Oakley Sound Systems

5U Oakley Modular Series

Diode Superladder (DSL) Voltage Controlled Filter

PCB Issue 2

Builder's Guide

V2.0.2

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the issue 2 Diode 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, and a list of the various interconnections.

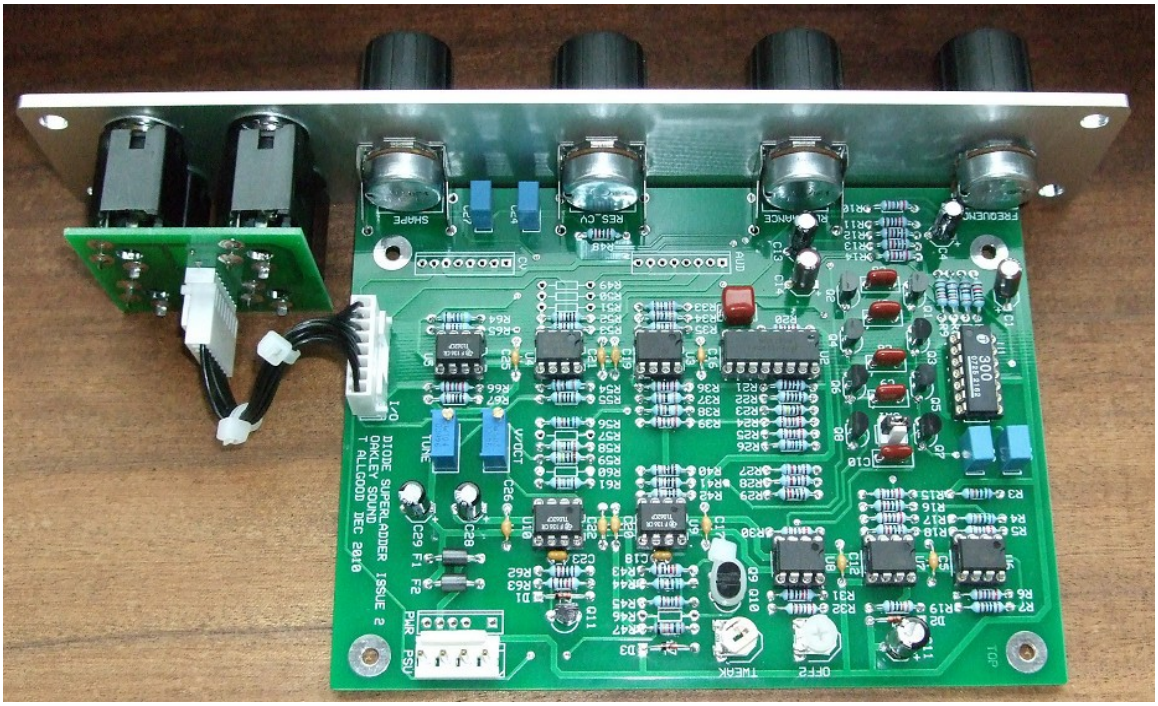
For the User Manual, which contains an overview of the operation of the unit, the history of the various board issues, and the calibration procedure, please visit the main project webpage at:

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

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

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

The issue 2 Diode Superladder PCB



This is the issue 2 Oakley Diode Superladder module behind a natural finish 1U wide Schaeffer panel. Note the use of the optional Sock4 socket board to facilitate the wiring up of the four sockets.

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

The design requires plus and minus 15V supplies. The power supply should be adequately regulated. The current consumption is about 30mA for each rail. Power is routed onto the main PCB by either a four way 0.156” MTA156 type connector or the special five way Synthesizers.com MTA100 header. You could, of course, wire up the board by soldering on wires directly. The four pins are +15V, ground, earth/panel ground, -15V. The earth/panel connection allows you to connect the metal front panel to the power supply’s ground without it sharing the modules’ ground line. More about this later.

The main PCB has four mounting holes for M3 bolts, one near each corner. These are not required for panel mounting if you are using the three 16mm pot brackets.

The board size is 106mm (deep) x 142mm (high).

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

Issue 2 Diode Superladder Parts List

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

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

A quick note on European part descriptions. R is shorthand for ohm. K is shorthand for kilo-ohm. R is shorthand for ohm. So 22R is 22 ohm, 1K5 is 1,500 ohms or 1.5 kilohms. For capacitors: 1uF = one microfarad = 1000nF = one thousand nanofarad.

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

Resistors

1% 0.25W metal film types are to be recommended. However, 5% ones can be used in all places except where 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.

330R	R19
1K	R21, R55, R22, R7
1K PTC +3000ppm/K	R47
2K2	R11, R43, R31, R63, R8, R9
3K9	R2
10K, 1%	R48, R13, R12, R29, R16, R18, R62, R1
12K	R28, R27
15K	R26
18K	R53
22K, 1%	R24, R10, R39, R6,R5, R20, R17, R36, R25, R15, R32, R30, R14, R66
27K	R45
33K	R34
47K	R65, R64
100K	R67, R42, R41, R40, R52, R50*, R51*, R49*, R46*
180K	R61
220K	R33, R35
270K	R58
390K, 1%	R54, R38, R37, R57*, R60*
750K	R56, R4, R44, R3
1M	R23, R59

Capacitors

100nF axial ceramic	C16, C20, C12, C19, C21, C5, C17, C22, C26, C25
100pF C0G 2.5mm ceramic	C23, C18
15nF polyester film	C10
1uF, 63V 5mm polyester film	C24, C15, C3, C2, C27
33nF polyester film	C6, C7, C8, C9
2u2, 63V electrolytic	C29, C28
10uF, 35V electrolytic	C1, C14
22uF, 35V electrolytic	C13, C4
47uF, 35V electrolytic	C11

Discrete Semiconductors

BC549 NPN transistor	Q1, Q2, Q3, Q4, Q5, Q6, Q7, Q8, Q9, Q10
BC560 PNP transistor	Q11
1N4148 silicon diode	D1, D3
10V zener diode	D2

Integrated Circuits

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

Trimmers (preset) resistors

50K cermet multiturn	V/OCT
100K cermet multiturn	TUNE
100K 6mm horizontal carbon	OFF2
20K 6mm horizontal carbon	TWEAK

Potentiometers (Pots)

All pots Alpha 16mm PCB mounted types

47K 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

Switch

On-On (SPDT) toggle switch	SW1

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

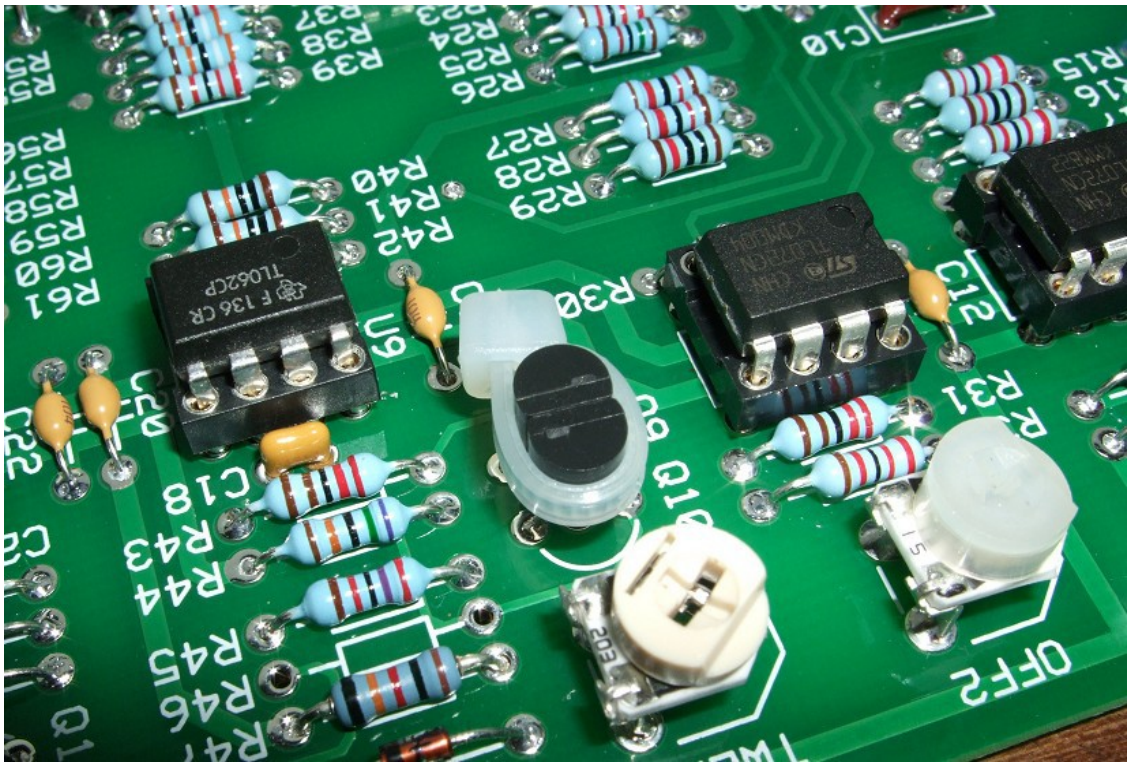
112APC Switchcraft 1/4” socket	SK1, SK2, SK3, SK4

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



Q9 and Q10 are thermally locked together.

Q9 and Q10 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 GND	2
Earth/PAN	3
-15V	4

Pin 1 on the I/O header has been provided to allow the ground tags of the jack sockets to be connected to the powers supply ground without using the module’s 0V supply. Earth loops cannot occur through patch leads this way, although screening is maintained. Of course, this can only work if all your modules follow this principle.

Power connections – Synthesizers.com

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

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

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

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

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

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

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

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

The prototype Diode Superladder 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.

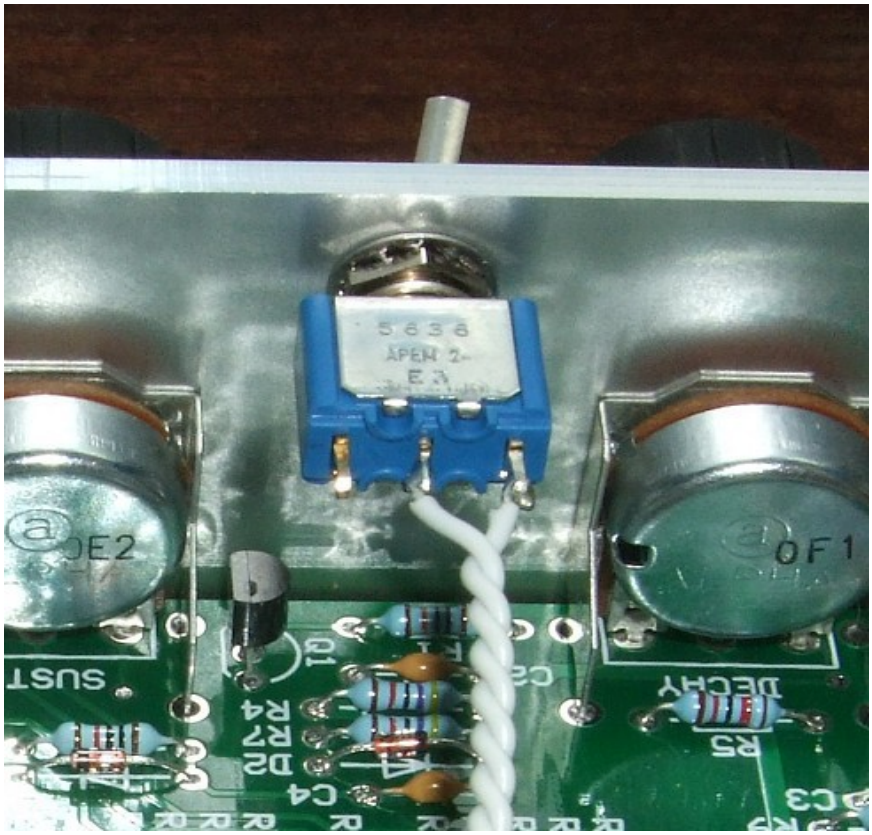
Once fitted to the front panel the ground tags of each socket can be all connected together with solid wire. I use 0.91mm diameter tinned copper wire for this job. It is nice and stiff, so retains its shape. A single piece of insulated wire can then be used to connect those connected earth tags to pin 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	NC	No connection	
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 RESONANCE	NC lug
Pin 8	INPUT	Connect to INPUT	Signal lug

Wiring the Stage Switch

If you are fitting the switch then you need to connect it to the pad marked as SW1 on the Diode Superladder PCB. The switch is wired with a twisted wire pair. This simply made by twisting two pieces of insulated multistrand wire together to form a simple cable. You can use two different colours but you don't have to as it does not matter which wire connects to which lug on the switch. All the stage switch does is short out the SW1 pads when its set to FIVE. However, to ensure the switch's behaviour tallies with the front panel legending you need to make sure the top two tangs of the switch are connected. The lower tang is left unsoldered.



This photograph shows the switch on the ADSR/VCA module, but the switch on the DSL is wired in much the same way.

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 four of these pads too. These will be pads 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.

Testing, testing, 1, 2, 3...

Apply power to the unit making sure you are applying the power correctly. Check that no device is running hot. Any sign of smoke or strange smells turn off the power immediately and recheck the polarity of the power supply, and the direction of the ICs in their sockets and the polarity of the electrolytic capacitors.

Assuming everything is OK so far, it is time to apply an audio input. Use a bright signal like a sawtooth output from a VCO. The A below middle C, 220Hz, is a good note to use.

Connect your amplifier or mixing desk input to the output socket. Click the switch into position FOUR. Set the FILTER SLOPE pot to the far right, at STEEP, 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 filter slope pot to the far left, at LOW, 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 filter slope pot you can get the sound to change from the one pole to four pole sound at either extremes of the pot's movements. Just left of the middle position you should get a 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.

Now click the switch into FIVE. You should notice the sound is now slightly more muted and less bright. The five pole filter has a steeper roll off in both the STEEP and LOW positions of the filter slope pot.

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 CV 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.

Final Comments

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

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

If you have a comment about this builder's guide, or have a found a mistake in it, then please do let me know. But please do not contact me or Paul Darlow directly with questions about sourcing components or general fault finding. Honestly, we would love to help but we do not have the time to help everyone individually by e-mail.

Last but not least, can I say a big thank you to all of you who helped and inspired me. Thanks especially to all those nice people on the Synth-DIY and Analogue Heaven mailing lists and those at Muffwiggler.com.

Tony Allgood at Oakley Sound

Cumbria, UK