp itself. However, the filter core circuitry will overdrive if the audio input level pots are set to over 2 o'clock with a 10V p-p signal in place. This distortion is normally described as pleasant, and works wonders on most VCO waveforms. With more complex program material it is best probably not to run it too hard. You won't damage the module but it will probably sound pretty horrible. Too little input signal will produce very little waveform distortion, but the relative noise level (mainly hiss) will be higher.

The summed output is not fed directly into the filter core like in the Oakley Multiladder, early issues of the Superladder or most other ladder filters. In the later Transistor Superladders we feed the output of the summing amplifier to the first VCA section. The VCA is based around one half of a dual OTA, U2 and the other half of op-amp U1. This VCA is controlled by the resonance pot or the relevant CV input. As the resonance, sometimes called 'Q' or emphasis, is turned up, the VCA turns on letting more audio signal into the filter core. This compensates for the drop in 'passband' gain normally associated with ladder filters at high resonance. So what we are doing is actually automatically turning up the input level to the filter as the resonance increases.

This VCA circuit is actually a 'leaky' one. R31 constantly bleeds audio into the filter even when the VCA is off. Thus audio is still getting to the filter at low values of resonance. In the old issues of the Superladder I did this Q-compensation at the output sections of the filter; in other words, turning up the volume at the final stage of the module. Although this worked very well, it did tend to make the filter noisier at high values of Q. The newer method is much quieter and leads to more consistent overdrive characteristics too.

C2 and R9 pass the audio signal into the ladder filter itself. The filter is based around the traditional ladder as designed originally by Dr. Moog in the 1960s. I have used an THAT300P matched transistor array for the top and bottom pairs in the ladder. This minimises control current breakthrough to a small value. Current breakthrough manifests itself as a copy of the modulating signals on the output. Generally, this is not a good thing. The trimmer 'BAL' biases the base of U6, via R37, by a small amount to even out any differences within the ladder. This minimises breakthrough still further.

Four CV inputs control the filter cut-off frequency via an exponential convertor, U9 and a pair of NPN transistors, Q8 and 9. KEY-CV, EXP_CV1 and EXP_CV2 all provide exponential CV responses. LIN_CV directly controls the current in the filter ladder, and as such is a sort of linear control input. In the 1U 'Filter core' module, we will only be using the EXP_CV1 input which is roughly set to a sensitivity 0.7V/octave. All the other frequency CV inputs are left unused. In the 3U modular set up we will be using four additional pots, mounted externally to the board, to control the level of the KEY-CV (for keyboard scaling), both EXP_CVs (general CV inputs) and LIN_CV inputs (another CV input).

The exponential convertor is temperature compensated. R65 is the positive temperature coefficient resistor providing an approximate cancellation of the exponential converter's inherent temperature coefficient. The Superladder uses a +3000ppm/K 1K 'positive temp co' and a 27K resistor for R64 to create the necessary temperature compensated -18mV/octave voltage input to the base of the expo converter's input NPN transistor. It should be noted that although the transistors are not hand matched, modern transistors do show remarkable similarities if bought from the same manufacturer these days. It is my opinion that perfect matching is not necessary for a filter's exponential convertor. However, to minimise temperature drift I do recommend that the transistors are placed and held together with a cable tie.

The V/OCT trimmer provides adjustment of the exponential scaling factor. This will allow the VCF to be set up to track a VCO's output if set to the standard 1V/octave. In the 1U version of the Superladder we have little need for this trimmer as our only CV input is to be set at approximately 0.7V/octave. The 3U version, however, has a dedicated KEY CV input socket, and associated scaling pot, which is designed to be used with the 1V/octave keyboard CV coming from a midi-CV convertor or analogue controller. In reality exact 1V/octave sensitivity is not really that critical unless you use the Superladder as an oscillator.

It must be said at this point that I have deliberately restricted the maximum resonance at very low frequencies, thus the VCF will not oscillate across the whole audio band. This has the wonderful effect of creating a 'punchier' bass. Decreasing the size of C12 to 100nF will enhance this effect even more. Although at this value the filter will not oscillate much below 1kHz.

Resonance is controlled by the 'RESONANCE' pot and the RES_CV input. The RES_CV input is fed into a reversible attenuator circuit. This circuit allows the gain to be varied from -1 to +1. In other words, a CV input may be controlled in level from inverting, to off, to straight through. Its easy to see how it works. One end of the pot is driven from a voltage follower, a simple circuit that presents at its output a copy of what it 'sees' at its input. The other end of the pot is connected to an inverting amplifier. This simple circuit inverts the input voltage, eg, 1V becomes -1V and -2V becomes 2V. The wiper on RES_CV pot can be moved from inverting at one end, to non inverting at the other. In any position in between, the voltages from each op-amp are combined in different degrees. At the centre the wiper receives signals from both op-amps in equal proportions and the two cancel out.

Both the voltage on the Resonance pot and that from the RES_CV input are fed into voltage followers and then onto the linear voltage to current convertor, U10a and Q7.

The current produced by U10a/Q7 actually drives two OTAs. One, we have already discussed, increases the signal level being fed into the ladder core. The other, U2b, increases the feedback between the top of the ladder and the inverting input of the ladder's base transistors. In the original Moog ladder this was achieved by a simple variable resistor. The more feedback you apply the greater the resonance.

The trimmer, TWEAK, allows you to adjust the maximum level of resonance you require and to compensate for any differences in transconductance of the dual OTA, U2.

Both halves of the dual OTA have a offset adjust trimmer. Offset manifests itself as a copy of the resonance control voltage on the outputs of the filter. Adjusting OFF1 & OFF2 will minimise this sufficiently as not to cause a problem.

The filter outputs are obtained by 'looking' at the differential voltage across the relevant filter capacitors in the ladder. So that both four and one pole outputs are available, two differential amplifiers (called 'diff-amps' for short) are used.

The two diff-amps are identical, based around the classic three-op-amp implementation. They are all DC coupled, and rely on close matching to remove any DC offset. A differential amplifier is a device that makes larger the voltage difference between two points. In our case, the voltage across each filter capacitor. The gain of the differential amplifiers is set so that the overall gain of the whole filter module is around unity with the cut-off frequency at its highest value.

The variable 'shape' output is obtained by mixing the four pole output with the inverted one pole output That is, we need to subtract one output from the other. Rather, than build a subtractor, I inverted the one pole output by simply wiring the differential amp in the opposite fashion to the other one. The two output signals from each diff amp are then simply added together by the action of the 'SHAPE' (or VARIABLE SLOPE on the 3U module) pot. The combined voltages are then buffered by unity gain follower circuit, U4b (pins 5,6,7).

The BPA trimmer, 'band pass adjust', is provided to compensate for any variations in the gain of each diff amp. This is set so that the band pass response is heard when the SHAPE pot is exactly central.

Note that the variable shape output's phase changes depending on the position of the SHAPE pot. This is not considered a problem, but it should be noted all the same.

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 and the polarity of the electrolytic capacitors.

Assuming everything is good 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. Set the SHAPE (or VARIABLE SLOPE on the 3U module) pot to the far right to put the filter into four pole low pass mode. Moving the FREQUENCY control should produce the usual and distinctive filter effect from the 4 pole low pass output. Now turn the SHAPE (or VARIABLE SLOPE) pot to the far left, you should hear the same sort of filter sweep sound, but with a far brighter edge to the sound. This is the one pole (or -6dB/octave) low pass mode.

Check that rotating the SHAPE (or VARIABLE SLOPE on the 3U module) pot you can get the sound to change from the one pole to four pole sound at either extremes of the pot's movements. In the middle you should get a two pole band pass filter. To test the BP response, sweep the filter frequency up and down. At the high frequencies, the output should just sizzle, while at the lower end the bass should be very smooth.

Turning the Resonance up will accentuate the 'electronic' nature of the sound on all three outputs. 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.

Listening to the four pole low pass output with the sawtooth input still connected, patch a LFO or EG output to the FREQUENCY input. The 0.7V/octave input sensitivity of the FREQUENCY 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.

If you wish the FREQUENCY input to conform to 1V/octave then R42 can be changed to 100K.

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

Final Comments

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

If you are in the UK and 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 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 Modwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK
© March 2012 – updated January 2024

No part of this document may be copied by whatever means without my permission.