Oakley Sound Systems

Eurorack Modular Series

SVF

PCB Issue 1

Builder's Guide

V1.0

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the issue 1 Eurorack SVF voltage controlled filter module from Oakley Sound. This document contains a basic introduction to the board set, a full parts list for the components needed to populate the boards, some basic help on how to purchase parts and build the project, and a simple test procedure.

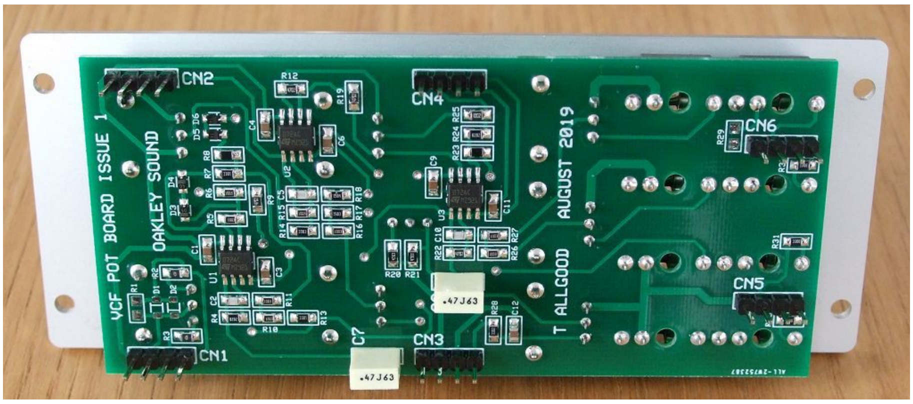
The suggested panel design.

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

For general information on how to build our modules, including circuit board population, and mounting front panel components please see our generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

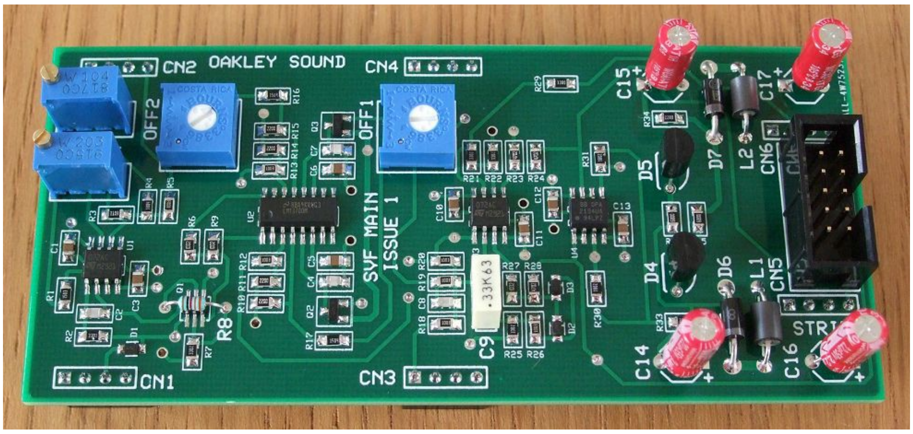
The SVF PCB Set

The electronics of the Oakley SVF filter module is built on two printed circuit boards (PCBs). The VCF Pot board holds the seven pots, one switch, input and output sockets, the soft clipping circuit, the audio mixer and the CV input mixer. All the front panel components are directly soldered to the pot board. The pot board is a double sided design meaning that there are electrically conductive copper tracks on the top and bottom surfaces of the board. The surface mount components are all soldered to the top side of the board while the pots and sockets are soldered to the underside which faces the inside surface of the front panel.



The top side of the VCF pot board. The board is already fitted to the front panel. Note that all the surface mount components are fitted to this side. The pots, switch and sockets cannot be seen as they are fitted to the underside of the board.

The second board is called the main board. This holds the exponential convertor, the filter core, the power supply conditioning, and the power inlet. The main board is a four layer design. This means that the board has layers of copper on top and bottom sides, as well as two internal copper layers. The top internal layer is solely reserved for the 0V connections – sometimes called module ground – used for the main and pot board's electronics.

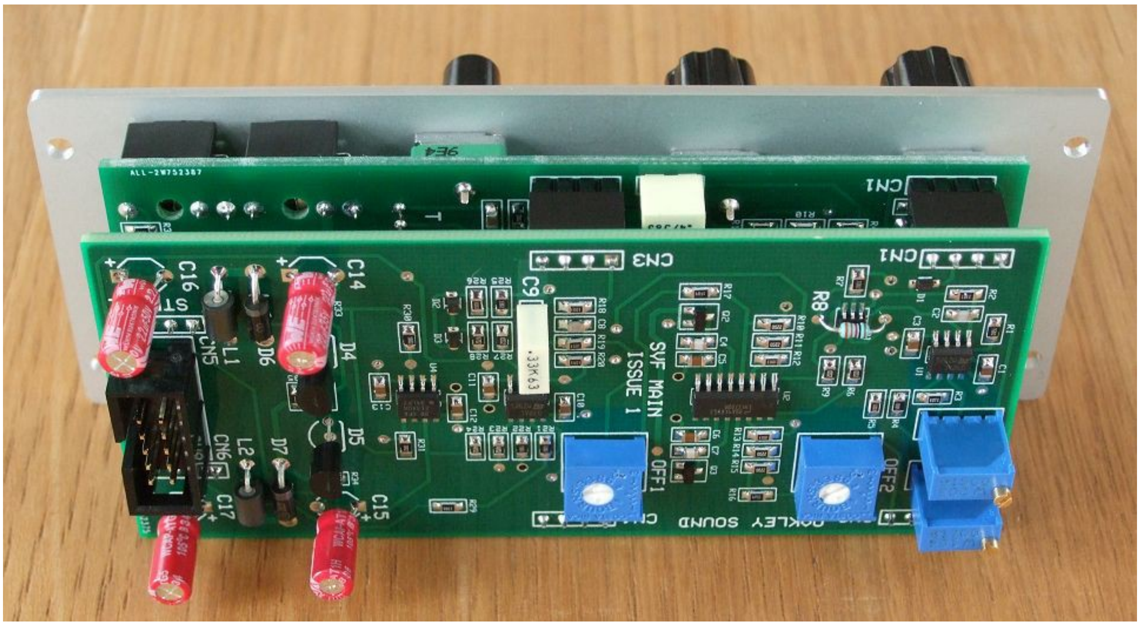


The SVF main board.

The components, again a mixture of through hole and surface mount devices, are soldered to the top of the board. The main board's components are all accessible from the rear of the module. Although this means that you need to be a little careful when handling the module, it does mean that fault finding is much easier than it would be if the main board was facing the other way and none of the components would be visible.

The surface mount components are mostly of relatively large geometries to make the build suitable even for beginners in surface mount soldering. The resistors and capacitors are all 0805 and the ICs are narrow body SOIC. The dual transistor array, Q1, is housed in a very small SOT457 package but can soldered easily enough. The larger capacitors, ferrite beads, power diodes, and all the interconnects are through hole parts.

Both circuit boards are the same size, that is, 49 mm (wide) x 107 mm (high). The boards are designed to go behind a panel that is 10HP wide.



The module comprises of two circuit boards connected together with six 4 way 0.1" SIL headers and sockets.

The grip of the multiple SIL interconnects is strong enough to hold the boards together for most uses. Should you require more rigidity then a **small** blob of silicone sealant or heat melt glue across the join between the plastic housings of CN1, CN2, CN5 and CN6 would hold everything together tightly. However, it is best not to do this until your unit has been thoroughly tested.

The design requires plus and minus 12V supplies. The power supply should be adequately regulated. The current consumption is around +36mA and -36mA at +/-12V.

SVF Parts Lists

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

Many of the parts for this circuit board are surface mount devices but not all of them. Take special care when ordering your parts that you order the correct type of part. This parts list shows the type of part needed whereas the circuit diagram does not.

A quick note on European part descriptions. R is shorthand for ohm. K is shorthand for kilo-ohm. 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. For electrolytic capacitors the maximum working voltage is normally given with the value, eg. 1uF/63V is a one microfarad capacitor with a working DC voltage of 63V.

To prevent loss of th 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.

SVF Main Board issue 1 Parts List

Resistors

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

22R	R33, R34
220R	R10, R11, R14, R15
330R	R29
680R	R32, R35
3K3	R2
10K	R6, R9, R27
18K	R30, R31
33K	R7, R21, R22, R25, R26
47K	R23, R24, R28
100K	R5, R12, R13, R18, R19, R20
110K	R3
150K	R1
1M	R4
1M5	R16, R17

R8 is a special positive temperature coefficient (PTC) through hole resistor with a nominal value of 1K and temp. co. of anywhere between +3000ppm/K and 3900ppm/K. eg. Arkaneohm LT16S102F33. Alternatively, it can be a standard 1/8W 1K through hole metal film resistor if you don't mind a small amount of temperature dependant drift in the centre frequency of the filter. It is mounted across the top of Q1, therefore can only be fitted once Q1 is soldered in place.

Capacitors

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

10pF	C8
100pF	C2
220pF	C4, C7

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

100nF/50V	C1, C3, C5, C6, C10, C11, C12, C13

The following capacitors are standard through hole polyester film capacitors with a lead spacing of 5mm (0.2”).

470nF, 63V polyester	C9

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

2u2/63V electrolytic	C16, C17
47uF/25V electrolytic	C14, C15

Discrete Semiconductors

The following devices are surface mount parts.

1N4148WS signal diode	D1, D2, D3
MMBFJ201 JFET transistor	Q2, Q3
BCM857DS dual PNP transistor	Q1

The following devices are standard through hole parts.

1N5819 Schottky diode	D6, D7
LM4040-10V	D4, D5

Integrated Circuits

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

LM13700M dual OTA	U2
TL072ACD dual op-amp	U1, U3
OPA2134UA dual op-amp	U4

Trimmer (Preset) Resistors

100K multiturn (eg. Bourns 3296W)	TUNE
20K multiturn (eg. Bourns 3296W)	V/OCT
100K 3/8" trimmer (eg. Bourns 3386F)	OFF1, OFF2

Miscellaneous

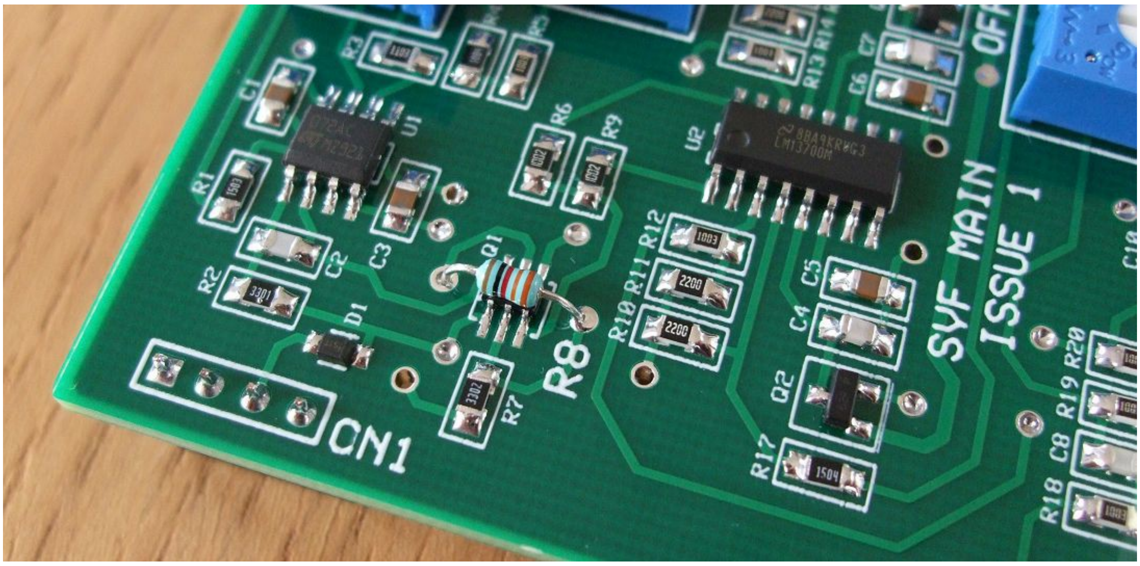
The following through hole parts are mounted on the topside of the board.

Axial ferrite bead	L1, L2
2 x 5 0.1" boxed header	STRIPE

The following parts are mounted on the underside of the board.

4-way 0.1" SIL socket	CN1, CN2, CN3, CN4, CN5, CN6

Special care must to be taken to ensure that connectors CN1 to CN6 are mounted perpendicular to the board surface.



R8 straddles Q1 which will make the two devices more or less the same temperature. You could add a little thermal paste between them but this is not necessary for a filter that does not self-oscillate.

Parts List for VCF Pot Board issue 1

Note: R1, R29, D1 and D2 are not fitted

Resistors

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

0R	R2, R3
100R	R8
330R	R30, R31, R32
3K3	R7, R13
10K	R25
22K	R23
33K	R28
47K	R9, R12, R19, R22
82K	R4, R24
100K	R5, R6, R10, R11, R15, R16, R26, R27
150K	R17
390K	R14
470K	R20, R21
1M5	R18

Capacitors

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

33pF	C2, C5, C10

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

100nF	C1, C3, C4, C6, C9, C11, C12

The following capacitors are standard through hole polyester film capacitors with a lead spacing of 5mm (0.2”).

470nF/63V polyester	C7, C8

Discrete Semiconductors

The following devices are surface mount parts.

BZX384-C2V7 zener diode	D5, D6
BZX384-C5V1 zener diode	D3, D4

Integrated Circuits

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

TL072ACD dual op-amp	U1, U2, U3

Potentiometers

All pots are Alpha 9mm vertical types. The pot's shaft type will depend on the control knob chosen for that particular pot. The front panel was designed so that the frequency, resonance and the two CV pots use 13mm diameter control knobs, and the three mixer pots use 'slip on' micro control knobs. I used four 13mm Davies 1900H clones, and three Thonk T18 Micro knobs. These need four 1/4" round shaft pots, and three 6mm splined shaft pots.

50K linear (1/4" round)	FREQUENCY, RESONANCE, CV1_POT, CV2_POT
50K linear (6mm splined)	ASYM, LVL_1, LVL_2

The pots are to be fitted to the underside of the board and their pins soldered from the topside. Note that there is no component identification on the reverse side of the board so use the front panel as guide to make sure the parts go into the correct locations.

Miscellaneous

SPDT on-on switch	Soft Clip

Seven control knobs

3.5mm socket	Notch, BP, HP, LP, Input 1, Input 2, CV 1, CV 2

The eight 3.5mm sockets and switch are to be fitted to the underside of the board and their pins soldered from the topside. See later for more details.

4-way SIL 0.1" header	CN1, CN2, CN3, CN4, CN5, CN6

Special care must to be taken to ensure that connectors CN1 to CN6 are mounted perpendicular to the board surface.

Part Sourcing

For general information regarding where to get parts and suggested part numbers for Oakley projects please see my Parts Guide at the project webpage or direct from here:

http://www.oakleysound.com/parts.pdf.

The front panel is obtained either from Schaeffer in Germany, or Front Panel Express (FPE) in the US. The database for the panel is provided on the project webpage and this file can be opened, edited and ordered using the Frontplatten Designer program available free from Schaeffer or FPE. The cost of the 10HP wide panel was around 28 Euros at the time of writing. The panel is 2.5mm thick and has a natural silvery finish. The black text is printed, although it is possible to edit the database to have the text engraved.

The Arkaneohm 1K temperature coefficient resistor is available from Thonk.

The pots are Alpha 9mm vertical pots as sold by Thonk and others. I use Thonk's standard ones with 6.35mm (0.25") round or 6mm splined shafts. Other shaft types are available, like D-type, and you should purchase those that are compatible with your choice of control knob. I prefer the round shafts for the larger controls as they require control knobs that have a fixing screw. Although these knobs take more time to place and secure, the height at which the knob sits on the pot shaft is under your control. Push fit knobs can sometimes not sit at the right height which is unsightly if they are too high, or graunch against the pot's securing nut and washer if they are too low.

For the knobs used on my prototype I used four black 13mm Davies 1900H clones, and three Thonk T18 black Micro knobs. These need four 1/4" round shaft pots, and three 6mm T18 splined shaft pots.

The toggle switch is a standard sub-miniature type that is designed to fit directly into a PCB. The on-on switches are available from Thonk as their DW1 sub-miniature switch. They are made by Dailywell and offer good performance for their price.

The eight 3.5mm sockets are the same type and are available from Thonk known either by their Thonkiconn moniker or as part number PJ398SM.

The fixed 2.54mm (0.1") interconnects are in two parts, the male header, and the female socket or receptacle. I use six four way single in line (SIL) connectors in this project. The ones I use are made by Multicomp, although more expensive types are available which may offer increased longevity. These are the Multicomp part numbers for the parts I used:

4 way socket	2212S-04SG-85
4 way header	2211S-04G

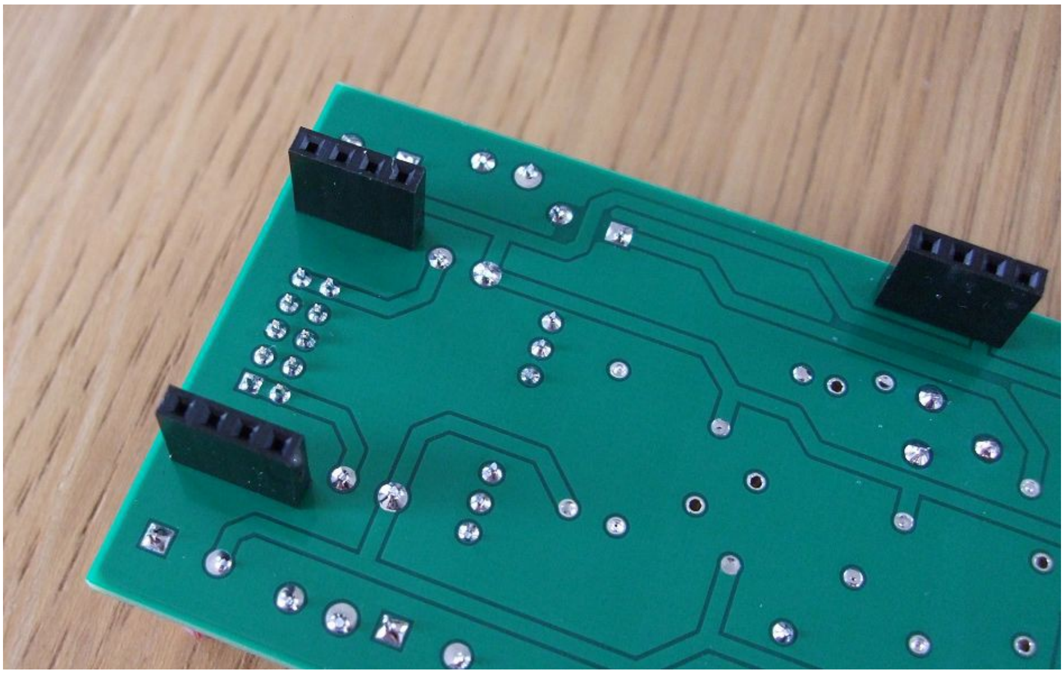
Populating the Circuit Boards

For general information on how to build Oakley modules, including circuit board population, mounting front panel components and making up board interconnects please see my generic Construction Guide at the project webpage or http://www.oakleysound.com/construct.pdf.

Main Board Construction

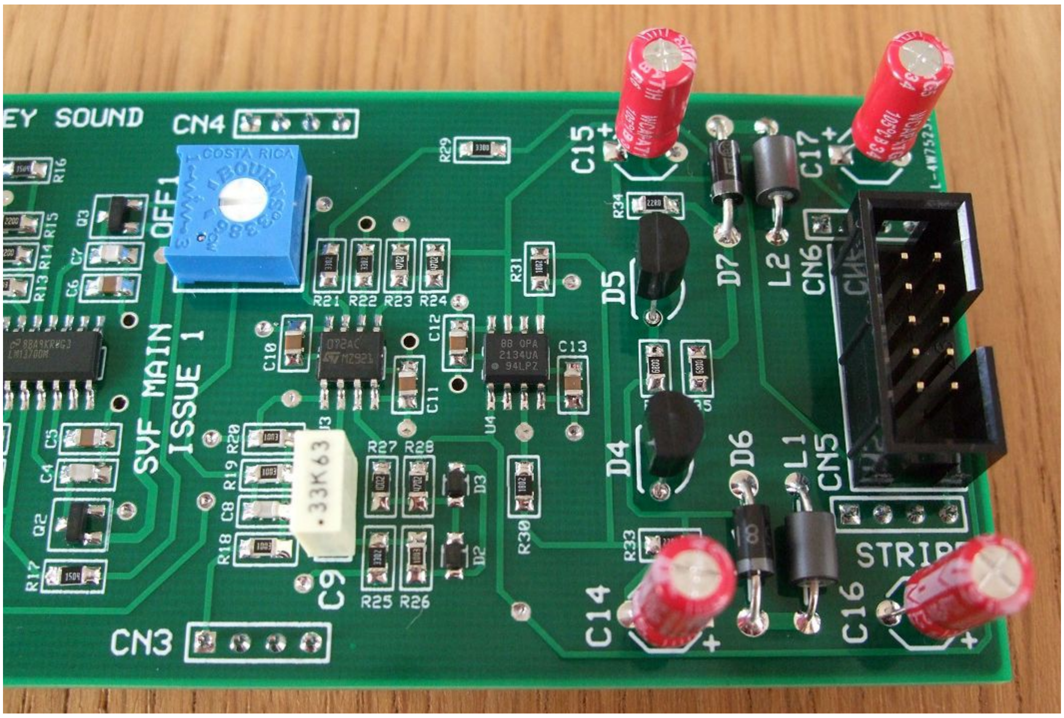
All the surface mount components should be soldered first. Take care to treat all transistors, diodes and ICs as static sensitive devices. I usually solder the resistors first, then the capacitors, then the discrete semiconductors, and then the ICs.

The next items to be soldered are the single in line (SIL) sockets. These are to be fitted to the underside of the board and soldered from the top side.



Special care should be taken to ensure that each SIL socket sits at right angles to the board surface. One way to do this is to temporarily fit the SIL headers into the SIL sockets and fit the Main board and Pot board together with the headers and sockets in between. Then solder the headers to the main board. Pulling the two boards apart should reveal all the sockets neatly in place and at the correct angle.

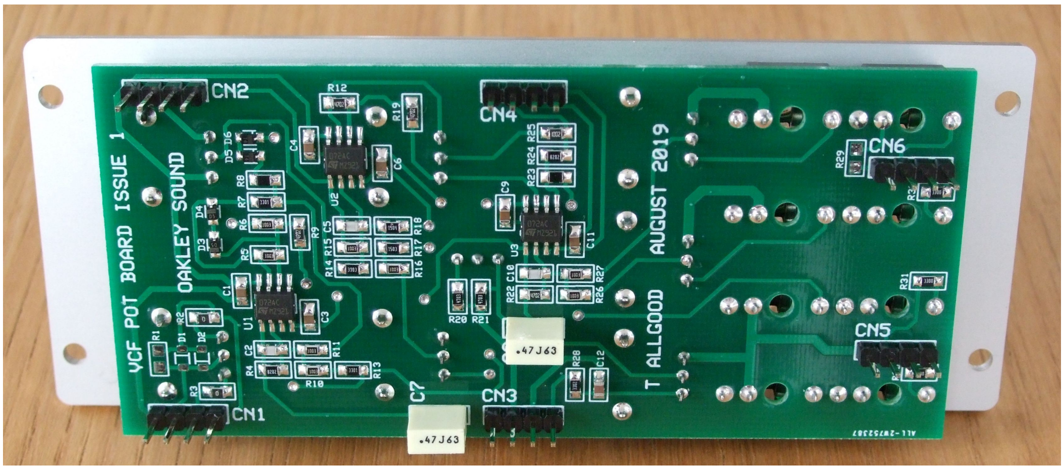
The next items to be soldered will be the through hole components. Remember that diodes and electrolytic capacitors are polarised so they need to be fitted the right way around. You also should especially make sure that the boxed header is correctly orientated. Pin 1 is normally designated with a little arrow shape on the plastic housing of the header and this should align with the square pad on the board. Also, the hole in the housing should correspond to the little box shape on the board's printed legend.



Pot Board Construction

Except for the pots, switch, and sockets all the parts are to be fitted to the topside of the board. The surface mount parts should again be soldered first.

Now fit the SIL headers into place and solder from the underside of the board. It may be worth temporarily fitting the Pot board to the Main Board so as to hold the headers exactly at right angles while you solder.



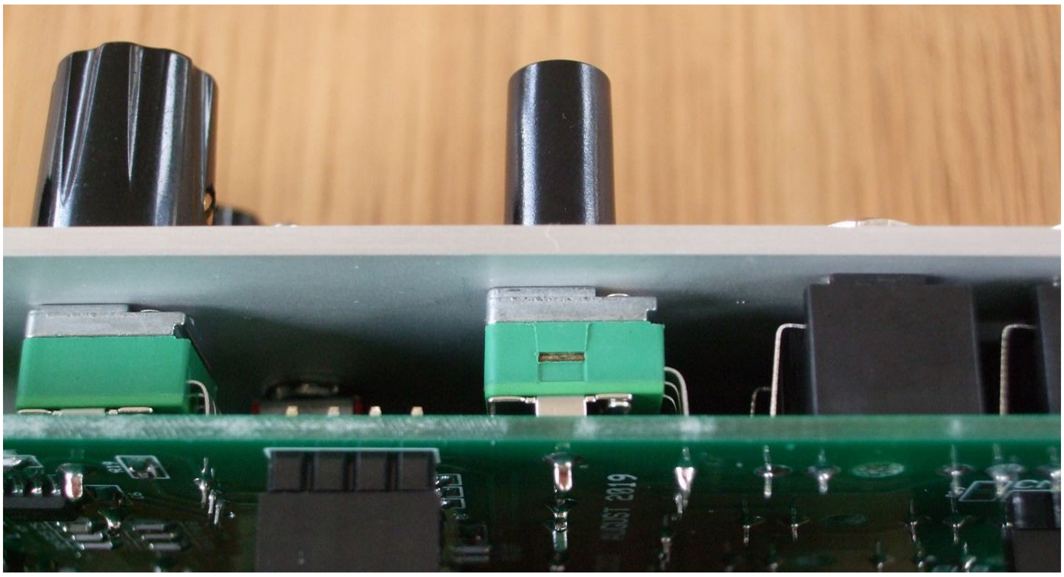
The remaining parts are the front panel components and these will be fitted to the underside of the board and soldered from the top of the board. There are no legends on the underside of the board to indicate where the parts will go but it should be clear where they need to be fitted. To ensure the correct alignment of these parts before soldering you should have your front panel ready. The panel will be used as a jig to hold the parts in the correct place while you solder. Failure to use the panel as a jig for these components could mean that the panel will not slide onto the components.

The first parts to place, but not solder, will be the pots. The pots should be fitted so that the three pot pins go into the board first, then ease in the two lugs into their holes and push in firmly so that the pot clicks into place. The pot will sit securely in the board even without soldering.

Now fit the eight sockets and switch into their places on the board but again do not solder yet. Place a toothed washer that came with the switch onto the threaded bush of the switch.

Now ease the panel down onto the pots, switch and sockets making sure that all their threads are sitting snugly in their holes. Place a washer and a nut on each of the top four pots and all the sockets, but not the three mixer pots nor the switch. Tighten the nuts but not too tightly. Turn the module over to reveal the topside of the circuit board. Now solder all the pots and sockets, but not the switch yet.

Flip the module over again and fit a single nut, and if you have one, a flat dress washer, to the exposed thread of the switch. You can ignore the other nut that came with the switch, as well as any washers that have locating lugs. Gently tighten the nut taking care not to scratch the panel. The switch will be pulled towards the panel and slightly off the board's top surface when you tighten the nut. The module can be turned over and the switch's leads soldered.



You can just see the toothed washer between the top of the switch and the inner surface of the front panel.

Initial Testing

It is prudent to test the main board on its own before fitting it to the pot board and panel. If you have a bench power supply that allows you to select the current limit then set both the +12V and -12V supplies to a current limit of 75mA. This should prevent any serious meltdowns if there is a problem with the build.

Power up the main board on its own. If you can measure power supply current then it should be around -23mA from the negative rail and +23mA from the positive rail. Anything significantly more than this, like 35mA, will indicate a problem. If you can't measure current then check that no devices are getting warm.

If all is well, then the main board can be powered down. Wait a minute and then attach the main board to the pot board. Be very careful to ensure all the SIL connections are correctly in place.

Power up the module and, if you can, check the current draw of the module. It should be between 30mA and 40mA. Anything significantly different to this, say over 60mA, will indicate a problem. The actual current taken will depend on the position of the frequency pot and Tune trimmer. A higher cut off frequency will take more current as the LM13700 has to work harder.

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 and patch this into Input 1. A low A, 220Hz is a good note to use. Set the soft clip switch upwards to engage the clean mode. Turn up the Input 1 level pot to its maximum.

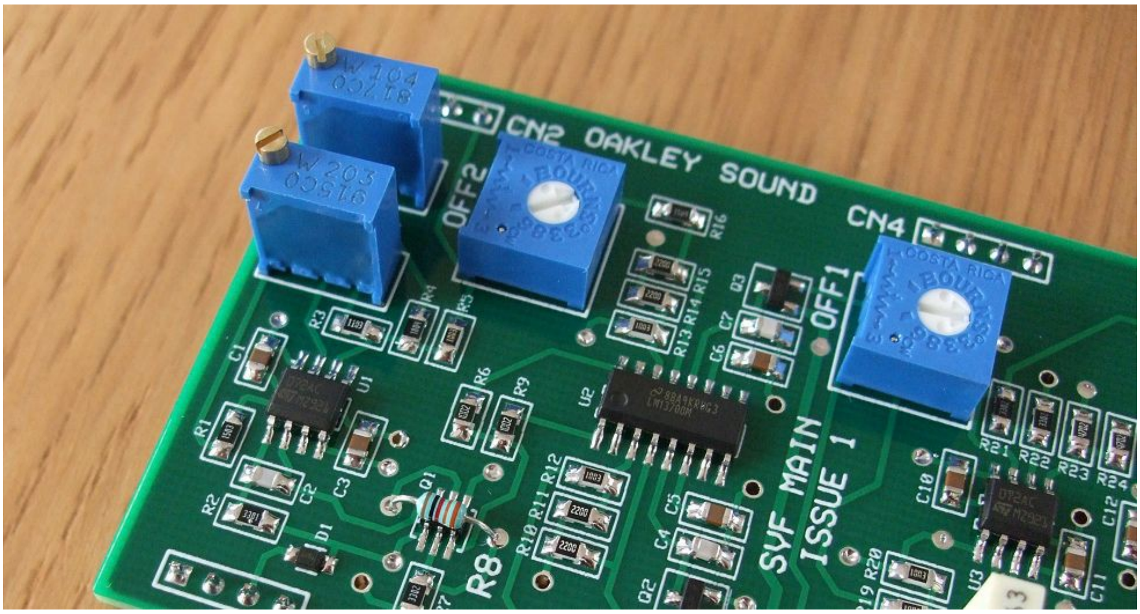
Moving the Frequency control should produce the usual and distinctive filter effect from the low pass output. From the high pass output you will the sound become increasingly more fizzy and faint as you turn up the frequency control. The band pass output will sound similar to the high pass output but when the frequency is turned to its minimum it will produce very little output. The notch output should sound similar to the input signal but will sound like weak phasing when the frequency control is moved back and forth. Turning the Resonance up will accentuate the ‘electronic’ nature of the sound on all four outputs. The SVF module should not self-oscillate.

Listening to the low pass output with the sawtooth input connected again, patch an LFO to the CV2 input. Check the CV2 pot allows you to control the depth of the sweep, which should go from no sweep to very large sweeps at its maximum position. Now swap the LFO over to the CV1 input. Notice that the minimum sweep depth should occur with the CV1 pot at its mid point. Use a slowish 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.

Patch a low A 110Hz triangle wave to Input 2 and check that the Input 2 level control behaves correctly. Now with the frequency control set to maximum and the resonance set low, flip the mode switch down to engage the soft clip circuit. You should notice that as you turn up the Input 2 level control the sound changes as the output becomes louder. It will become less bright and more hollow. If you are monitoring the output on a oscilloscope you'll see the triangle wave's peaks become squashed at higher input levels. Turning up the asymmetry control should change the sound to become more pulse wave like. It will also get quieter too as the input signal is more heavily compressed.

If all this happens, the chances are that you have a working module, and you can now calibrate the module.

Calibration



In this build I have used two Bourns 3296W multiturn trimmers (V/OCT and TUNE) and two Bourns 3386F 3/8" trimmers (OFF1 and OFF2).

OFF1

This determines the offset voltage needed to be applied to the first integrator stage of the state variable filter core to minimise the DC error voltage on the low pass output.

Monitor the the output voltage (with respect to 0V) on the low pass output. Turn the frequency control to its maximum value. Adjust OFF1 so that the voltage is as close to 0mV as you can get it.

OFF2

This determines the offset voltage needed to be applied to the second integrator stage of the state variable filter core to minimise the DC error voltage on the band pass output.

Note that OFF1 needs to be calibrated before OFF2. Monitor the the output voltage (with respect to 0V) on the band pass output. Turn the frequency control to its maximum value. Adjust OFF2 so that the voltage is as close to 0mV as you can get it.

V/OCT

This multiturn trimmer adjusts the scaling of the two CV inputs. Adjust this so that there is an octave jump in cut-off frequency when the CV1 input is raised by one volt and the CV1 pot is turned up full. The SVF is not capable of self oscillating and as such it is not that important to get the tracking perfect. Indeed, you can simply leave this trimmer in its mid position and save yourself some bother. To set it correctly requires either a spectrum analyser (these are often included with your DAW software) or a keen pair of ears.

Plug a 1V/octave source into the CV1 socket. This may be your keyboard or sequencer's pitch CV output, or the CV output of a midi-CV convertor. Patch a VCO's sawtooth output into the SVF's Input 1 and tune the VCO to an A at 110Hz. Do not patch the CV into the VCO as we want this to be simply oscillating at a fixed frequency.

Now listen to the output coming from the low pass output. Set the resonance control to its maximum value and tune the frequency control so that the SVF accentuates only the fundamental (first harmonic) note of the input signal. You should hear a sine type note and it'll be much louder than any other. If it sounds distorted turn the input level down until the sound heard is clean. Now play a note two octaves higher on your sequencer or keyboard and the filter's cut-off point will jump up roughly two octaves. An ideally scaled filter will now be accentuating the second overtone (third harmonic) of the sawtooth as the CV has risen 2V. If not, adjust the V/OCT trimmer so that it does. Now play the original note again and retune the filter from the front panel to pick out the fundamental again if it isn't already doing so. Then go up again two octaves and re-adjust V/OCT to once again emphasise the third harmonic. You may need to repeat this process several times so that the filter does not need adjusting when the CV is changed by 2V.

TUNE

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

Final Comments

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

If you can't get your project to work and you are in the EU, then Oakley Sound Systems are able to offer a 'get you working' service. If you wish to take up this service please e-mail me, Tony Allgood, at my contact e-mail address found on the website. I can service either fully populated PCBs or whole modules. You will be charged for all postage costs, any parts used and my time at 25GBP per hour. Most faults can be found and fixed within one hour, and I normally return modules within a week. The minimum charge is 25GBP plus return postage costs.

If you have a comment about this builder's guide, or have 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 the great people on the Synth-diy and Analogue Heaven mailing lists and those at Muffwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK