Oakley Sound Systems

5U Oakley Modular Series

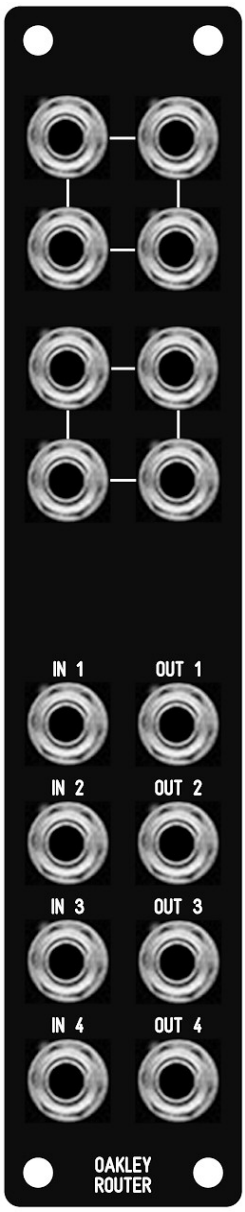
Router

PCB Issue 1

Builder's Guide

V1.1

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom



The suggested panel layout for the 1U wide Router module.

Introduction

This is the Project Builder's Guide for the issue 1 Router module from Oakley Sound. This document contains a basic introduction to the module and a full parts list for the components needed to populate the boards.

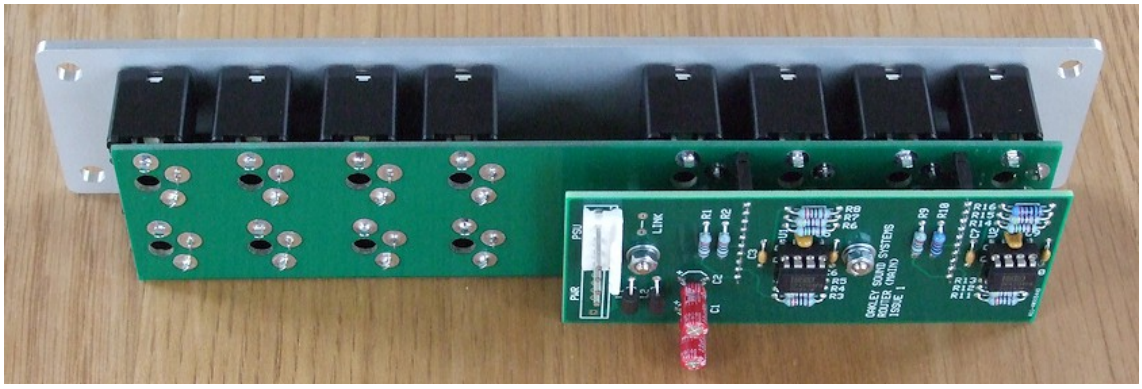
The project web page can be found at:

http://www.oakleysound.com/router.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 Oakley Router Module



The issue 1 Oakley Router as a single width MOTM format module in a natural finish Schaeffer panel.

The Oakley Router is a simple but useful module. The top section is two passive multiples. A multiple, or 'mult', is a number of sockets, in this case four, that are connected together in parallel. It can be used to send one output to up to three other destinations, splitting the input signal into three. This is useful for sending, for example, the gate output of a midi-CV convertor, to three envelope generators so that all three envelopes trigger together. Or a single sawtooth output from one VCO to two filters for parallel processing.

The lower section contains four identical inverting amplifier circuits. The amplification can be set when building the unit but the suggested gains are either -1, -0.32, or -0.1. Some audio interfaces and most guitar effects pedals distort with the high signal levels coming from a modular synthesiser so reducing the signal level can aid interfacing troubles. For audio interfaces that have such problems I recommend the gain be set to -10dB which is a voltage gain of 0.33. For guitar pedals it is best to reduce it to a tenth of the original signal which is voltage gain of 0.1. Each of the four output stages can have a different gain if required, for example, you can set the top two amplifiers to -1 and the bottom two to -0.1 if you wish. The

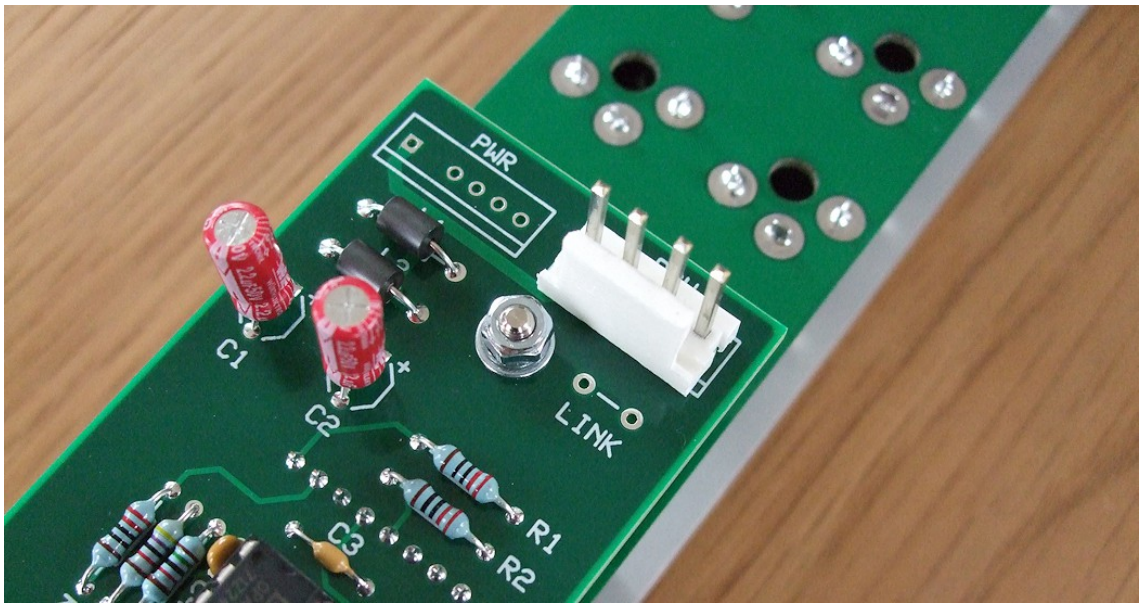
parts list in this document assume you will set all four amplifiers the same but you don't have to.

Each output is balanced and uses a 1/4” TRS (tip ring sleeve) socket which can be connected directly, using a balanced cable, to a balanced input of an audio interface or mixing desk without introducing ground loops. The outputs can, however, also be used with unbalanced inputs, and the outputs will not be damaged if you use standard mono unbalanced (TS) cables.

Each of the output amplifiers are inverting, that is a positive going signal is turned into a negative going signal. This has no audible effect on the audio signal but will change the polarity of any control voltages passed through it. The module could thus be used as an effective quad inverter if required.

The module comprises of two circuit boards, the larger two layer socket board and the smaller four layer main board. The main board has layers of conductive copper on the top and bottom sides as well as two internal copper layers. You'll notice that this board feels heavier than ordinary two layer boards of a similar size. It is more expensive to make a board this way but it does mean that fewer compromises need to be made when laying out the design. Both boards are required to build the project and are sold as a set.

The design requires a power supply of +/-12V to +/-16V. The power supply should be adequately regulated. The current consumption is around +16mA and -16mA at +/-15V, although this will go up slightly when the amplifier sections are being used. Power is routed onto the main PCB by either our standard four way 0.156” MTA156 type connector or the special five way Synthesizers.com MTA100 header.



The MOTM style power MTA156 header.

Router issue 1 Parts List

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

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

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

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

Main Board

Resistors

All resistors should be 1% 0.25W metal film types.

220R	R1, R2, R3, R8, R9, R10, R11, R16
47K	R4, R7, R12, R15

For a gain of -1, ie. 0dB, use the following value for the feedback resistors:

47K	R6 (Channel 1), R5 (Channel 2), R14 (Channel 3), R13 (Channel 4)

For a gain of -0.32, ie. -10dB, use the following value for the feedback resistors:

15K	R6 (Channel 1), R5 (Channel 2), R14 (Channel 3), R13 (Channel 4)

For a gain of -0.1, ie. -20dB, use the following value for the feedback resistors:

4K7	R6 (Channel 1), R5 (Channel 2), R14 (Channel 3), R13 (Channel 4)

Capacitors

100nF axial multilayer ceramic	C3, C6, C7, C10
2u2, 63V electrolytic	C1, C2

For a gain of -1, ie. 0dB, use the following value for the feedback capacitors:

22pF	C5 (Channel 1), C4 (Channel 2), C9 (Channel 3), C8 (Channel 4)

For a gain of -0.32, ie. -10dB, use the following value for the feedback capacitors:

47pF C5 (Channel 1), C4 (Channel 2), C9 (Channel 3), C8 (Channel 4)

For a gain of -0.1, ie. -20dB, use the following value for the feedback capacitors:

220pF C5 (Channel 1), C4 (Channel 2), C9 (Channel 3), C8 (Channel 4)

All feedback capacitors are C0G ceramic capacitors with a lead spacing of 2.5mm.

Integrated Circuits

OPA2134PA dual op-amp U1, U2

IC sockets can be used if you wish. You need two 8-pin DIL sockets.

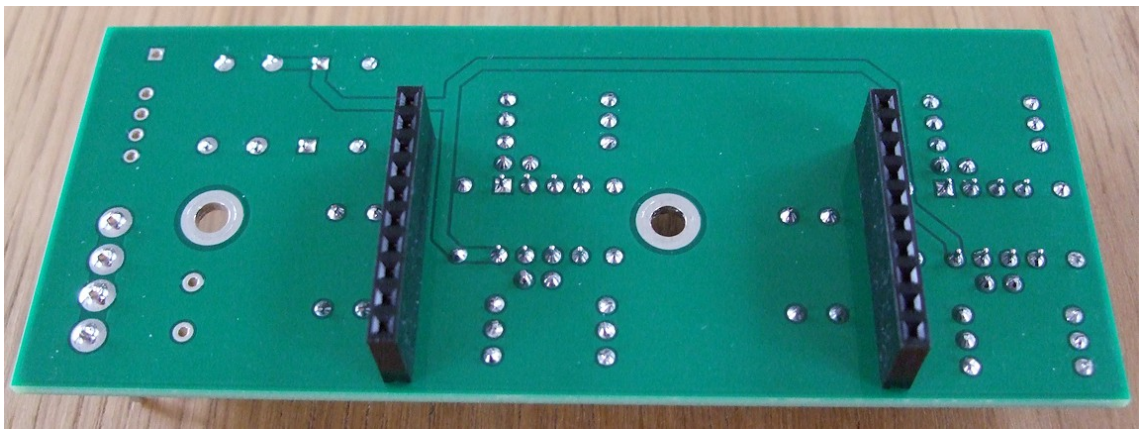
Miscellaneous

Leaded axial ferrite beads L1, L2

MTA156 4 way header PSU – Oakley/MOTM power supply
MTA100 6-way header PWR – Synthesizers.com power supply

A small wire link needs to be fitted into LINK if PWR is fitted. See later for more details.

10 way SIL socket CN1, CN2 – fitted to the underside of the board



Both headers are fitted to the underside of the board and soldered from the top side.

Socket Board

Do not solder any 1/4” sockets into place before you have read 'Assembling the Socket Board' section later in this document.

Switchcraft 112APC 1/4” sockets	SK1, SK2, SK3, SK4, SK5, SK6, SK7, SK8 IN1, IN2, IN3, IN4
Switchcraft 114BPCX 1/4” sockets	OUT1, OUT2, OUT3, OUT4
10 way SIL header	CN1, CN2 – fitted to underside of board

Miscellaneous

2 off 6mm M3 screw
2 off 11mm male-female hex spacer
2 off M3 flat washer
4 off shakeproof washer
2 off M3 nut

Part Sourcing

Most of the parts are easily found. However, you may find the following useful.

The 1/4” sockets are recommended to be Switchcraft 112APCX and 114BPCX because of the high quality and longevity. Alternatives are available in the form of the cheaper Rean NYS232 and NYS234 respectively.

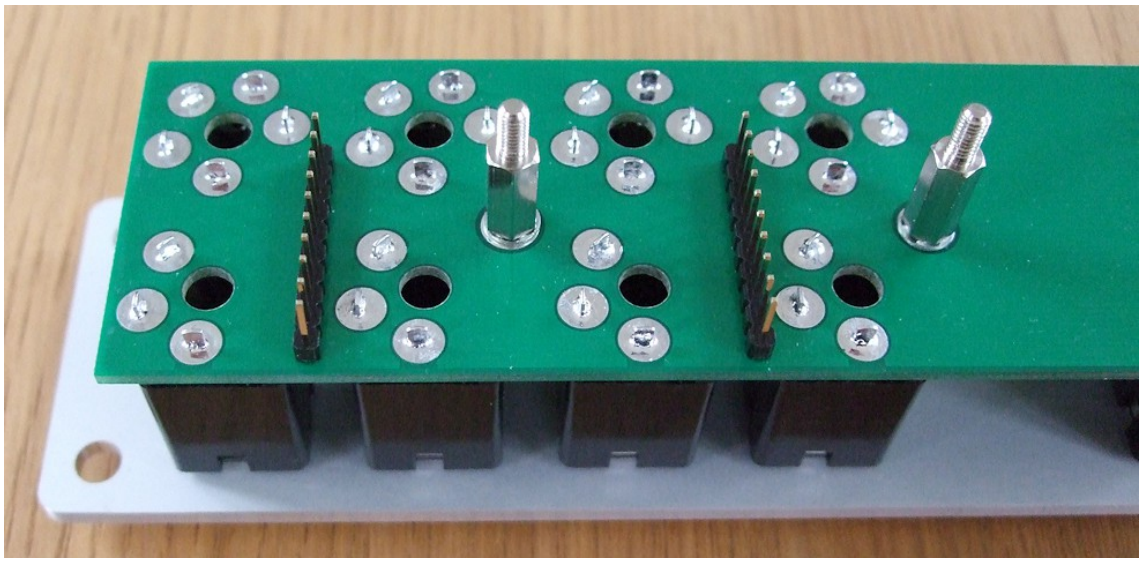
The board interconnects used to connect the main board with the socket board, are in two parts, the male header and the female socket or receptacle. Both interconnects are ten way single in line (SIL) connectors. The ones I have used in my builds are made by Multicomp, although more expensive types are available which may offer increased longevity. These are the Multicomp part numbers for the parts I used:

10 way socket	2212S-10SG-85
10 way header	2211S-10G

For the Multicomp headers above I needed 11mm spacers plus a shakeproof washer to fit the two boards together without stressing them. The 11mm M3 male-female hex spacers are made by Harwin, their part number R30-3001102.

Assembling the Socket Board

CN1 and CN2 must be soldered into place before any sockets are fitted. CN1 and CN2 are to be fitted to the underside of the board. Although there is no legending to show where they go it should be obvious. Because both headers are fitted to the underside they should be soldered from the top side of the circuit board. The topside of the board is the one with the text.



The completed socket board mounted on to the front panel. Note the shakeproof washer under each of the 11mm male-female hex spacers.

Because space is quite tight around the sockets there was limited room in which to site the two PCB mounting holes. As such both the holes are very close to their nearest sockets so it is nigh on impossible to fit the screws into these holes once the sockets are fitted. It is therefore essential that you fit the two 6mm M3 screws into their holes **before** you solder the IN1 and IN3 sockets in place. The screws should be fitted so that the head of the screw is on the same side as the sockets and the thread sticks out away from the front panel. You should temporarily fix each screw loosely into place with a nut so they don't fall out when you are soldering the sockets.

The sockets should only be soldered when you have soldered the two ten way headers and loosely fitted the two M3 screws.

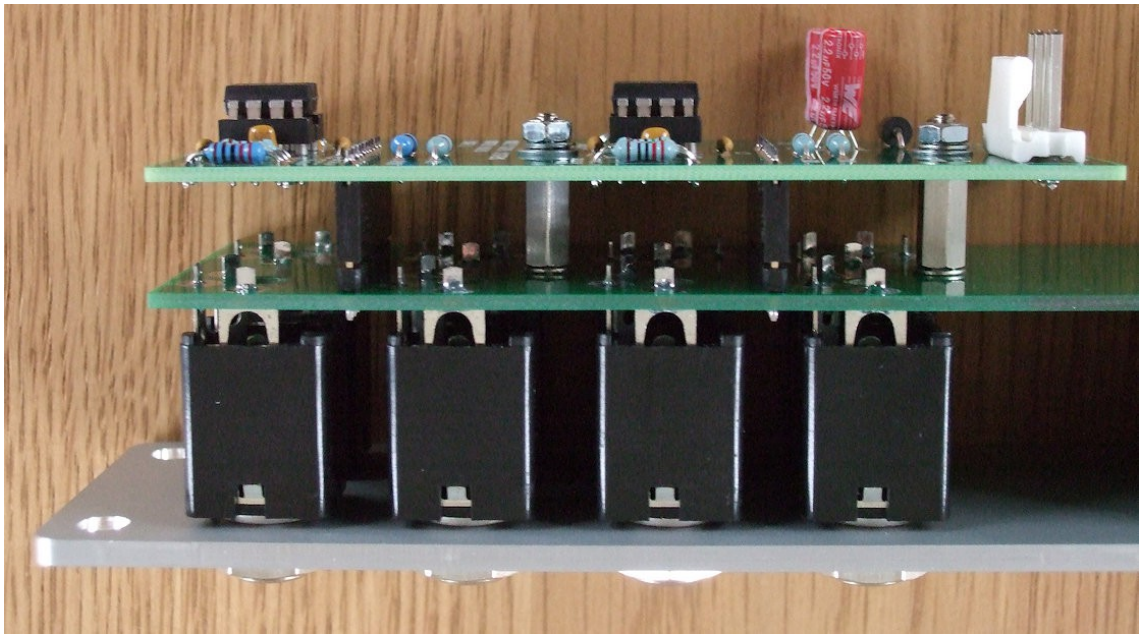
The best way to solder the sockets is to first place them into the front panel, making sure they are orientated to mate correctly with the socket board. I normally do this by placing the front panel face down with the ends of the panel held just above the bench with a couple of supports. I use some Lego Duplo blocks for this but a couple of bits of wood will do equally well.

Then I place the sockets into the front panel, making sure the bevel of each socket is facing towards the bottom left of the panel. I don't secure the sockets onto the panel at this stage but let gravity hold them in place. Take note that four of the sockets are TRS types and have more pins than the usual mono TS ones. Once the sockets are slotted into the panel, the socket board can be gently lowered and teased onto the socket pins. The sockets can then be soldered

from the underside of the board. If the board is slightly bowed then push down on the board to keep it flat while soldering.

You should now secure the two board spacers into place by fitting each one onto the M3 screws. A shakeproof washer should be fitted in between the spacer and the socket board. A small screwdriver should be able to reach the head of each screw to tighten them up.

The socket board is now ready to accept the completed main board which should just slide into place.



The exposed thread of each of the spacers should be fitted with firstly a flat washer, then a shakeproof washer and finally the securing nut. Gently tighten each one to fix the main board into position.

Connections

MOTM and Oakley Power

The PSU power socket is 0.156” Molex/MTA 4-way header. Friction lock types are recommended. This system is compatible with MOTM systems.

Power	*Pin number*
+15V	1
Module ground (0V)	2
Socket ground	3
-15V	4

Pin 1 on the LWR header, and the pot brackets are connected to pin 3 of the PSU header and has been provided to allow the ground tags of the jack sockets and panel metalwork to be connected to the power supply ground without using the module’s 0V supply. Earth loops cannot occur through patch leads this way, although screening is maintained.

MU and Synthesizers.com Power

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 the pin in location 2 should be pulled out. In this way location 3 is actually pin 2 on my schematic, location 4 is actually pin 5 and so on.

Power	*Location number*	*Schematic Pin number*
+15V	1	1
Missing Pin	2	
+5V	3	2
Module ground (0V)	4	3
-15V	5	4
Socket Ground*	6	5

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

If fitting the PWR header and using it with a standard MU power distribution system, you will also need to fit a wire link into the position marked LINK. This link connects the socket and panel ground with the module ground. Simply solder a solid wire hoop made from a resistor lead clipping, or bit of thin solid core wire, to join to the two pads of LINK.

* If you are using an Oakley MU Dizzy distribution board with a five way power cable, the wire link is not required. This enhanced system will allow the socket ground to be kept separate from module ground to prevent ground loops.

Final Comments

If you have any problems with 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 UK, then Oakley Sound Systems are able to offer a 'get you working' service. If you wish to take up this service please e-mail me, Tony Allgood, at my contact e-mail address found on the website. I can service either fully populated PCBs or whole modules. You will be charged for all postage costs, any parts used and my time at 25GBP per hour. Most faults can be found and fixed within one hour, and I normally return modules within a week. The minimum charge is 25GBP plus return postage costs.

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

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

Tony Allgood at Oakley Sound

Cumbria, UK