

Oakley Sound Systems

Eurorack Modular Series

MOSPhaser

PCB Issue 1

Builder's Guide

V1.2

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

This is the Project Builder's Guide for the issue 1 MOSPhaser Eurorack 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 to build the project, and the simple calibration procedure.

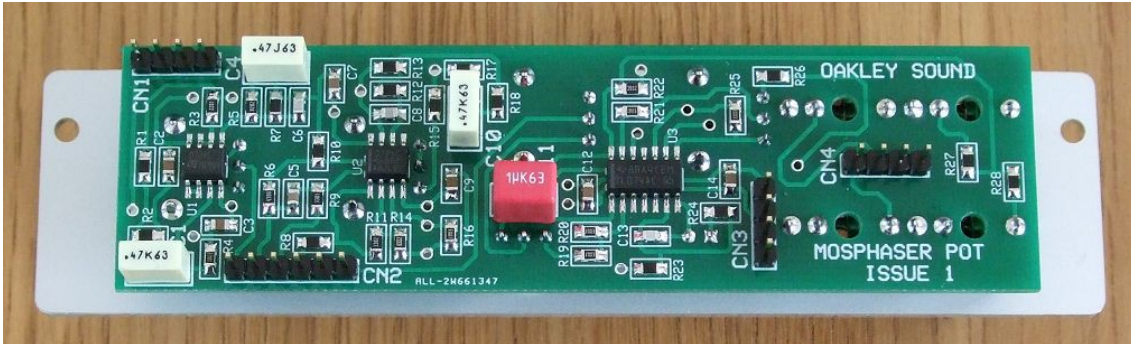


For general information regarding where to get parts and suggested part numbers please see our useful Parts Guide at the project webpage or <http://www.oakleysound.com/parts.pdf>.

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>.

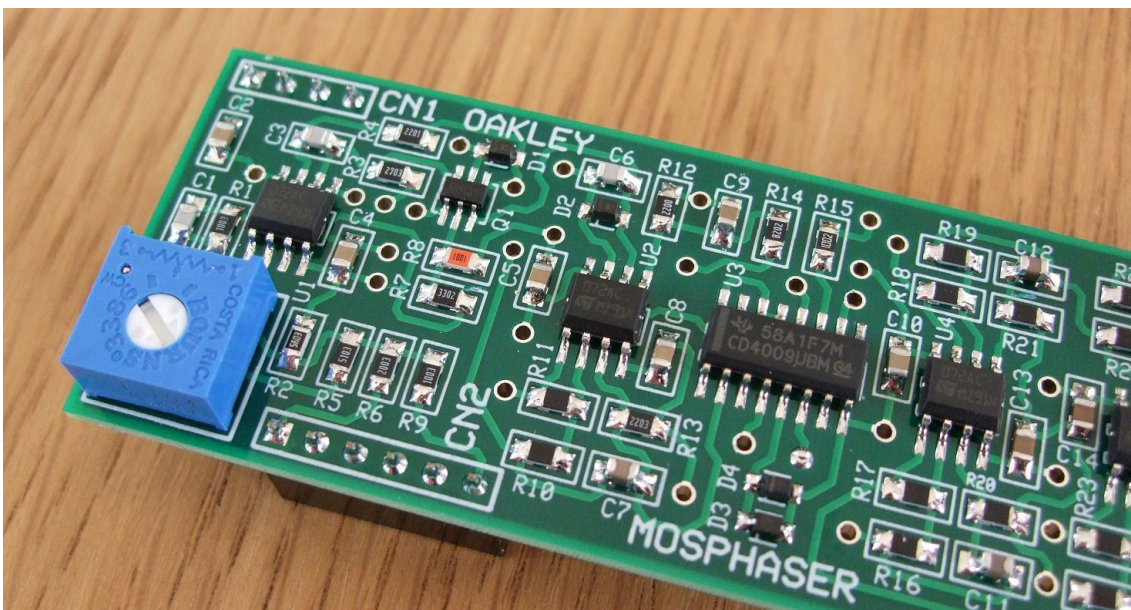
The MOSPhaser PCB Set

The electronics of the Oakley MOSPhaser is built on two printed circuit boards (PCBs). The Pot board holds the four pots, switch, LED, input and output sockets, and the audio input and output circuitry. 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 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 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 five all pass filter stages, the frequency control circuitry, 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 top board's electronics.



A close up of the main board showing the frequency control circuitry and MOSFET array.

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 NPN transistor, Q1, is housed in a very small SOT457 package but can be 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, 29 mm (wide) x 107 mm (high). The boards are designed to go behind a panel that is 6HP wide.



The module comprises of two circuit boards connected together with three six way and one eight 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 joint between the plastic housings of CN1, CN2 and CN4 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 slightly below +40mA and -40mA at +/-12V.

MOSPhaser 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 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.

Main Board issue 1 Parts List

Resistors

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

22R	R28, R29
220R	R12
2K2	R4
4K7	R22, R19, R24, R25, R17, R10, R11, R18, R23, R16
10K	R15
33K	R7
82K	R14
100K	R1, R9
200K	R6
220K	R13, R27, R26, R20, R21
270K	R3
510K	R5
560K	R2

R8 is a special positive temperature coefficient resistor with a nominal value of 1K and temp. co. of anywhere between +3000ppm/K and 3900ppm/K. It is surface mount in 0805 size. Alternatively, it can be a standard 0805 1K metal film resistor if you don't mind a small amount of temperature dependant drift in the centre frequency of the phaser.

1K PTC	R8
--------	----

Capacitors

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

100pF	C1
470pF	C3, C6
47nF	C7, C12, C11, C15, C16

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

100nF/50V	C4, C2, C5, C10, C14, C17, C13, C8
1uF/25V	C9

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

2u2/50V electrolytic	C20, C21
22uF/25V electrolytic	C18, C19

Discrete Semiconductors

The following devices are surface mount parts.

1N4148WS signal diode	D1, D2
5V1 zener diode	D3
BAT42WS Schottky diode	D4
BCM847DS dual NPN	Q1

The following devices are standard through hole parts.

1N5819 Schottky diode	D5, D6
-----------------------	--------

Integrated Circuits

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

CD4009UBM	U3
TL072ACD dual op-amp	U1, U2, U4, U5

Trimmer (Preset) Resistor

100K 3/8" trimmer TUNE

eg. Bourns 3386F or similar

Miscellaneous

Axial leaded ferrite bead L1, L2

4-way SIL socket CN1, CN3, CN4

6-way SIL socket CN2

2 x 5 0.1" boxed header STRIPE

CN1 to CN4 are mounted on the underside of the board, facing down, and soldered from the top. Special care must be taken to ensure that these connectors are mounted perpendicular to the board surface.

Pot Board issue 1 Parts List

Resistors

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

100R	R24, R26
470R	R6
510R	R2, R10
1K	R8, R13, R15, R23, R27, R28
2K2	R3
3K3	R9
4K7	R1, R12, R17, R18
10K	R14
22K	R7
33K	R11
47K	R22
82K	R5
100K	R4, R25, R21, R16, R20, R19

Capacitors

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

100pF	C6
470pF	C8
1nF	C13
47nF	C5

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

100nF	C2, C3, C7, C9, C12, C14
-------	--------------------------

The following capacitors are through hole components.

470nF/ 63V polyester film	C1, C4, C10
1uF/63V polyester film*	C11

*C11 is the timing capacitor for the LFO. Using 1uF will create a frequency range of roughly 0.011Hz to 5Hz. If this is too slow then you can reduce the capacitance accordingly. 680nF will produce roughly 0.016Hz to 8Hz, 470nF will produce 0.024Hz to 11Hz, and 330nF will produce 0.034Hz to 15Hz.

Discrete Semiconductors

The following devices are standard through hole parts.

3mm bi-colour LED Front panel LED

See later in this document for details regarding the fitting of the LED.

Integrated Circuits

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

TL072ACD dual op-amp U1, U2
TL074AC quad op-amp U3

Potentiometers

All pots are Alpha 9mm vertical pots with 6.35mm round shafts.

10K linear Emphasis
50K linear Frequency
50K log Modulation, LFO Rate

The pots are to be fitted to the underside of the board and their pins soldered from the topside.

Four knobs to suit. Davies 1900H or clones thereof are to be recommended.

Miscellaneous

SPDT 'on-on' toggle switch Invert

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

Special care must be taken to ensure that connectors CN1 to CN4 are mounted perpendicular to the board surface.

Thonkiconn 3.5mm socket Input, Output, CV In, LFO Out

The four 3.5mm sockets are to be fitted to the underside of the board and their pins soldered from the topside.

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 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 8HP wide panel was around 27 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 four pots are Alpha 9mm vertical pots as sold by Thonk and others. I use Thonk's standard ones with 6.35mm (0.25") round shafts. Other shaft types are available, like splined or D-type, and you should purchase those that are compatible with your choice of control knob. I prefer the round shafts 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 control knobs I use Davies 1900H clones available from Thonk and others.

The toggle switch is a standard sub-miniature type that is designed to fit directly into a PCB. These 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 PTC temperature coefficient resistor, R8, can be obtained as part number LT7339002A1K0JTE from Farnell and others.

The four 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 three four way and one six way single in line (SIL) connectors in the MOSPhaser. 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:

6 way socket	2212S-06SG-85
6 way header	2211S-06G
4 way socket	2212S-04SG-85
4 way header	2211S-04G

All the dual op-amps are recommended to be TL072ACD. Note the A in the part number suffix, this is the improved specification version of the standard TL072 which has lower offset voltages. Offset voltage is an unwanted voltage on the inputs of an op-amp which then manifests itself as an error voltage on the output of the op-amp. Other dual op-amps may be used such as the LF412CD and TLE2072CD.

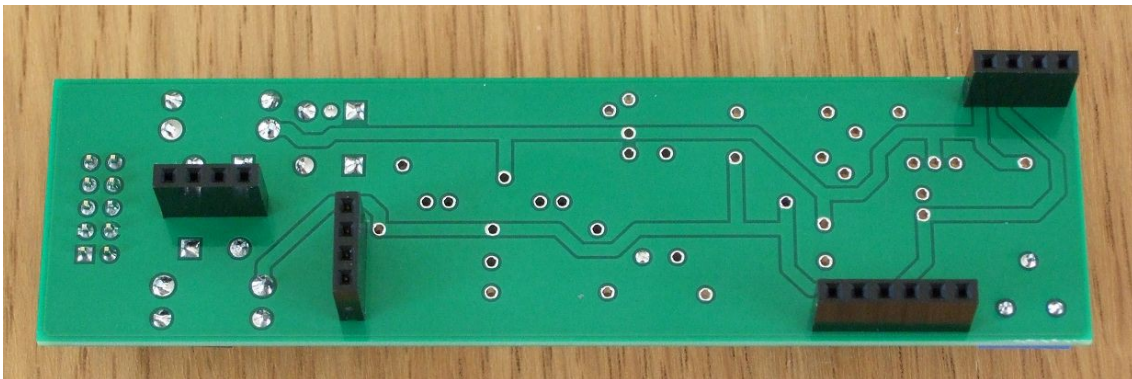
Populating the Circuit Boards

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>.

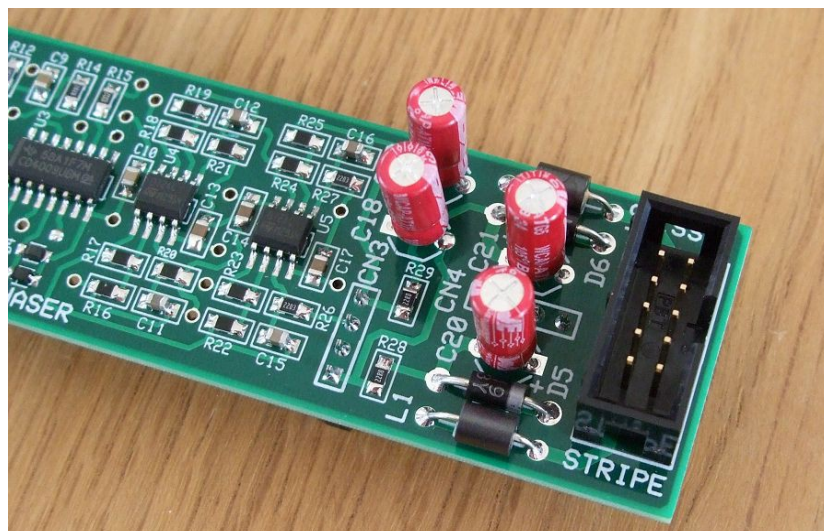
Main Board Construction

All the surface mount components should be soldered first. Take care to treat all ICs as static sensitive devices, especially the 4009UBM (U3) which is particularly vulnerable. 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.



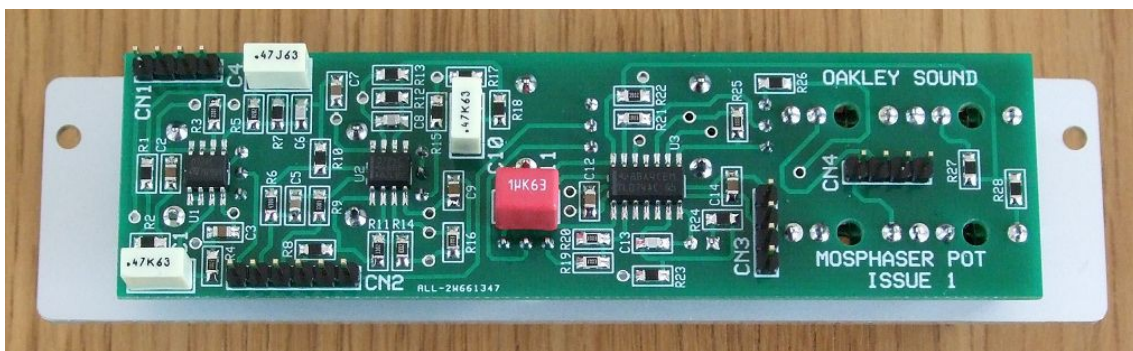
The next items to be soldered will be the through hole components. Remember that diodes and electrolytic capacitors are polarised so they need to be fitted the right way around. You also should

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, switches, LEDs 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. Note that the pots are of different types. The 50K linear will fit into the top location. The 10K linear will fit into the next one down. The two 50K audio (log) pots are fitted to the lower two locations.

Now fit the four sockets and switch into their places on the board but again do not solder yet. Place one of the toothed washers that came with the switches onto each switch.

Fit the LED loosely into its holes. I try to make my bicolour LEDs go red with a positive signal and green with a negative one. You may need to check with the datasheet for the types you bought but the ones I used had the long LED lead going into the hole with the square solder pad. Once again we are fitting the device into place but are not soldering the leads just yet.

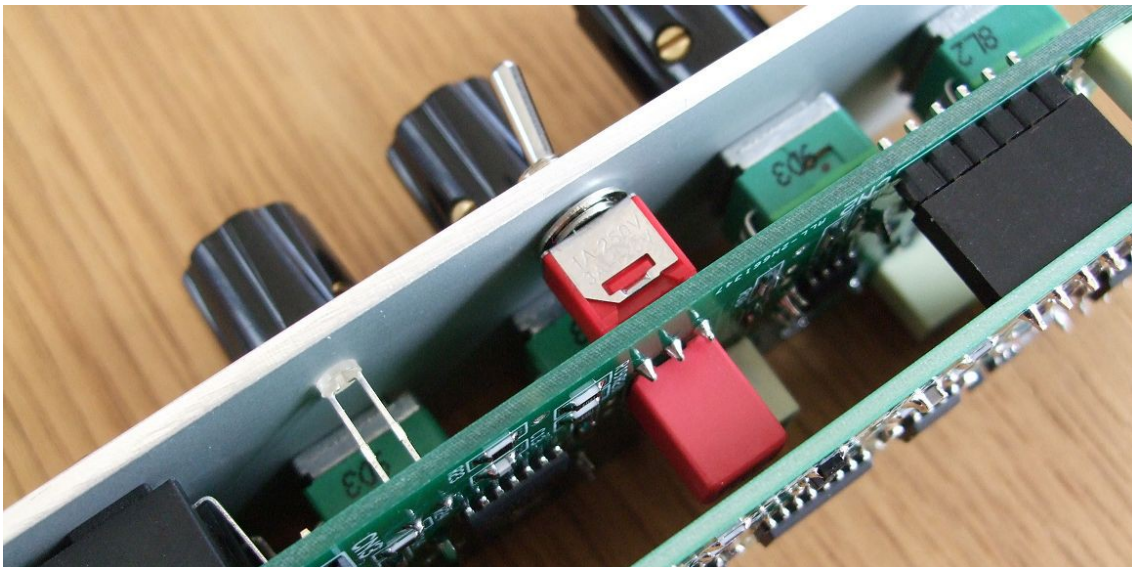
Now ease the panel down onto the pots, switch and sockets making sure that all their threads are sitting snugly in their holes. You can ignore the LED as it'll just sit below the panel for the time being. Place a washer and a nut on each of the pots and sockets. Tighten the nuts but not too tightly. Turn

the module over to reveal the topside of the circuit board. Now solder all the pots and sockets, but not the switches or LED 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.

Now tease the LED through its panel hole by using its leads that are sticking out the bottom of the board.

If you are using a standard dome topped LED then simply push it into the hole as far as it will go and solder each lead. If you are using flat topped LEDs then you may wish to have them align with the top surface of the panel. This is not a trivial task but one way is to use a bit of sticky tape over the holes and let the LED push up against that before soldering the leads.



This dome topped LED is fitted so that its base sits up against the underside of the front panel.

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 50mA. 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 -18mA from the negative rail and +18mA from the positive rail. Anything significantly more than this, like 50mA, 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 the module up again and, if you can, check the current draw of the module. It should be around +38mA and -38mA. Anything significantly different to this, say over 50mA, will indicate a problem.

The front panel LED should light up moving between red and green. If it doesn't, turn off and check all the parts again thoroughly. If your LED is OK try the LFO Rate pot. It should control the frequency of the LED's flashing. From around one cycle every 90 seconds in the counter clockwise position to around 5 cycles a second at its maximum if C11 is the suggested 1uF.

Now connect up an audio signal of some sort, any will do, but a simple sawtooth wave at around 220Hz is quite sufficient. Listen to the audio output, and play with the controls. With all other controls set to their minimum setting, sweep the Frequency pot up and down. Do you hear the characteristic phase sweep? If not, you have got a problem. If yes, now turn up the Emphasis. Using the frequency pot again, does the sweep have a more metallic ring to it.

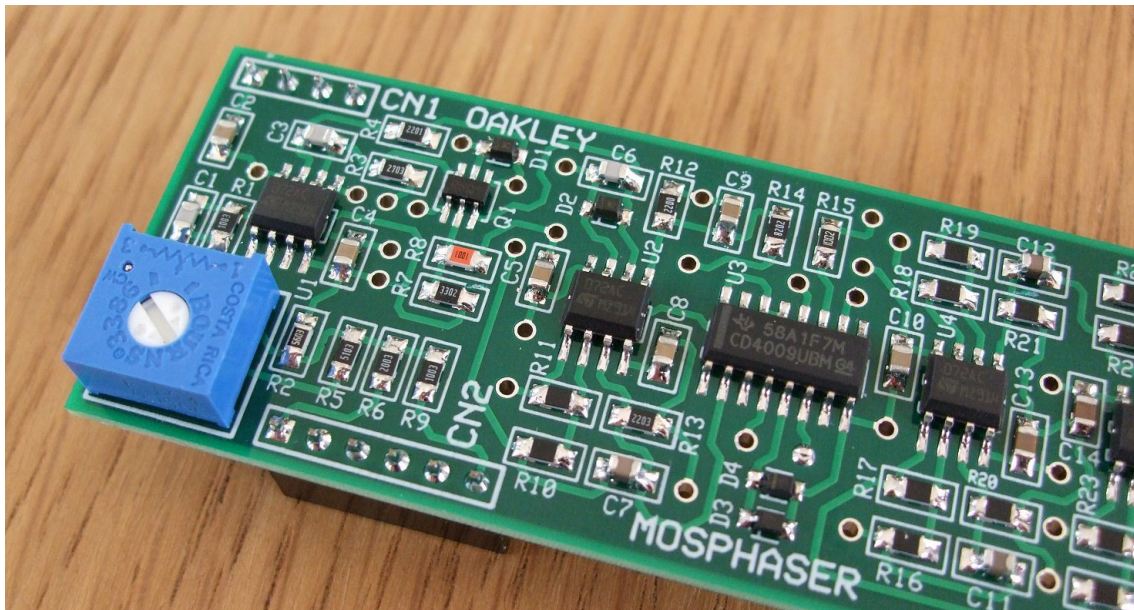
Now set the frequency and emphasis pots to their middle position. Turn up the Modulation. The LFO should now be modulating the phaser. Check that the LFO Rate control affects the speed of the modulation.

If all these things happen you almost certainly have a working module and you can now calibrate it.

Calibration

Apply power to module and allow to warm up for a few minutes. Turn the Frequency and modulation controls to their minimum positions.

Measure the voltage across zener diode D3 – which is to the right of U3, the 4009UBM. The red lead of your meter should go to the end of D3 that faces towards the top of the module, the black lead can go to the via hole just to the left of C10 which is connected to 0V. Adjust the trimmer so that the voltage reads +0.75V.



There is one trimmer on the MOSPhaser module. This is a Bourns 3386F.

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 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
© July 2019