Oakley Sound Systems

Sample & Hold with Slew Limiter
PCB issue 2

CV and Audio Processor

User's Guide

V2.3.1

Tony Allgood
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

The Oakley Sample and Slew module is a true two stage module that features both a high quality sample and hold circuit and a separate slew generator.

The job of a Sample and Hold in a modular synthesiser is to memorise the voltage applied to it on command of a clock signal. The Oakley Sample and Hold circuit has two modes of operation.

The first is the 'sample' mode. The clock signal is any signal input you like, but it is traditional to use a gate type pulse. If the clock signal is inserted into the 'Clock [rising]' input, then when the signal rises over 2V or so, the command is given to hold. The ‘hold output’ will then remain fixed no matter what the ‘hold input’ signal is now doing. That is until the command to hold is given again. Note, if the clock signal is connected to the 'Clock [falling]', then the hold command is given when the clock input falls below 2V or so.

The second mode is 'track'. This functions like the so-called 'sample and hold' module on the Korg MS-20. Here, the hold output actually follows the input signal slavishly, but only when the clock input is below 2V for the clock rising input (or above 2V for the clock falling input). But when the command to hold is given, the 'hold output' is fixed and unchanging.

The Restriction pot affects the way the hold output moves with respect to its input and the previously held output level. With the pot fully off the unit behaves just like any other S/H, but at full, the output is restricted to move only a little after each new sample. It'll still have the same range as the input, but will need more hold commands to get there.

The Slew generator can be accessed separately from the S/H section, but is normalised to the output of the S/H when it has no input connected. The slew generator smooths any sudden changes in the input signal. Thus a square wave becomes more like a triangle, and keyboard CVs start sliding between notes.

The 'clock' LED lights when the clock input is active. The output LED is a bi-colour LED that gives a visual indication of the hold output voltage.

The PCB

The Sample/Slew module features a two PCB set. One board called the ‘main board’ carries the electronics. The other board carries the sockets. The main board holds the single power header.

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 two MTA solderless or Molex connections. 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 another manufacturer’s 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 the two control pots on the PCB. 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 arranged on a vertical grid of 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 1U wide panel.

Two LEDs are board mounted for easier construction and only the switch need to be wired up with individual connections.

The main board size is 105 x 74 mm.

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 appreciable 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 25 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 main parts of the circuit are split into four on the circuit diagram.

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 C15 and C8 for the positive rail, and C19 and C9 for the negative rail.

A special filtered version of the positive rail is created by R25, C18 and C20 for the noisy logic lines used by the gate input circuitry and the gate LED. This keeps any gate transitions from disturbing the quiet analogue lines and preventing unwanted clicks on the audio outputs.

Additional decoupling is also provided elsewhere on the board by four 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.

The power supplies to each of the op-amp ICs are shown separately from the main schematic to avoid cluttering the diagram.

The sample and hold section is that part built around U2, the LF198 or LF398. Essentially, the LF398 is a ‘one chip solution’ to sample and hold. However, to use this device in a modular synthesiser we need to add a few extras. U1a (pins 1,2 & 3) acts as a voltage follower or buffer to the input signal. This circuit block does not amplify the signal, but merely copies what it ‘sees’ on the input and presents that voltage at its output. It acts to isolate the input signal from anything we do to the output signal. R4 limits the current if the input voltage exceeds the supply voltage for whatever reason. R16 holds the input gently low should there be no input connection, and acts a protection against static discharge too.

The output of U1a feeds the restrict pot which then goes to the sample and hold chip itself. The sample and hold chip contains the circuitry to memorise the input voltage on command of the clock signal. When the clock signal is high, any input voltage is placed across C13. When the clock signal falls, the capacitor is isolated from the input and the voltage left across C13 is held there, frozen in time until the clock signal rises again.

The voltage across C13 will actually start to droop the moment the clock signal goes low. But this droop is pretty negligible and isn’t normally noticeable in most applications. If you are using the S/H module to control a VCO’s pitch you will hear a slight fall (or rise) over time. The PCB and circuitry has been designed to minimise this droop but you can never get rid of it entirely with analogue sample and holds.

The voltage across C13 is sniffed, but not stolen, by the output stage of the LF398. The output of which is then passed onto the other half of U1. This circuit forms another follower circuit that is configured as a line driver. C12 and R21 allow the op-amp to drive long distances of cable without spurious oscillations.

The restrict pot controls the ratio of input signal to sampled output signal that is fed back into the sample and hold chip. With the knob turned fully anti-clockwise, the pot is wired so that the output of the input buffer is fed straight into the S/H chip. This is the traditional sample and hold procedure. With the pot turned fully clockwise the wiper of the pot carries a filtered and slightly smaller version of the sampled output back into the S/H chip. This essentially restricts the *change* in output level of the S/H. When the unit is sampling, with the restrict pot fully clockwise, the sampled output will only rise (or fall) slightly on each successive sample pulse. Simply put, it's a way of reducing the size of the jumps the S/H module makes without affecting the *overall* output signal amplitude. Let's have a look at an example to see this in action:

Consider that a steady 1V is connected to the input of the S/H module. This is then sampled by applying a brief pulse to the clock input. The output of the S/H module now reads 1V, and it will stay there for a long time. Now set the input voltage to 4V. With the restrict pot at the minimum, on the next sample clock the output will rise to 4V pretty much the instant the clock signal arrives. But if you have had the restrict pot set at maximum, the output would have only risen a fraction of a volt when the second clock pulse arrived. Hit the S/H module with another clock pulse, and the output rises a little bit more. In fact, you can keep on hitting it with more clock pulses, and each time the output will try and get closer to the input voltage. Eventually, it will more or less get there, but the biggest jumps will be at the start.

In practice the actual events are a little more complex. The LF398 produces some odd side effects when initiating the sample and hold functions. The output of the LF398 pulses high and low when a command is given. In an ideal world this shouldn't happen, and although these pulses happen for a very small time [they last only 2uS or so], they are unwanted. With the restrict pot at its minimum value, these pulses will not manifest themselves as being a nuisance in most applications. The only time they become audible is if you are clocking the S/H module at audible frequencies and listening directly to the output. However, in restrict mode, they do become more apparent. This is because in this mode, the S/H chip is actually sampling its own output, or at least a fraction of it. This means that any stray blips are then recycled and put back into the IC. The worst case scenario is at full restriction; for each successive clock pulse, the output of the S/H module flips by +/-50mV even if the module has a steady state input signal. If you were driving a VCO with its 1V/octave input this would be equivalent to over half a semitone of error.

But is this a problem? Not really, the whole idea of the restrict pot was to 'restrict' the movement of the output with a white or other random noise input and not a steady state input. Thus the output was going to be changing continually anyhow. I'm pretty sure most of you will find a musical use for it.

The clock control circuitry is shown in the upper left of the schematic. Q3 acts as a simple logic invertor and acts on the 'clock [falling]' input socket. Basically, a logic-1 is turned into a logic-0 and vice versa. To put that in voltage terms, a 0V input is turned into a +15V one, and a +15V input is turned into a 0V one. Any voltage over 3V or so is counted as a logic-1. Negative voltages are tolerated, and are counted as logic-0. D3 does the job of clamping any large negative voltages so that Q3 isn't harmed.

The output of this invertor is fed to the normally closed (NC) lug of the 'clock [rising]' input. The signal lug of which goes straight into the main part of the clock control circuitry. Q2 and associating circuitry act as another logic invertor. The use of a transistor based circuit makes this input very rigged against abuse of the signal kind. D1 protects the junction of Q2 from excessive negative voltages.

The inverted output of Q2 then goes to two schmitt NAND gates based around half of U5 which contains four such gates. These pair of gates essentially act as further invertors, but they also ‘square’ up the signal too. The schmitt function of these type of gates will make sure that any slow edges [slow rise and fall times on the clock inputs] are made faster. In other words, if the output of Q2 takes a longish time to rise from 0V to 15V, then these gates will speed up the transition. The LF398 requires nice fast edges to work well.

The output of U5d (pins 12, 13 & 11) controls the clock LED. When it goes high the LED is turned on via Q1. The output also goes to SW1 and when switched in will control the S/H chip in the so-called ‘track’ mode. You can consider the output of U5d at pin 11 as a sharpened up version of the clock input.

The output U5a (pins 1, 2 & 3) drives a passive differentiator. This circuitry based around C17 and R26 create a sharp pulse every time the output of U5a changes state. This pulse would normally be positive going for every transition from 0 to 1, and negative going for 1 to 0. We need only the rising edges, so we clamp the negative ones with R28 and D2. Pin 5 of U5b, the input to another NAND gate, therefore sees only the fast moving positive spikes. The output of this gate, at pin 4, is thus a 20uS long sharpened pulse going from +15V to 0V and back up to +15V again. U5c (pins 8, 9 & 10) inverts this to give a 20uS pulse positive going pulse. It is this pulse that is used to create the sample command to the S/H chip in ‘sample’ mode.

So in reality the sample mode, as opposed to the pure track mode, is actually just a very short lived ‘track’ mode.

The slew generator is fairly traditional stuff and is based around dual op-amp U4. The first half of U4 acts as a simple voltage follower, or buffer, like we saw in the sample and hold section. It has additional components around it, C14 and R20, to allow it to drive high capacitances without spurious oscillations. But the main action of this circuit is to isolate the input signal from the affects of loading from C1. C1 is charged up a rate determined by both R13 and the ‘slew’ pot. The higher the resistance that feeds C1, the longer it takes to charge up, thus increasing the slew rate. The voltage across C1 will rise and fall in an exponential manner. The other half of U4 is configured as another line driver, just like in the S/H section.

The input and output of the slew generator are shown essentially separate from the rest of the circuitry in the module. However, when the socket board is fitted the input to the slew generator is normalised to the output of the S/H via the input socket’s NC lug.

U3 is solely there to act as a LED driver. The first half of U3 is yet another follower to protect the S/H output from the largish current required by the LED itself. R24 and R19 act as a 3:1 attenuator to isolate the S/H further.

R17 sets the sensitivity of the LED driver. The current through the LED is configured therefore to be -3 mA/V. That means a voltage of 4V, which is about the maximum voltage to be seen at pin 1 of U3, will give an LED current of -12 mA. In conjunction with the attenuator at the input of U3a, the overall sensitivity of the whole driver is approximately 1 mA/V. The O/P LED should be a bicolour type so that it can represent both negative and positive signal levels.

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

Resistors 1/4W, 5% or better.

10R	R25
22R	R10, R23
75R	R21, R11, R9, R20
100R	R6, R8, R28
330R	R13, R17
1K	R5, R33
1K8	R2
2K2	R22, R12
10K	R29, R7, R4, R18, R14, R19
22K	R24, R34
100K	R1, R3, R27, R30, R26, R32, R31
470K	R16, R15

Capacitors

33p low-K ceramic	C12, C14, C4
100p low-K ceramic	C5
220p low-K ceramic	C17
10nF, 63V polypropylene	C13
4n7,63V polyester	C2
100nF axial multi layer ceramic	C8, C18, C9, C7, C11, C10, C6
2u2, 63V electrolytic	C15, C19
2u2, 63V polyester	C1
22u, 35V electrolytic	C20, C3, C16

Semiconductors

1N4148 silicon signal diode	D1, D2, D3

CD4093 quad schmitt NAND gate	U5
BC549 NPN transistor	Q2, Q1, Q3
LF398N or NE5537N	U2
TL072CN bi-FET op-amp	U1, U3, U4

Other

25K linear single gang pot	RESTRICT
100K log single gang pot	SLEW
Oakley/Spectrol pot brackets	2 off for the above
One SPDT on-on switch	Mode

4-way 0.156” MTA header	PWR	1 off
5-way 0.1” MTA header	ANALOG (Main PCB and socket PCB)	2 off
5-way 0.1” MTA housing	Analogue cable	2 off
4-way 0.1” MTA header	LOGIC (Main PCB and socket PCB)	2 off
4-way 0.1” MTA housing	Logic cable	2 off
Leaded Ferrite beads	L1, 2	2 off
Green LED clip/lens	CLK	1 off
Bicolour LED clip/lens	O/P	1 off
Sockets	Switchcraft APC112	6 off

Around 2 m of insulated multistrand wire (26awg)

Power lead MTA to MTA connector

You may well want to use sockets for the ICs. I would recommend low profile turned pin types as these are the most reliable. You need four 8-pin and one 14-pin DIL sockets.

Populating the Main Circuit Board

Warning:

Oakley PCBs are now supplied with a RoHS compliant finish. This is a high quality finish but does possess slightly different soldering characteristics to the traditional lead based HASL finish. Handle the boards with care, and avoid touching the silver coloured parts since this can cause premature tarnishing of the finish. Shelf life is hard to predict but we recommend soldering in all the components less than one year from when you receive your board.

We 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 with four of these 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.

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 diodes can be treated much like the resistors. However, they must go in the right way. The cathode is marked with a band on the body of the device. This must align with the vertical band on the board. In other words the point of the triangular bit points *towards* the cathode of the diode.

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

IC sockets are to be recommended, 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.

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.

The 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.

For the transistors match the flat side of the device with that shown on the PCB legend. Push the transistor into place but don’t push too far. Leave about 0.2” (5mm) of the leads visible underneath the body of transistor. Turn the board over and cinch the two outer leads on the flip side, you can leave the middle one alone. Now solder the middle pin first, then the other two once the middle one has cooled solid.

Sometimes transistors come with the middle leg preformed away from the other two. This is all right, the part will still fit into the board. However, if I get these parts, I tend to ‘straighten’ the legs out by squashing gently all the three of them flat with a pair of pliers. The flat surface of the pliers’ jaws is parallel to the flat side of the transistor.

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, small non-polar capacitors, diodes, transistors, electrolytic capacitors, and connectors. Then the final water wash.

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 are the headers. These are 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, ie. the opposite side, of the board.

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 one of the boards. 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 probably be stunned how easy this was compared with the making of wire frames done previously.

Mounting the Pots and the LEDs

NOTE: This procedure is rather different to that of the Alpha 16mm pots you may have used on our newer 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**
25K linear	M248 25K M	1 off	Main PCB
100K log	248 J 100K	1 off	Main PCB

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, 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 both pots and snip off any excess wire from the pot's pins at this point.

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.

The Schaeffer panel database uses 6.3mm diameter holes which are designed for Cliplite LED lenses available from Maplin in the UK. Other LED lenses may be used with a suitable sized hole, although you may have alter the procedure below slightly to suit your chosen clips.

The LEDs are able to be soldered directly into the board. Preform the LED legs using a pair of fine nosed pliers, bending the leads at right angles to the body of the LED at a distance of around 8mm from the base of the LED’s package. When the LED is in its final placement the bend should be directly aligned with the holes in the PCB. Now poke the LED through the correct holes on the printed circuit board from the underside of the board but do not solder them yet.

Make sure you get the CLK LED connected the right way if you are not using a bipolar LED. Pin 1, which is the square pad on this board, must be connected to the anode of the LED. The cathode of the LED is normally, but not always, marked with a flattened edge on the base of the package. It doesn’t matter which lead goes into which hole of the LED pad if you have used bipolar LEDs since these can go in any way.

The O/P LED may go in any way around but I normally try to make sure that positive output voltages make the LED go red.

Fit the LED clips if you have them at this point to the panel. Now move the board into place on the front panel. The LEDs dangling loose should be carefully aligned with the LED clips in the panel. It is a little bit fiddly, but because you haven't soldered the LEDs there is a good deal of slack so nothing will get damaged.

Once the LEDs are in place you can then attach the pots to the panel. 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.

You can now solder the LED's legs to connect with the solder pads. Snip off any excess lead length from the two LEDs.

Note that the pot shafts of the two pots will not need cutting to size. They are already at the correct length.

Board Interconnections

If you are using the recommended MTA interconnections this section will be very easy indeed. All the wiring between the sockets and the main board is done with one 4-way jumper and 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 first. This should be made from wires 130 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 ANALOG on each board.

The second lead is a 4-way interconnect. This is made up to be 120 mm long. This should connect the LOGIC headers on the I/O board and the main board.

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.

Switch Connections

The switch is wired up next. Fix the switch to the panel so that the toggle moves up and down. The switch should now have its three terminals in a vertical row which I will call; top, wiper, and bottom. Connect the top terminal to the top pad of MODE. Connect the switch wiper to the middle pad of MODE. The bottom terminal should then wired to the bottom pad of MODE. You can either use flexible insulated multistrand wire for your connections, or use uninsulated solid core wire. The latter is probably the easiest to do.

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 PAN pad 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.

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. The first thing you should notice is that the CLK LED lights up.

If you can monitor current, check that it doesn’t exceed 25mA 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.

You will now need a audio VCO and an LFO of some sort. Some form of audio monitoring will be needed too so you can hear the output of the VCO and Sample and Slew module.

Connect the audio output of the VCO to the SLEW IN socket. A sawtooth wave in the bass registers is best. Listen to the audio output from the SLEW OUT socket. Adjusting the slew pot you should notice the sound getting duller as you turn the pot clockwise. This should sound like simple filtering, which is exactly what it is.

Now connect the SLEW OUTPUT to a CV input of the VCO. Hook up your mixer or amplifier to listen to the output of the VCO. Connect a 1Hz square wave from a suitable LFO module to the SLEW IN socket. You should find that the VCO’s pitch will change in time with the LFO’s output. At low settings of the slew pot, the pitch changes should be fast, giving rise to a two tone siren effect. As you turn the slew pot upwards the effect should become more slurred and less deep.

If all is well so far we can now progress to test the sample and hold section.

Set the RESTRICT pot to its minimum value. Connect your 1Hz square wave from the LFO to the HOLD IN socket. Make sure the switch is set to TRACK. The output LED should be alternating red and green at a speed determined by the LFO. Connect the VCO up again to the SLEW OUTPUT and listen again to the VCO’s output. It should warble as it did before, but this time we are sending the LFO through the sample and hold circuit, thus testing the TRACK mode and the normalised connection to the slew generator.

Now change the LFO waveform to a sine or a triangle wave. Set the SLEW pot to the minimum value. The VCO should now be being modulated up and down at 1Hz or so.

Click the switch down to HOLD and the pitch of the VCO should stop sweeping up and down. It should now hold steady with only perhaps a barely imperceptible droop or rise in pitch over several seconds. Click the switch back to TRACK and the VCO should sweep again. Clicking it back to hold will again stop the modulation and hold the VCO at a new pitch.

If this all works you have now successfully tested the sample and hold core. Now the next thing to test is the clock inputs.

Unplug all the connections. Connect the LFO’s square wave output to the CLOCK RISING input. The clock LED should now flash on and off at a rate determined by the LFO. Do the same for the CLOCK FALLING input and again the clock LED should flash in time with the LFO.

Set the VCO so it is producing around 100Hz, which is a low buzzing or humming sound. Use the sine wave output and connect it to the HOLD IN of the sample and hold module. Set the switch to HOLD. The LFO should now be controlling the sample process and you should see the output LED change colour and intensity every time the clock LED lights up. This is because you are sampling the VCO’s sine wave output at regular points, and getting what looks as if the output is changing randomly over time. If you turn the RESTRICT pot clockwise you should find that the LED changes colour less frequently and less abruptly.

If this is all correct you almost certainly have a working module.

If you have found a fault, then the usual course of action is to try and find the wrong component. It’ll usually be a wrong value resistor, like swapping a 100K resistor for a 100R.

Other things to check are the connections between the socket and main boards. Make sure too that your headers have been fitted the right way.

Final Comments

I hope you enjoy building and using the Oakley Sample and Hold and Slew generator module.

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
April 2005

Last Updated: January 2010