Oakley Sound Systems

5U Filter Core Series

TSL - Transistor Superladder

Voltage Controlled Filter

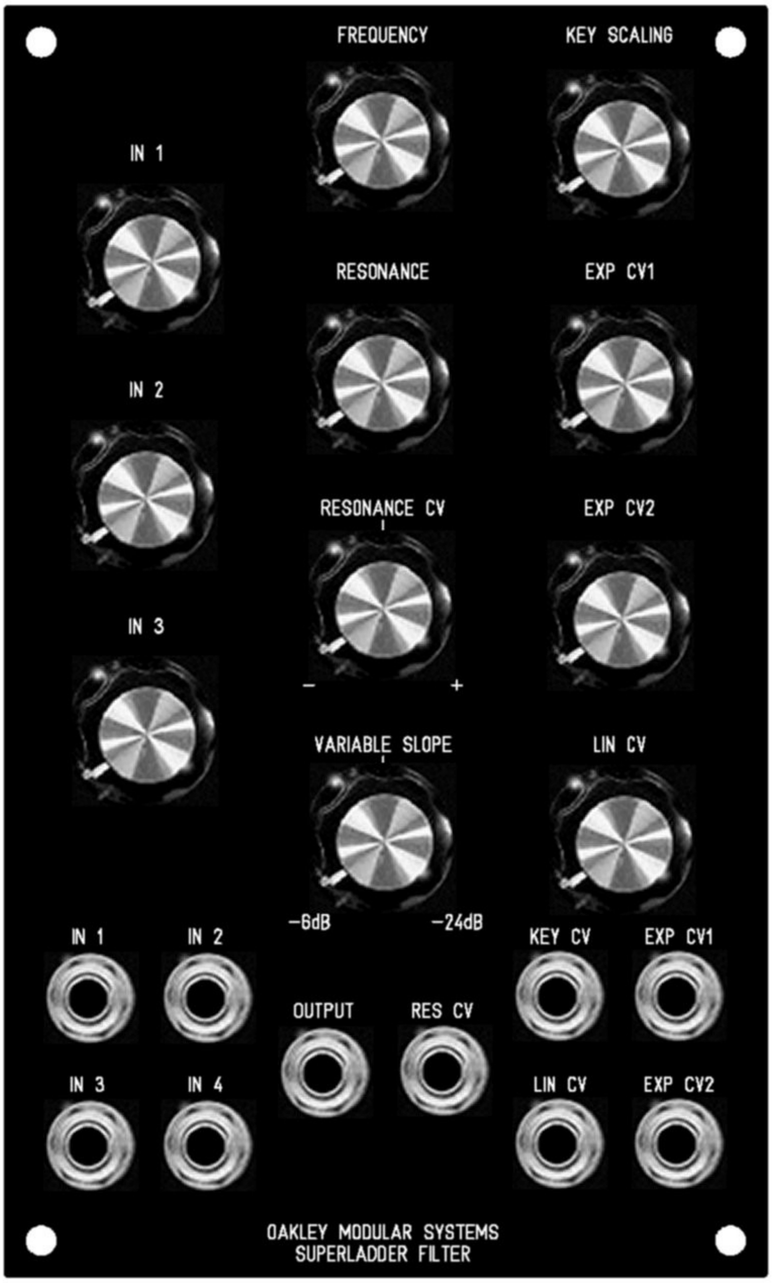
PCB issue 2

Project Builder's Guide

V2.0.5

Tony Allgood B.Eng
Oakley Sound Systems
CARLISLE
United Kingdom


FREQUENCY
RESONANCE
RESONANCE CV
–
+
SHAPE
INPUT
FREQUENCY
OUTPUT
RESONANCE
OAKLEY
LADDER VCF



FREQUENCY
KEY SCALING
IN 1
RESONANCE
EXP CV1
IN 2
RESONANCE CV
EXP CV2
IN 3
–
+
VARIABLE SLOPE
LIN CV
–6dB
–24dB
IN 1
IN 2
KEY CV
EXP CV1
OUTPUT
RES CV
IN 3
IN 4
LIN CV
EXP CV2
OAKLEY MODULAR SYSTEMS
SUPERLADDER FILTER

Introduction

This is the Project Builder's Guide for the issue 2 Transistor Superladder module which uses the issue 2 Superladder Filter Core PCB from Oakley Sound.

This document contains a basic summary of its operation, a little about the history of the module, a how it works section, a full parts list for the components needed to populate the board and some basic testing methods.

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 Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

The Transistor Superladder Module

The Transistor Superladder, or TSL, is a voltage controlled filter (VCF) module. Cut-off frequency and resonance are both voltage controlled. The basic filter topology is the old classic, the transistor ladder filter. This was originally postulated by the late Dr Robert Moog in the 1960's and has found its way into many classic synthesisers. However, the Oakley Superladder contains some interesting twists in the implementation of the good doctor's design.

The module has a variable shape output. This is controlled by the 'Variable Slope' pot. You can smoothly slide from one pole low pass, through band pass in the middle, and on to the classic four pole low pass.

The output level is 'Q-compensated'. This means that the output level will not drop significantly when you turn up the resonance pot. The traditional Moog ladder 'suffers' greatly from this. This design uses two VCAs to achieve voltage controlled resonance without the hassle.

The Resonance CV depth pot has an inverting feature. Turning the pot fully anti clockwise produces an inverted sweep. The middle position produces no sweep, and fully clockwise produces the usual positive going sweep.

The Filter Core Idea

The Filter Core idea has come from the fact that many of our customers were buying different filter types, eg. they may have a 700S clone, a Moog ladder filter and an SVF. Each filter type gives a different sound so its worthwhile having a few in your modular set up. However, each filter module also has its own input mixer for audio and an input mixer for CVs. This adds to panel real estate and soon your modular is filling up very quickly. While this does look very impressive, it does mean that, in many patches, you have a lot of redundant electronics in your modular.

Step forward the 'filter core'. This is quite simply a 1U module that contains only the filter and a few important front panel pots. All the audio and CV mixing is done externally with a dedicated mixer module, like the Multimix. The good thing about this is that any unused filter module is only 'wasting' 1U of panel space. So you can afford to have many different flavours of filter without the additional cost and panel space of mixers.

However, as with all things, there are disadvantages too. The lack of inbuilt mixers mean that you will need to get more dedicated mixer modules. But remember that these relatively cheap mixer modules can be used for any mixing or level controlling within your modular. Thus, you have more flexibility, at the expense of a little more patching.

The great thing about the new Oakley Filter Core modules is that they will all be designed so that they can still be used in the full format design. All the Filter Core modules will have input summing amplifiers built onto the PCB. You won't be using these circuits in the 1U format, but they are there if you want to go for the larger 2U or 3U designs.

The Filter Core panel design is a lot easier to make of course. It only has four sockets and all the wiring is done by using some big solder pads at the bottom of the PCB. For the 3U format, you will need to use these pads and some additional 0.1" headers which are placed near the pots. In a possible future addition to our range, we may provide special pot PCBs to directly attach to these headers to make building the 3U module a lot easier. These pot PCBs have proved very successful in the VCO and other modules where we use them.

Issue 2 of the Superladder Filter Core module is the seventh issue of the Oakley Superladder. The sixth issue being the first of the filter core modules.

The Superladder Legacy

The Superladder design was spawned by a project called the Walshbank. This was a complete analogue monophonic synthesiser that incorporated a voltage controlled digital oscillator that used an additive process to generate complex waveforms. The synthesiser also featured a voltage controlled ADSR, a flexible analogue VCF and a fairly traditional VCA. The VCO was a very big project on its own, so I decided to separate the VCF out and make it into a PCB project that people could build themselves. This was the first Superladder and it was released in late 1999.

The Superladder did various things that had not been done before to my knowledge at the time. Firstly, it sported a one pole output, a bandpass output and a four pole output. Secondly it could be built into either a diode ladder (like a TB303) or a transistor ladder (like a Moog). Thirdly, it incorporated a Q compensation circuit that kept the volume of the output fairly constant as the resonance was changed. Of course, as time passed I realised that I had not been the first to use these ideas, but I still think it was the first time that all three had been available in one design.

The issue 1 boards were a relative success - I managed to cover my costs, which was all that I could ask for. However, there were some mechanical problems with fitting the board into a MOTM panel and rack rails, so I decided to make an issue 2. This used a similar circuit, but made it easier to wire up and made it more MOTM compatible.

Both the issue 1 and 2 boards used the 2SC1583 matched pair in the differential amplifiers and the exponential converters. However, these were becoming harder to source so I needed to make a new issue board. Issue 3 used the more common CA3046 NPN array and op-amps in the differential amplifiers. It also featured a change in the way the Q-compensation worked; the level compensation circuitry was now fitted at the input to the filter core as opposed to after it. This was a big change in circuitry and the issue 3 boards sounded slightly different. I would say that the new board sounded better, it was cleaner and less noisy. However, I am waiting for the day when issue 1 Oakley boards are talked about in hushed tones as being the best.

All the issues so far used Omeg pots. These UK made pots were quite popular at the time and had a good reputation. However, my own experience was proving to be less than perfect. In particular, I had one batch that was very poor, with many units having to be returned. At great expense I began the slow roll out of a series of new designs that used the more expensive Spectrol 248 pots. These superior pots would prove to give excellent results and, so far, have shown good longevity.

Issue 4 Superladder PCBs and modules used Spectrol pots throughout. There were also some subtle differences in circuitry, but not much, and mostly to allow for easier wiring.

Issue 5 was being designed as I decided to pull out of the modular market in 2005. It was laid out on my CAD system and only one board was ever made. It however sowed the seeds for the current filter-core board, and its use of the THAT300P NPN array proved very successful. The premise of this design was for a RoHS (lead free) compliant Superladder. This meant that it could no longer use the non compliant and now obsolete CA3080 and CA3046 devices. The THAT300P fitted the job of the core array perfectly. In fact, its actually better than the CA3046 it replaced.

The CA3080's job in the SL-4 board was to control the depth of one of the incoming CVs, usually the envelope generator for velocity control. However, it was easier to remove the function from the module, rather than trying to find a RoHS alternative to the CA3080. The CV controlled EG depth was really a hangover from the Walshbank days, and our ADSR already had a CV controlled level built into it. This partial loss of functionality started me thinking about the way these rather extraneous functions are handled, and so the filter core idea was formed.

As stated earlier, this new board was effectively the sixth reincarnation of the Superladder. However, the board was labelled as the issue 1 Ladder Filter of the new Filter Core series. It was mostly the same as the short lived issue 5, but with one important change; there was no TB303 filter mode. The filter topology is solely the Moog ladder and no diode ladder mode can be made with the stock PCB. The reason for this is that since issue 3 the changes in the differential amplifiers and the feedback paths have meant that the newer boards didn't sound much like the filter in the 303. True, the basic characteristic was the same, but it wasn't close enough for me. I think the TB303 deserves a more faithful clone of its filter and this should have its own module and PCB.

The other change is the loss of the separate one and four pole outputs. From feedback from customers I now realise that most people did not use the separate outputs simultaneously and use only the variable output. Thus, the new Superladder even in its 3U format has only one output. This allows us to save on sockets and gives a little extra room on the front panel for more description around the shape pot. It also gives the module a more interesting socket field and makes it stand out more in the sea of knobs and sockets that can be the Oakley/MOTM modular.

Issue 2 of the Superladder Filter Core PCB is identical to the issue 1 but for one powerful addition. The board now sports a location for a Synthesizers.com power header which can be used in place of the usual MOTM/Oakley one.

Power supply requirements

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.

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 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 convertor’s inherent temperature coefficient. The Superladder uses a +3000ppm/K 1K ‘temp co’ and a 27K resistor for R64 to create the necessary temperature compensated -18mV/octave voltage input to the base of the expo convertor’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.

D1 and D2 behave as non linear pass elements. Together they only allow voltages above 0.6V and below -0.6V through. This creates a dead band around the centre of the RES_CV pot's movement, making it easy to set it to zero without it being exactly in the centre. However, the dead band only works properly if the CV is a unipolar signal, ie. its either always negative or always positive in value. This would be like an output from an ADSR or a CC from a midi-CV convertor, which are both always positive. AC signals, like an audio signal or an LFO output, will get slightly distorted by the dead band circuitry. But in practice this shouldn't be any problem.

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. Rather, than build a subtractor, I inverted the one pole output by simply wiring the diff amp in an 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.

Parts List

The components are grouped into values, the order of the component names is of no particular consequence. Please read the above section for more details about the parts used in this module.

A quick note on European part descriptions. R is shorthand for ohm. K is shorthand for kilo-ohm. For capacitors: 1uF = 1000nF. 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 are 1/4W 5% carbon or better, unless stated.

For the 1U 'Filter Core' version you do not need to fit any resistors marked with a *.

150R	R11
330R	R28, R27, R19, R24
680R	R56
1K	R34, R2, R30, R5, R6, R8, R7, R52
2K2	R45, R59, R41
3K3	R3, R4, R25
4K7 1% metal film	R49, R47, R63, R61
10K	R29, R32, R15
15K	R14, R38, R55
22K	R35, R58, R1, R23, R51
22K 1% metal film	R46, R48, R60, R62, R43, R53, R67, R57
27K	R64
33K	R22
68K	R31
100K	R9, R17, R10, R33, R16, R21, R18*, R12*, R13*, R68*
120K	R26
180K	R54
270K	R42
390K	R36, R20, R40*, R39*
680K	R37
750K	R44, R66
1M	R50

Capacitors

33nF, 100V polyester	C7, C8, C9, C10
100nF multilayer ceramic	C14, C15, C16, C24, C5, C21, C22, C6, C20, C19, C4, C3
1nF, 100V polyester	C13, C25
1uF, 63V polyester	C23, C26, C12, C2
2u2, 63V electrolytic	C17, C18
22uF, 25V electrolytic	C11

220uF, 16V electrolytic	C1

Discrete Semiconductors

BC549 NPN transistor	Q1, Q2, Q3, Q4, Q5, Q6, Q8, Q9
BC560 PNP transistor	Q7
1N4148 silicon diode	D1, D2, D3, D4

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.

Integrated Circuits

TL072CN dual fet op-amp	U1, U3, U4, U6, U7, U8, U9, U10
LM13700N dual OTA	U2
THAT300P NPN array	U5

Trimmers (preset) resistors

50K cermet multiturn	V/OCT
100K cermet multiturn	TUNE
100K horizontal carbon	BAL, OFF2, OFF1
1K horizontal carbon	BPA
22K horizontal carbon	TWEAK

Onboard Pots

50KA Lin	FREQ, SHAPE, RESONANCE, RES_CV

Offboard Pots (3U format only)

50K Log	IN1, IN2, IN3
50K Linear	KEY SCALING, EXP CV1, EXP CV2, LIN CV

Miscellaneous

Leaded Ferrite beads	F1, 2
1K PTC +3000ppm/K	R65

You may well want to use sockets for the ICs. I would recommend low profile turned pin types as these are the most reliable. You need seven 8-pin DIL, one 14-pin DIL and one 16-pin DIL socket.

Mounting the PCB mounted Pots

If you are building the 1U format Filter Core, then you only need four pots. These are mounted onto the PCB directly, and are held in place by our specially made pot brackets.

If you are building the 3U format full Superladder module, then you will need a further seven pots. These are mounted on the front panel directly and are wired to the PCB and sockets with flying leads.

The first thing to do is to check your pot values. Spectrol do not make it that easy to spot pot values. If you have been supplied with Spectrol pots your pot kit should contain:

Value	**Marked as**	**Quantity**	**Location**	**Format**
50K linear	M248 503 M	4 off	PCB	1U Filter core only
50K linear	M248 503 M	4 off	CV inputs	3U Superladder
50K log	M248 J 503	3 off	Audio inputs	3U Superladder

If you have bought TT P260P pots then you should have the following:

Value	**Marked as**	**Quantity**	**Location**	**Format**
50K linear	B50K	4 off	Main PCB	1U Filter core only
50K linear	B50K	4 off	CV inputs	3U Superladder
50K log	A50K	3 off	Audio inputs	3U Superladder

Note that the pot shafts of the pots will not need cutting to size. They are already at the correct length. However, if you are using the TT pots you will need to trim off the little locating lug near the bush of the pot. Simply snap this off with a pair or wire cutters or fine nosed pliers.

Fit the four pot brackets to the four 50K linear pots by the nuts supplied with the pots. You should have two nuts and one washer per pot. Fit only one nut at this stage to hold the pot to the pot bracket. Make sure the pot sits more or less centrally in the pot bracket with legs pointing downwards. Tighten the nut up carefully being careful not to dislodge the pot position. I use a small pair of pliers to tighten the nut. Do not over tighten.

Now, doing one pot at a time, fit each pot and bracket into the appropriate holes in the PCB. Solder two of the pins attached to the pot bracket. Leave the other two pins and the three pins of the pot itself. Now check if the pot and bracket is lying true. That is, all four pins are through the board, and the bracket should be flat against the board's surface. If it is not, simply reheat one of the bracket's soldered pads to allow you to move the pot into the correct position. Don't leave your iron in contact with the pad for too long, this will lift the pad and the bracket will get hot. When you are happy with the location, you can solder the other two pins of the bracket and then the pot's pins. Do this for both pots and snip off any excess wire from the pot's pins at this point.

You can now present the front panel up to the completed board to check that it fits. However, I usually fit the sockets before I do this, and wire up the ground tags first. Then I mount the board up proper. You need to add the washer between the panel and the nut. Again, do not over tighten and be careful not to scratch your panel.

The pots shafts of the four pots will not need cutting to size. They are already at the correct length.

The Spectrol pots are lubricated with a light clear grease. This sometimes is visible along the top of the mounting bush of the pot body. Try not to touch the grease as it consequently gets onto your panel and PCB. It can be difficult to get off, although it can be removed with a little isopropyl alcohol on cotton wool bud.

Power supply

Power connections – MOTM and Oakley

The PSU power socket is 0.156" 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

The earth/pan connection 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.

Connections

Whether you have chosen to make your module in a 1U or 3U format will determine what you do next. The 1U format is considerably easier to build and test. The 3U is more daunting, but if you take your time, you should not find it excessively difficult. I shall deal primarily with the 1U format first since this is the recommended approach.

1U Filter Core module

You have just four sockets to wire up. If you have used Switchcraft 112 sockets you will see that they have three connections. One is the earth lug or ground tag, this is indicated by a bevel in the socket's housing. The second 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) lug. The NC lug is internally connected to the signal tag when a jack plug is not inserted. This connection is automatically broken when you insert a jack.

In this module we are going to 'common' the sockets' ground lugs. This means that the sockets' earth lugs are going to be joined together. I normally do this part of the wiring without the PCB or pots in place.

Fit the four sockets onto the panel so that the bevel on the side of the socket is facing top left as you look at the rear of the panel.

The first lugs we are connecting together will be the ground or earth tags on the two upper sockets and then on the two lower ones. I use 0.91mm diameter tinned copper wire for this job. Its nice and stiff, so retains its shape. Solder a length of this solid core wire right across the two earth tags on the top row. Trim off any excess that sticks out on either end. Then do the same on the lower two. What you have now done is common each row's earth tags together, but each row is separate for now.

Fit the Superladder PCB against the front panel if you haven't done so already. Solder a piece of ordinary insulated multistrand wire to the earth lug on the socket furthest on the left on the top row. The other end of this wire needs to go to the pad on the PCB marked PN1. Now solder another piece of wire to the earth lug of the socket furthest left on the bottom row. This wire will be going to the pad PN2. Your earth tags are now commoned together.

Now its time to wire up the four signal lugs to the board. Use multistrand hook up wire to connect each socket's signal lug to the relevant pad on the PCB. Keep your wires short but not too short and use as many different colour wires as you can. There is absolutely no need to use screened cable for such short runs.

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

Socket name	*PCB pad name*
INPUT	I/P
FREQUENCY	EXP
OUTPUT	O/P
RESONANCE	RES

The final connection now needs to be made. With another small length of wire connect the GND pad to the NC lug of the FREQUENCY socket.

That's it, the Superladder Filter Core module is now ready to test and calibrate.

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 solder pads at the bottom of the board are mainly provided for the 1U version, but you will need to use four of these pads too. These will be the PN1 and PN2 pads, which connect to the panel ground. Also O/P which is the main audio output, and RES which connects to the Resonance CV input socket.

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 schematic shows this clearly, with the headers depicted on the far left hand side of the diagram in the middle. 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.

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 wire; each one placed horizontally across each pair of adjacent sockets. Then connect each solid wire back to the PN1 or PN2 pads on the PCB with thin insulated multistrand wire. Each solder pad will happily take two or three thinner pieces of wire before it is soldered. The PN1 and PN2 pads are connected to panel ground on the power socket, pin 3 on the MTA/Molex connector.

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.

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. 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 on the 3U module) 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 all this happens, the chances are that you have a working module.

Trimmers

There are quite a few trimmers on the PCB. Run through them in this order only.

TUNE: This adjusts the filter's cut-off frequency. Set this so that the filter's FREQ pot covers your chosen range. I would normally place this in the middle position for now, that is 10 turns or so, from one of the end points.

OFF1: This should be done with a scope or digital voltmeter (DVM). Monitor the output voltage of U1 pin 7 with respect to ground. Turn the resonance up full and set OFF-1 so that the voltage is as close to zero as you can get it. +/-50mV or so. Turn the resonance down, and check that it is still nearly zero volts.

OFF2: Listen to the 4 pole low pass output. Set the Resonance and Frequency pots to their mid positions. Now connect a triangle wave signal, about 500Hz at 5V p-p, into the RES CV socket and turn the RES CV pot up full. You should hear the 500Hz signal through the low pass output. Adjust OFF2 until the sound becomes minimised, you will not get rid of it completely.

BAL: Now connect the triangle wave signal to the FREQUENCY input socket. Listen to the Four Pole low pass output very carefully. Again you should hear the 500Hz breaking through slightly. Adjust BAL to minimise the breakthrough.

TWEAK: Adjust this trimmer so that the filter just breaks into oscillation when the RES pot is moved to around 80% of its maximum setting. Alter the FREQ pot over its whole range to check whether you are getting a good frequency range of oscillation. If you set TWEAK too low, then the filter may only oscillate over a very small range.

V/OCT: This adjusts the scaling (or sensitivity) of the exponential inputs. If you have the 1U 'Filter Core' version of this module there is no need to adjust this. Simply leave it in its middle position. You will be controlling scaling with your connected external mixer module. For 3U users you need to adjust this so that there is an octave jump in cut-off frequency when the KEY CV input is raised by one volt. The best way to set this is to turn the resonance full up and let the filter oscillate. Set the KEY SCALING pot to its maximum setting. Then connect the KEY CV socket to the 'keyboard CV' out of your midi-CV convertor or analogue keyboard. It is a fiddly adjustment and it takes a while to get it right. Changing the V/OCT will also change the range of the filter as well, so you may need to alter the TUNE trimmer afterwards. For most people it is NOT essential to get exactly right unless you are using your filter as a oscillator.

BPA: Listen to the BP output so set the SHAPE (or VARIABLE SLOPE on the 3U module) pot to exactly central. Turn the frequency pot up quite high, connect a lowish frequency square wave to the audio input. Adjust BPA until the signal becomes 'fizzy' and loses the lower harmonics. On a scope, you will see the waveform will become a series of spikes and lose the square wave's top and bottom levels. It's easier to do than explain it.

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 user 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 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 to all those at Muffwiggler.

Tony Allgood at Oakley Sound

Cumbria, UK