Oakley Sound Systems

Octal Resonator

PCB issue 1

Audio Processor

User's Guide

V1.1

Oakley Sound Systems
PENRITH
CA10 1HR
United Kingdom

Introduction

This is a highly effective audio processor that features eight fixed frequency resonators in parallel. This device is not a filter bank in the strictest terms but it can be used like a filter bank to change the frequency response of an audio signal. With this module one can replicate the subtle nuances of cavities on hollow bodied instruments, or at the other extreme, mangle the response so much that the output sounds nothing like the input.

Each resonator is essentially a band pass filter. The frequency response of each filter operates in such a way as to let only a narrow band of frequencies through. The selectivity of each resonator can be controlled by a front panel pot. At its highest settings the resonator will ring at its selected frequency.

You can think of each resonator as being analogous to a bottle. When you blow air over a bottle it resonates and a note is heard. Fill the bottle up with water and the note changes. The Oakley Octal Resonator is eight such bottles, each with a different level of water in it.

You can't change the frequency of each resonator but you can change the depth of the effect each resonator has on the sound. Each pot controls the Q or resonance of the filter. With all the pots fully off, there is only a little sound from the unit. Turning up each pot will accentuate a particular frequency.

The resonant frequencies are approximately: 250Hz, 350Hz, 500Hz, 700Hz, 1000Hz, 1400Hz, 2000Hz, 2800Hz.

The Octal Resonator has three outputs. The first output is a combination of all eight resonators. This can be used on its own or perhaps mixed with the original signal with a suitable mixer. The Oakley MultiMix is an ideal companion because you can not only add to the resonators' outputs, but also subtract.

The other two outputs take the outputs of the left and right hand sides of the resonator module. That is, the 250Hz, 500Hz, 1000Hz and 2000Hz, come out of the 'left out' socket. And the 350Hz, 700Hz, 1400Hz and 2800Hz resonators come out of the 'right out' socket. You can use these two outputs to generate unusual stereo effects.

The PCB

The Octal Resonator module features a three PCB set. Two boards, the main and auxiliary boards, carry the electronics. Another board carries the sockets. The largest board holds the single power header. The two larger boards are interconnected with a 12-way 0.1” preformed jumper.

Previously, many Oakley modules have had the sockets, switches and extra pots wired to the board by individual wires. This module allows all the socket wiring to be done via the socket PCB and one MTA solderless connection. If you are building this module in the standard Oakley format this new system will reduce assembly time and possible wiring errors.

Some people will wish to use this Oakley design in a non standard format, such as fitting it to a Analogue Solutions rack or one of their own invention. This is perfectly easy to do. Simply do not use the socket board and wire the main board to the sockets as per usual.

I have provided space for four control pots on each of the two large PCBs. The pots are directly soldered onto the board and the board is held rigidly at right angles to the front panel by the Oakley pot brackets. The pot spacing is 1.625” and is the same as MOTM modular synthesiser. The board is fully MOTM panel compatible if the board is fitted vertically, ie. in a 2U wide panel.

No mounting holes are provided in the PCBs for additional mounting. The only way to support the board is via the Oakley pot brackets.

The board sizes are:

Main board: 83mm x 143mm
Aux board: 52mm x 143mm

There are detailed instructions later in the document about how to build all the boards.

Power Supplies

This module is designed to run from plus and minus 15V supplies. These should be adequately regulated. Although moderate perturbations in the supply will not cause any major depreciable effect, output noise will be worsened by the use of an unregulated supply. A voltage of 17.5V or higher on either rail may damage the module.

The current consumption is about 35 mA per rail. Power is routed onto the PCB by a four way 0.156” MTA or Molex type connector. You could, of course, wire up the board by soldering on wires directly. The four pins are +15V, ground, panel, -15V. The panel connection allows you to connect the metal front panel to the power supply’s ground without it sharing the module’s ground line.

This unit will also run from a +/-12V supply with a slight reduction in dynamic range.

Circuit Description

The schematic is split over three pages, each page corresponding to one of the three PCBs provided in the PCB set. The first page covers the main board, the second page the auxiliary board and the last page covers the socket board.

Let us first look the main board and at the power supply section in the lower right of page one. Power is initially supplied via the usual four way MTA or Molex connector labelled PWR. As is the custom for Oakley modules, I have used ferrite beads to act as high frequency filters on the power lines. Decoupling at the point of entry is provided by C24 for the positive rail, and C25 for the negative rail. Additional decoupling is also provided elsewhere on the board by the host of other capacitors shown to the left of the power circuitry. These capacitors keep the power supply clean of noise, and provide a reservoir for the little bursts of current that the circuit takes in normal operation.

Two grounds are provided, one for the circuit itself, and one for the earthing of the jack sockets on the front panel.

There are six ICs, U1, 2, 3, 4, 5, and 6, that require power. The power supply to each IC is shown separately from the main schematic to avoid cluttering the diagram.

To the left of the IC power supplies we see the 12-way interconnect that joins the main board to the auxiliary board. There will be more about this later.

The main board features three basic sections; the resonators, of which there are four on this board, and the input and output amplifiers.

The input signal arrives from the socket board via a 5-way 0.1” interconnect called ‘sockets’. Pin 5 carries the input signal and this is passed to the input amplifier. This amplifier is based around U5a (pins 1, 2, 3). R25 and R27 act as an attenuator cutting the input signal down to around 10% of its value. This is to ensure that each resonator will not be overloaded with a standard +/-5V (10V peak to peak) input signal. If you are using the Octal Resonator with line level signals, ie below 2V peak to peak, you can make R27 a 47K resistor.

The circuitry based around U5 is a noninverting amplifier of unity gain. This means it doesn’t actually amplify but provides two purposes:

1. It provides the input signal with a large degree of isolation from the resonators input impedance. What this means is that the input impedance of this module is fixed by the values of R25 and R27 and not the unpredictable input of the resonators themselves. U5a effectively buffers the input and thus any input signal is unaware of the ensuing mayhem performed on that signal later on in this module.

2. The feedback around U5a is not just a simple resistor. The additional combination of R24 and C23 provide stability for the output of U5a when driving all eight resonators.

Each resonator is essentially the same circuit. Each one differing only in the value of capacitance in the circuit. Each resonator consists of a dual op-amp and a handful of passive components. Let us look at the one based around U4 in the middle of page 1 of the schematic.

The resonator circuit is based on the classic Deliyannis bandpass filter first published back in 1968. A variation of this design was used in the Teisco, also known as Kawai, 110F synthesiser in its filter bank. That filter bank used eight Deliyannis derived band pass filters each tuned at different frequencies and summed the outputs together to form a single new output. This output could then be summed with the input signal if desired to produce interesting tonal structures. It is the topology of that filter bank that spawned the version you see in the Oakley Octal Resonator.

The standard Deliyannis bandpass filter is a fixed frequency, fixed Q band pass filter. Q is the ‘quality’ of the filter and is often called the ‘resonance’ in most synthesiser filters. Essentially the Q determines the gain of the filter at its centre frequency. High Q means that the filter will have a dramatic increase in gain at the centre frequency of the filter. Teisco allowed the Q in each of their eight filters to be controlled by a reverse log pot that fed back some of the output back into the input. This was very effective and provided a means whereby each filter's effect on the input signal could be controlled.

The Oakley Octal Resonator works in more or less the same way, but instead of using a reverse log pot we use a linear pot and op-amp buffer. This actually has two distinct advantages over using reverse log pots. The first, and most important as far as I'm concerned, is the down to the lack of availability in high quality reverse log pots. Secondly, the output impedance driving the feedback point [the junction of R22 and R14] is pretty much constant over the audio band resulting in a fixed centre frequency as Q is changed.

I'm not going to go into the mathematics that define the Deliyannis filter since that would be beyond the scope of this User Guide, but for those that are interested in deriving the centre frequency you can use this formula:

Fo = 1/[235400 x C]

Where C is the value of each of the capacitors used; C15 and C16 in the case of the resonator based around U4.

The value 235400 comes from values of R12, R22, R14. More details about the Deliyannis filter can be found in Electronotes EN73 and EN91. But other sources are available on the internet.

U4b (pins 5, 6, 7) is the op-amp around which the bandpass filter is made. U4a (pins 1, 2, 3) is simply a buffer circuit. This buffer circuit 'sniffs' the voltage on the wiper of the pot and produces at its own output, a copy of this voltage. The op-amp thus isolates the wiper voltage from the low resistance of R14 and the actions of the resonator circuit.

Notice the pot is actually wired backwards. Unlike a normal volume control, this pot passes more of the audio output back when it is at its minimum value. This means that for the more Q, the bandpass filter needs less feedback. This is similar to the feedback around a state variable filter (Oakley SVF), but opposite to that of a Moog Ladder filter. The bandpass filter will oscillate uncontrollably at its centre frequency if there is no feedback at all. R8 ensures that the pot can never be fully turned off, so there is always a small feedback path even when the Q control is at its maximum.

R7 loads the pot's wiper with a defined resistance. When you load a pot with a resistance its law changes. The pot's law is a definition of how the resistance changes between the wiper and one of the pot's end points as you rotate the pot's shaft. A linear pot, such as this one, will increase its resistance in an even fashion as the pot is turned. So without R7 you would see the voltage increase evenly as you turned the pot at a constant speed. By loading up the pots wiper with R7 we can distort the law. In this circuit R7 acts to slow the increase in voltage as the pot is turned. The wiper's voltage is dragged down by the action of R7 until the pot gets very close to its maximum position. In this way we have actually created a pot with a log type law.

As we have heard the actions of U4a prevent the pot from being loaded from any effects of the resonator. Only R7 will affect the law of the pot.

The centre frequency of each resonator is set by a capacitor network. In the resonator we have been looking at, there are only two capacitors, C16 and C15. Both of them must be the same value within the allowable tolerances of the design. In this case, each is 2.2nF which gives us a theoretical centre frequency of 1930Hz or so. Its labelled as 2kHz on the front panel and this is understandable when you consider that the capacitors could actually be anything from 1.98nF to 2.42nF with a 10% tolerance.

Using low tolerance capacitors like polystyrene is an option, and I will leave it up to you as to whether you do this. In my opinion this is an effect device for a synthesiser not a precise equalisation unit, so accuracy is not of paramount importance.

Unfortunately, we are not able to get the capacitances exactly as required for the exact match of the frequencies on the front panels. For many of the resonators we have to use parallel combinations of capacitors to get the desired capacitance. Placing two capacitors in parallel produces an equivalent capacitor equal to the sum of the individual capacitors. Thus for the lowest frequency resonator at 250Hz and based around U1, we needed 17nF in theory. I used a 15nF and a 2n2 which gives us 17.2nF. The theoretical centre frequency of this resonator is pretty close to its panel value at 247Hz.

Each resonator is fed with the same input signal and each resonator’s output is passed onto two summing amplifiers. Note that the output of each resonator can normally be considered as being out of phase with the input, ie, inverted.

For the four resonators on the main board, their individual outputs are passed to the summing amplifiers based around U6a and U5b. All the summing amplifiers are based on the classic inverting summing amplifier circuit. The extra inversion is handy since it corrects the phase inversion from the resonator circuits.

U6a sums the outputs of the four main board resonators only. U5b sums the outputs of all eight resonators, with the additional four outputs coming from the auxiliary board via the AUX interconnect. U6b sums only the four outputs of the auxiliary resonators.

The summing resistors are all of the same value and they are implemented on this board as resistor networks. This saves on board space and simplifies building.

R13, R23 and R15 provide the usual protection for the outputs of each summing amp. The series resistor stops the output amplifier from oscillating when driving long cables, and also prevents damage should you accidentally patch two outputs together.

The auxiliary board is basically a cut down version of the main board. It features just four resonators and the interconnect that joins the two large boards together.

Buying the components

All of the parts are easily available from your local parts stockist with the exception of the PCBs and the pot brackets. I use Rapid Electronics, RS Components and Farnell, here in the UK. Rapid's telephone number is 01206 751166. They offer a traditional 'paper' catalogue as well as an on-line ordering service.

In North America, companies called Mouser, Newark and Digikey are very popular. In Germany, try Reichelt, and in Scandinavia you can use Elfa. All companies have websites with their name in the URL. In the Netherlands try using www.telec.com.

The resistors can be 5% carbon 0.25W types except where stated. However, I would go for 1% 0.25W metal film resistors throughout, since these are very cheap nowadays.

The two types of resistor packs are what are called 'commoned'. These are the types that have a multiple of resistors inside the pack but all the resistors are joined at one end. So a four pack will have five pins; four for the individual ends and one for the commoned connection. The resistor packs are SIL, that is single in line packages, ie. all the pins lie in a straight line.

The pots are Spectrol 248 conductive plastic types and are held onto the boards with specially made Oakley pot brackets. Eight pot brackets are required. These are provided with the 'pot bracket kit' which also contains the eight extra nuts required to correctly fit the panel to the boards.

The two electrolytic capacitors are radial types. These are the types that are cylindrical in shape with their legs sticking out of the bottom. I use bog standard 2.2uF, 63V types for this non critical situation.

The PCB is another Oakley board to allow you to incorporate axial multilayer ceramics for the power supply decoupling. These are good components with an excellent performance. Various types exist but I tend to use the better quality C0G types from Farnell. They look like small yellow or orange beads with wires that stick out on either side like a resistor. They normally come delivered on tape. You can use 63V or 100V types.

All the other capacitors are metallised polyester box types. They have a lead pitch of 5mm (0.2") and come in little plastic boxes in a variety of colours. Typical working voltages will be 63V, 100V or even 400V. They key thing to look out for is physical size and lead pitch. Most of Rapid's are either 100V and 400V and all will fit on the board.

All the resonator capacitors may be replaced by axial polystyrene types if you can make them fit on the board. It is a shame that polystyrene capacitors are not as common place as they were as they do have better characteristics than polyester. Having said that I don't think there would be any sonic differences between using polystyrene and polyester types in this particular module. Other alternatives to polyester include polypropylene [get the miniature types] and multilayer ceramic [get the radial types].

All ICs are dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP, CN or a N in their part numbers. For example; TL072CP. Do not use SMD, SM or surface mount packages.

L1 and L2 are leaded ferrite beads. These are little axial components that look like little blackened resistors. They are available from most of the mail order suppliers. Find them in the EMC or Inductor section of the catalogues. Farnell sell them as part number: 108-267. Rapid sell them as part number: 24-4860.

The 12 way jumper interconnect is a one piece assembly bought ready made from several places. They come in various lengths, I use 105mm ones, but if you can get a 55mm long one that will fit too. They come pre-stripped and pre-tinned with solder too. Make sure you get the 0.1” (2.54mm) pitch variety. The Rapid part number is 22-1665.

Input and output sockets are now board mounted in the standard module assembly. You could of course use any types if you would like to wire your sockets up with individual wires.

I use the Switchcraft 112A 1/4” sockets in all my ready made modules. These excellent parts are also used on the Moog and MOTM modulars. The version you need to fit in the socket board is the 112APC. This part is stocked by most suppliers. Both Rapid and Farnell sell is at a reasonable price. Rapid also sell a Far Eastern clone of the 112APC part that is considerably cheaper, but doesn’t look as nice. Please note that the standard 112A will not fit easily into the boards as it has solder tags.

For the 0.1” interconnections I use the 26 awg MTA parts. These are made by Amp, now part of the massive Tyco empire. To use these effectively you need a special tool to poke the wires into the special ‘housings’ The housing contains specially shaped contacts that cut through the insulation of the wire so you don’t need to do any stripping. Just simply push down on the wire with the tool to lock it into place in the housing.

You can also use Molex strip and crimp systems. More about this one later on.

Name	***Farnell Part number***	***Amp’s part number***
Five way housing	588-283	640442-5
Five way header	588-600	640456-5
Handtool	589-494	59803-1

The special handtool you need is rather expensive if you are only doing one module. But following the success of our trials we will be using the MTA system for all new boards and new issues of older boards.

Finally, if you make a circuit change that makes the circuit better, do tell me so I can pass it on to others.

UK builders should know that there is now a ‘Oakley Preferred Parts List’ online. This can be found at **www.oakleysound.com/parts.pdf**.

North American builders should visit **www.wiseguysynth.net** for more information about buying parts in the US.

Parts List

This is an early issue of the documentation, I have checked the parts list, but I can miss things. If in doubt, check against the circuit diagram, this is usually always correct. Please e-mail me if you find any discrepancies.

A quick note on European part descriptions. 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.

There are three boards in the PCB set, 'board A', 'board B' and the 'sockets board'. Each one has its own parts list. Note that there are two R1, two R2 etc, when the project is looked at as a whole. However, each component name is unique on its own particular board. Therefore you must make sure that you match the parts list with the relevant circuit board.

Main Board or BRD A

Resistors

5% 1/4W carbon or better.

75R	R2, R4, R6, R8, R24
1K	R15, R13, R23
1K5	R16, R18, R20, R14
4K7	R27
6K2	R1, R3, R5, R7
22K	R17, R19, R22, R21, R26
47K	R29, R28, R30, R25
1M	R9, R10, R11, R12

100K X 4, 5 pin SIL pack	RN1, RN3
100K X 8, 9 pin SIL pack	RN2

Capacitors

33pF low-K ceramic plate	C23
100nF axial ML ceramic	C2, C1, C8, C6, C5, C4, C3, C28, C29, C26, C27, C7
1n0, 100V polyester box	C22, C21
2n2, 100V polyester box	C16, C15, C18, C17
3n3, 100V polyester box	C13, C14, C20, C19
4n7, 100V polyester box	C11, C12
15nF, 100V polyester	C10, C9
2u2, 63V electrolytic	C25, C24

Integrated Circuits

TL072 dual Bi-FET op-amp	U1, U2, U3, U4, U5, U6

Pots

100K linear Spectrol 248	VR1, VR2, VR3, VR4
Oakley-Spectrol pot brackets	VR1, VR2, VR3, VR4

Miscellaneous

4-way 0.156" MTA header	PWR
5-way 0.1" MTA header	SOCKETS
Leaded Ferrite beads	L1, 2

Auxiliary Board or BRD B

5% 1/4W carbon or better.

75R	R2, R8, R6, R4
1K5	R19, R17, R13, R15
6K2	R1, R7, R5, R3
22K	R20, R18, R14, R16
1M	R10, R9, R11, R12

Capacitors

100nF axial ML ceramic	C6, C8, C1, C2, C7, C3, C4, C5
1n5, 100V polyester box	C15, C16, C19, C20
2n2, 100V polyester box	C17, C18
3n3, 100V polyester box	C14, C13
4n7, 100V polyester box	C12, C11
10n, 100V polyester box	C9, C10

Integrated Circuits

TL072 dual Bi-FET op-amp	U1, U2, U3, U4

Pots

100K linear Spectrol 248	VR1, VR2, VR3, VR4
Oakley-Spectrol pot brackets	VR1, VR2, VR3, VR4

Socket Board

Switchcraft 112APC 1/4" sockets	INPUT, MIX-OUT, OUT-L, OUT-R

5-way 0.1" MTA header	SOCKETS	

Interconnections

5-way 0.1" MTA housing	Socket connections	2 off
12-way 0.1" jumper	Board 1 to Board B connection	1 off

Around 1 m of insulated multistrand wire (26awg)

Populating the Main and Auxiliary Circuit Boards

The Octal Resonator PCB set is flashed with solder around the pads. This helps the soldering process and keeps the board solderable for many years. This flashing is with solder that contains lead. You should therefore wash your hands after handling the boards and do not place the boards in your mouth. It is also recommended that for best results this board is soldered with lead-tin solder.

Oakley Sound are not responsible for any accidents caused whilst working on these boards. It is up to you to use your board responsibly and sensibly.

Occasionally people have not been able to get their Oakley projects to work first time. Some times the boards will end up back with me so that I can get them to work. To date this has happened only a few times across the whole range of Oakley PCBs. The most common error was parts inserted into the wrong holes. Please double check every part before you solder any part into place. Desoldering parts on a double sided board is a skill that takes a while to master properly.

If you have put a component in the wrong place, then the best thing to do is to snip the component's lead off at the board surface. Then using the soldering iron and a small screwdriver prize the remaining bit of the leg out of the hole. Use wick or a good solder pump to remove the solder from the hole. Filling the hole with fresh solder will actually make the hole easier to suck clean!

For construction of the PCB I use water washable flux in solder. The quality of results is remarkable. In Europe, Farnell sell Multicore's Hydro-X, a very good value water based product. You must wash the PCB at least once an hour while building. Wash the board in warm water on both sides, and use a soft nail brush or washing up brush to make sure all of the flux is removed. Make sure the board is dry before you continue to work on it or power it up. It sounds like a bit of a hassle, but the end result is worth it. You will end up with bright sparkling PCBs with no mess, and no fear of moisture build up which afflicts rosin based flux. Most components can be washed in water, but **do not** wash a board with any trimmers, switches or pots on it. These can be soldered in after the final wash with conventional solder or the better new type of 'no-clean' solder.

I have found that if you are using a very hot soldering iron it is possible to run your iron so hot as to boil the flux in the 'water washable flux' solder. This is not a good idea as it can create bubbles in the solder. If you prefer to have a fixed temperature iron, then it is best to get a

18W one for this purpose. I use an ordinary Antex 25W iron with a Variac power supply running at 205V. This seems to work well for me. Overseas builders will obviously

All resistors should be flat against the board surface before soldering. It is a good idea to use a 'lead bender' to preform the leads before putting them into their places. I use my fingers to do this job, but there are special tools available too. Once the part is in its holes, bend the leads that stick out the bottom outwards to hold the part in place. This is called 'cinching'. Solder from the bottom of the board, applying the solder so that the hole is filled with enough to spare to make a small cone around the wire lead. Don't put too much solder on, and don't put too little on either. Clip the leads off with a pair of side cutters, trim level with the top of the little cone of solder.

Once all the resistors have been soldered, check them ALL again. Make sure they are all soldered and make sure the right values are in the right place.

The resistor networks are thin coloured devices with all their legs arranged in a line. They need to put in the correct way. Pin 1 is the common connection to all the internal resistors. This is normally marked with a white spot at one end. All the resistor networks should be fitted into the board so that the white spot corresponds to the square pad on the PCB. Failure to do this will result in the mix amplifiers not working. Removal of these devices without destroying it or the board takes great skill.

The axial multilayer ceramics can be treated like resistors. Simply bend their legs to fit the 7.5mm (0.3") spacing holes.

The polyester film capacitors are like little coloured boxes. Push the part into place up to the board's surface. Little lugs on the underside of the capacitor will leave enough of an air gap for the water wash to work. Cinch and solder the leads as you would resistors.

IC sockets are to be recommended for the dual in line chips, especially if this is your first electronics project. Make sure, if you need to wash your board, that you get water in and around these sockets. Also, make sure that any water drops left between the pins of the sockets are fully dried up before switching the board on. The 78L05 doesn't need a socket.

The smaller electrolytic capacitors are very often supplied with 0.1" lead spacing. My hole spacing is 0.2". This means that the underside of these radial capacitors will not go flat onto the board. This is deliberate, so don't force the part in too hard. The capacitors will be happy at around 0.2" above the board, with the legs slightly splayed. Sometimes you will get electrolytic capacitors supplied with their legs preformed for 0.2" (5mm) insertion. This is fine, just push them in until they stop. Cinch and solder as before. Make sure you get them in the right way. Electrolytic capacitors are polarised, and may explode if put in the wrong way. No joke. Oddly, the PCB legend marks the positive side with a '+', although most capacitors have the '-' marked with a stripe. Obviously, the side marked with a '-' must go in the opposite hole to the one marked with the '+' sign. Most capacitors usually have a long lead to depict the positive end as well.

The 0.1" headers are fitted on the main board so that pin 1 is towards the right hand side of the board. The clip on the side of the header will match with the picture on the PCB legend.

I would make the main circuit board in the following order: resistors, IC sockets, non-polar capacitors, electrolytic capacitors, and the headers. Then the final water wash.

For the auxiliary board you also need to solder on the 12-way jumper that joins the two larger boards together. Solder the interconnect so that it sticks upwards out of the auxiliary board. Do not yet connect the other end to the main board. You may need to trim of any excess wire from the underside of the board.

Do not fit the pots or LEDs at this stage. The mounting of the pots and LEDs requires special attention. This will be covered later in this User Guide.

Populating the Socket Board

You have one socket board to populate and the method is a little unusual.

On the board the first things to solder is the 5-way header. This is fitted to the **BOTTOM** of the board and are soldered from the top side. This is obviously opposite to what you are normally used to. The legending is on the top of the board, and the bottom of the board is marked as such in copper on the underside.

Fit both the headers so that pin 1 is the square pin. The friction lock on the header should correspond to the legend on the top of the board, ie. the opposite side.

The sockets will be fitted on the top of the board, and therefore be soldered on the bottom of the board. You may well find your own way of soldering the sockets, but the way I do it is as follows:

Fit all your sockets into the board. The bevel edge should align with the picture on the board legending. Do not solder them at this stage. Take your front panel and align this over the sockets.

Now carefully place your front panel with PCB and sockets upside down onto your bench [or kitchen table!]. The holes where the sockets will be should hang over the edge of the bench so that the sockets aren't forced back up through the holes. You'll also probably need a small counter weight to stop the panel from falling over the edge. Now allowing the PCB to rest flat on top of the sockets, you can begin to solder all the pins to the board.

Those of you who have built older Oakley modules will be stunned how easy this was compared with the making of wire frames done previously.

Mounting the Pots

NOTE: This procedure is rather different to that of the Omeg/Piher pots you may have used on older Oakley boards.

The first thing to do is to check your pot values. Spectrol do not make it that easy to spot pot values. Your pot kit should contain:

Value	Marked as	Quantity	Location
100K linear	M248 100K M	4 off	BRD A
100K linear	M248 100K M	4 off	BRD B

Fit the pot brackets to the 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 you should have all eight pots fitted into their pot brackets. Let us now fit these assemblies into the PCBs. Do the smaller auxiliary board first as you may need to get access to the bottom of the board when it is fitted to the panel.

Doing one pot at a time, fit each pot and bracket into the appropriate holes in the PCBs. 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 all four 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. If it does, then I mount the board up proper. You need to add the washer between the outside of the panel and the final nut. Again, do not over tighten and be careful not to scratch your panel.

Now repeat the procedure for the main board.

The pots shafts of the 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 a cotton wool bud.

Connections

The first thing we will do is to connect the two larger boards together. This is done with the 12-way jumper interconnect which has already been soldered in at one end into the auxiliary board. Gently bend the jumper so that the tinned ends can be teased into the correct location on the main board. Now solder the jumper into place from the underside of the main board. Snip off any excess leads poking through.

If you are using the recommended MTA interconnections joining the sockets to the main board will be very easy indeed. All the wiring between the sockets and the main board is done with one 5-way jumper. Here you will be using either the MTA system or the slower, but cheaper, Molex system.

Make up the five way jumper. This should be made from wires 85 mm long. Make sure you get pin 1 going to pin 1 on the other housing, pin 2 to pin 2, etc. This cable will connect to the headers called SOCKETS on the Main board and the socket board.

The Molex Alternative to MTA

For those of you who want to use the cheaper Molex system, the following information may be useful:

A quick note on the female plugs; these are sometimes called housings, since they aren't plugs themselves but merely housings for the individual crimp terminals.

Terminals have to be bought in packs of one hundreds, but this is OK, because they are not expensive. These are normally designed to be crimped but they can be easily soldered with care.

Make each wire the correct length. I normally strip back the wire by just 2 to 3mm. Place all the bare wire into the crimp on a heatproof surface. I use 12mm MDF board to protect my bench top, which although not at all burn proof will take plenty of heat from a soldering iron without major damage. Rest a pair of pliers on top of the wire to hold it in place. Slip the crimp under the wire, so that the wire's insulation butts up to the edge of the terminal. Then solder in place. Sometimes I find I need to gently squash the crimp part of the terminal so that it will fit into the housing. This is easier to do before you solder it, although it can be done after with care.

Do not use the water washable flux solder in this application. You must use either good old fashioned ersin or rosin flux based solders or the newer so called 'no-clean' types. I actually prefer the rosin based ones for this because I find they flow better. Once you have soldered it, wait a bit for it to cool, and then push it into the housing until it clicks. If it doesn't go in, then take it out and bend the crimp slightly backwards. Now try again.

Power Connections

The 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

The earth connection on the PCB has been provided to allow the ground tags of the jack sockets to be connected to the powers supply ground without using the modules 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.

Front Panel

At the rear of this user guide I have included a 1:1 drawing of the suggested front panel layout. Actual panels can be obtained from Schaeffer of Berlin, Germany, or Front Panel Express in the US. The cost is about £25 per panel. All you need to do is e-mail the appropriate fpd file that is found on the Octal Resonator web page on my site to Schaeffer or FPE, and they do the rest.

The panel is black with white **engraved** legending. The panel itself is made from 3mm thick anodised aluminium. The fpd panel can be edited with the Frontplatten Designer program available on the Schaeffer or FPE web site.

The pot spacing on the PCB is equivalent to the vertical spacing of the MOTM series of modules.

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

Apply power to the unit making sure you are applying the power correctly. Reversal of the power’s polarity will usually destroy all the op-amps in an instant.

If you can monitor current, check that it doesn’t exceed 40mA or so from each rail. If you can’t check current directly, check with your finger that none of the op-amps are getting warm.

Now connect an audio signal of some sort to the INPUT socket, any will do, but a simple sawtooth wave in the upper bass registers is quite sufficient. Listen to the audio output from the main ALL OUT socket. With all the pots at their minimum values the output will be more or less the same as the input but a lot quieter. Unlike a filter bank it will not be silent. Now its time to play with the controls.

With all the other controls to their minimum setting, sweep the 250Hz pot clockwise. Do you hear the an increase in volume and a woody tone to the output? If not, you may have a problem and its worth checking the interconnections again.

If this pot appears to be working, try the other pots, one at a time. Each should add its own quality to the sound. The further down the module the pot is the higher the frequency operation. So the highest pots on the module will introduce a woody nature to the sound, while the lower pots will introduce a ringing quality.

The action of each of the pots as they are turned should be smooth, although the final section before reaching the maximum value will be more dramatic.

Now check the output of the LEFT OUT socket. This should give you the output from the left hand filter segments only. The right hand pots should have no effect on the sound. Now check the RIGHT OUT socket’s output. This time this should only reflect the output of the right hand resonators.

Final Comments

I hope you enjoy building the Oakley Octal Resonator. If you cannot get your project to work, do get in touch with me, and I will see what I can do. Alternatively try the Oakley Synths mailing list on Yahoo-Groups. Sometimes, it can be the simplest things that can lay out a project.

I do offer a get-you-working service. Send your completed non-working module back to me and I can fix it for you. I will inspect it for free and will report back to you with my recommendations. If you would like me to repair the module I will charge 15UKP for every hour of work. Most repairs take under this. You will also have to pay for the postage both ways, and for any replacement parts needed. Make sure any returned goods are wrapped carefully and please include a full description of the fault. If you are sending the item from outside the EU, then be sure to say on the customs label 'item being sent for repair only'.

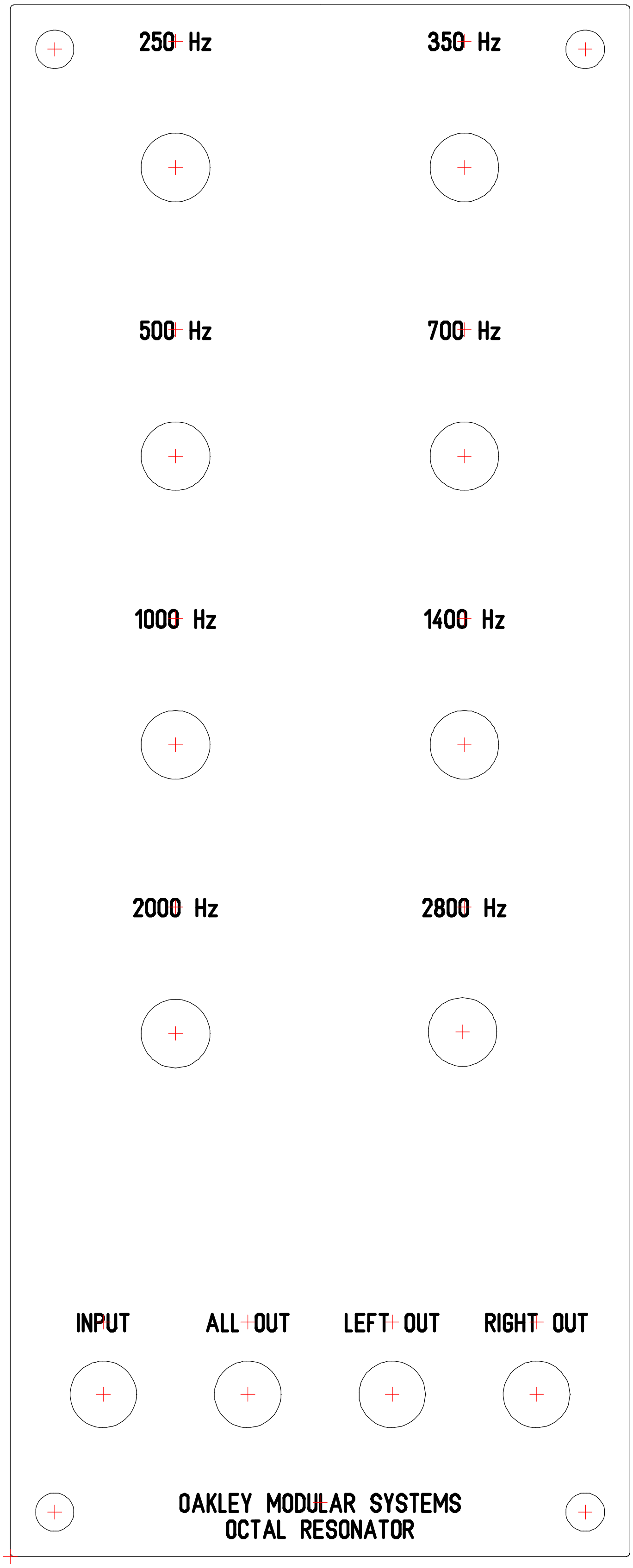
Occasionally, there may be an error in the parts list. I have checked the documentation again and again, but experience has taught me to expect some little error to creep past. The schematic is always the correct version, since the parts list is taken from the schematic. So if there is any problem, use the schematic as the guide. If you do notice any error, please get in touch.

Please further any comments and questions back to me, your suggestions really do count. If you have any suggestions for new projects, feel free to contact me. You can e-mail, write or telephone me. If you telephone then it is best to do this on Monday to Friday, between 9 am and 6 pm, British time.

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, Oakley-Synths and MOTM mailing lists.

Tony Allgood

June 2005


250 Hz
350 Hz
500 Hz
700 Hz
1000 Hz
1400 Hz
2000 Hz
2800 Hz
INPUT
ALL OUT
LEFT OUT
RIGHT OUT
OAKLEY MODULAR SYSTEMS
OCTAL RESONATOR