Oakley - Modular

Dual Comparator and Gate Delay
CV and Audio Processor

PCB Issue 1

User’s Guide

V1.2

Tony Allgood B.Eng PGCE
Oakley Sound Systems
PENRITH
CA10 1HR
United Kingdom

Introduction

This is an effective CV and audio processor that features two comparators and a gate delay circuit in a compact 1U panel design.

Dual Comparator

A comparator is a functional circuit block that allows two signals to be compared with one another. The output of the comparator is either high (+5V) or low (-5V) depending on which signal is more positive.

Each comparator has two inputs, input A and input B. Input B is directly controlled by a reversible attenuator that affects the level of the signal fed to the comparator circuit block. A reversible attenuator can not only alter the level but also the polarity of the signal too. With the pot in the positive direction, the output of the comparator will be +5V if the input A is more positive than the attenuated input B. If input B is more positive than input A, then the output is -5V. A LED will light when the output is in the high state.

If no jack is inserted into the input B, then input A will be compared to a fixed voltage. This fixed voltage is controlled by the pot, and can be set to anything between -7V and +7V.

The output of the comparator can be treated as either an audio signal or as gate signal to control envelope generators and sync LFOs.

Dual Gate Delay

If no jack is inserted into one of the input A sockets, then you can use that channel as a gate delay circuit. Inputting a standard gate signal into the 'gate delay input' will cause the output of the comparator to go high after a certain length of time. That length of time is controlled either by the appropriate 'amount' pot directly, or by a voltage present at input B which can then be controlled by the 'amount' pot. The gate may be delayed by around 4ms to around 5 seconds.

Both comparator channels are fed from the same gate delay circuit, so you can simply set up two different delays from one gate. This can be used to create multiple triggers from a single 'note on' event.

Please note: in common with most gate delay circuits the delayed output will immediately drop low when the gate goes low. Thus the gate delay process affects only the start of the note, not the end

Uses

This compact and versatile module can be used for a variety of purposes. These are some examples of its use in a small to medium size modular:

1. To create delayed gate signals for pseudo echo effects

2. To create powerful multiple phase pulse width modulation from triangle or sawtooth VCO/LFO outputs.

3. To create a voltage controlled switch in conjunction with a VCA.

4. Creating hard distortion effects from an audio signal

5. Creating random gate events in conjunction with the Oakley Noise/Filter module.

The PCB

The Dual Comparator module features a two PCB set. 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 an extra PCB and two MTA solderless connections. If you are building this module in the standard MOTM and 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 Systems' 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. These can be mounted away from the PCB, but in the standard build they form part of the mounting process. If you use the specified pots and brackets, the PCB can be held very firmly to the front panel. 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 1U wide panel. Four M3 sized holes are provided on the PCB for supporting the board if you choose to use other methods of mounting.

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

Circuit Description

There are three main circuit blocks within the Dual comparator module. The power supply, the comparators, of which there are two, and the ramp generator. The two identical comparator circuits are shown on page 1 of the schematic. The power supply and ramp generator are shown on page two.

Let us first look at the power supply section on the lower part of page two. Power is initially supplied via the usual four way MTA or Molex connector. 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 C16 for the positive rail, and C17 for the negative rail. Additional decoupling is also provided elsewhere on the board by the other capacitors shown. 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, 5, 6 and 7, that require power. The power supply to each IC is shown separately from the main schematic to avoid cluttering the diagram.

To reduce noise on the main power supply lines a dedicated +5V supply is created with the circuitry based around U4. This is a simple 'three terminal regulator' and provides a stable voltage output for the LEDs and comparator output stages. R37 and C20 effectively isolate the +5V line from the +15V supply from which the regulator takes its power.

The dual comparator and ramp generator circuits require a 'reference' voltage to work correctly. It doesn't have to be hugely stable and the current demands are very low. This is created with one half of a dual op-amplifier, U3b (pins 1, 2, 3) and associating circuitry. Its non inverting pin 'sniffs' the voltage of the +5V supply rail. R11 and R12 set the gain of the amplifier to exactly two, so the output at pin 1 is +10V. R14 and R15 pass the reference voltage to the output board via pins 2 and 3 of the UPPER interconnect. More about this later.

The ramp generator can be seen at the top right hand side of the schematic on page 2. Its job is to generate a rising voltage from -10V to +10V the moment a gate signal is detected. A gate signal is traditionally the control signal derived from a keyboard when any note is pressed. In an Oakley modular it is more likely to come from the gate output of the midiDAC. This is a 'logic' type output that goes to +5V when a midi note-on is detected and drops to zero volts when a note-off is received. You can also use the pulse output of a VCO or LFO to drive the ramp generator's gate input. But the circuit won't work reliably with a slow moving signal such as that from a sine wave output.

Let us first consider what the circuit does with no gate signal applied. The first transistor, Q4, is turned off since there is no base current injected into pin 2. Its collector (pin 1) is being pulled high by R27 so the voltage at the collector is in a high state. Q3's base is directly connected to Q4's collector and so Q3 is turned on. The voltage across C8 is therefore held low by Q3.

When a gate is applied, Q4 will turn on thus drawing current through R27 and pulling the base of Q3 low. Without any base current, Q3 will turn off leaving its collector to float. C8 will then be charged up via R25. The speed at which it charges will determined by the size of both C8 and R25. Its pretty slow, taking around 4 seconds to get up to around 10V.

The moment the gate is removed, Q3 will turn on again dragging the voltage down on C8. R16 reduces the amount of current through Q3 during the discharge process to a respectable level.

U3 is configured as a non-inverting amplifier with a gain of two. However, unlike a normal non-inverting amplifier, this one takes its reference from +10V instead of ground. The output from the amplifier thus goes from -10V, when C8 is fully discharged to around +10V when fully charged.

The ramp generator's output is fed to the normally closed (NC) contacts on both of the INPUT A sockets on the front panel. This means that when a jack plug is not inserted into a one of these sockets, the first input signal to that particular comparator channel is fed from the ramp generator.

Why generate a ramp signal? To create the delayed gate output we need to create a signal that goes high at a determined time after a gate signal is detected. So when the gate arrives the ramp slowly charges up. The comparator's threshold point is set by the front panel pot or signal level into input B. When the ramp and the threshold point are the same the output of the comparator will swing high creating our delayed version of the gate. The moment the gate is removed the ramp signal falls quickly low. The comparator detects the loss of the ramp signal and its output drops accordingly.

Let us see how the comparator part of the circuit actually works. Both sets of comparator circuit are identical and they are both shown on the first page of the schematic. We'll consider only the top one, which is comparator channel 1.

Channel 1 takes its two inputs, INPUT A1 and INPUT B1, from the socket board. However, by clever arrangement of the NC contacts of the input sockets, the inputs themselves can come from different sources. As we have seen, INPUT A1 can come from either the ramp generator or an inserted input jack plug. But INPUT B1 can also be affected from two sources as well. As we would expect an inserted jack can carry a signal into B1. But if there is no jack inserted then a +10V reference is automatically fed to the B1 signal line. This allows us to compare INPUT A1 with a fixed reference voltage whose actual value can adjusted with the front panel pot.

INPUT A1 is firstly attenuated and buffered by U1c (pins 8, 9, 10) and associating circuitry. The signal is attenuated to around 75% of its value by R22 and R22. This protects the op-amp from static damage and also stops any 'naughties' from the op-amp itself. The TL07X series of op-amps will do some horrible things if their inputs get anywhere near the supply rails. Cutting the input down a bit reduces any chance of getting near the supply rails.

A buffer is a circuit that merely 'sniffs' the input voltage and creates a copy of that signal at its output. Buffering the input with the op-amp has double benefits. One, it reduces any loading by the Dual Comparator module. That means that this module merely 'sniffs' at the input signal, ie. it doesn't take any significant current from it. Secondly, it allows the comparator circuit itself to receive a low impedance signal. This helps maintain a good clean switch when the comparator's output changes state.

INPUT B1 goes through a more complicated procedure before reaching the comparator. Its circuit path is made from three op-amps. U1b (pins 5, 6, 7) acts simply as a voltage buffer like in the previous input circuit. Its output signal is split between feeding one end of the LEVEL1 pot and the input to another op-amp circuit, U1d (pins 12, 13, 14).

As we can see this is a different circuit to the one around U1b. This one is an amplifier with a gain of -1. This means that its output is the opposite polarity to the input voltage. The input voltage is sensed by R21, and the current flowing through R21, is matched by an equal current flowing in R20. The op-amp does this because it adheres to the 'golden rules'. Actually, it can only try to adhere to these golden rules.

Now the golden rule in question is that an op-amp with negative feedback must move its output so that its two input pins are both the same voltage. The negative feedback is provided by the resistor R20. So as the input signal tries to inject current into the op-amps inverting (-) pin via R21, the output will move against this by taking that current away through R20.

So a positive input voltage at R21 will lead to the output going negative. The ratio of the resistors will determine the gain of the inverting circuit. Making both the resistors the same value will mean that the gain of the op-amp is -1. That is 2V at the input gives us -2V at pin 14. The inverted output feeds the other end of the LEVEL1 pot.

We now have the two versions of the input signal, B1, placed across the two ends of the level pot. As the wiper moves from one side to the other, it will tap off a proportion of each signal. Consider what will happen with a +10V input signal at B1. This will give +7.5V at the output of U1a and -7.5V at the output of U1b. The wiper of the pot can thus move from +7.5V to -7.5V. In the middle position, both voltages take equal precedence and the voltage at the wiper is zero.

The voltage at the wiper is connected to another buffer amplifier based around U1a (pins 1, 2, 3). So a voltage of +7.5V at the wiper of the pot, will give +7.5V at the op-amp's output. It is this pot controlled voltage that will then be fed to the comparator.

A comparator simply compares two voltages and its output will be determined by which of its inputs is more positive in value. The output can change from a low value, in this case 0V to a high value, in this case +5V. It will produce a high value when its + input is higher in voltage than its - input.

The comparator chip is basically an op-amp that has been designed especially for this one purpose only. In this design we use the LM311. Its a fast comparator and its been around for years now. But it can be a temperamental beastie especially if you don't know what sort of inputs it will see. In a modular synth, our comparator will see fast moving audio signals as well as very slow moving control voltages (CVs). The problem for any comparator is what to do when both inputs are roughly the same. Because of noise, interference and the general jittery nature of any electrical signal, the comparator will struggle to deliver a clean output signal around this transition point. By 'clean' I mean a quick transition from one output state to the other. An unclean transition would some show sign of indecisiveness; an output that would rapidly flip from one state to the other before finally settling down.

So to keep the output clean this circuit incorporates various methods of control. R31 and R39 create hysteresis around the comparator. This means once the comparator flips, it will take a larger input change in the opposite direction to turn it back around again. Think of it a bit like a latch. Once closed, it can't open without a bit of effort. Hysteresis is a form of positive feedback. The input is reinforced with the output to prevent those unwanted bursts of indecisiveness.

Unfortunately, with a high speed device like the LM311, this is not normally enough. We could add a heap more hysteresis, but this makes our device less accurate. All that positive feedback is altering the actual input values we are trying to compare. So we use some AC only hysteresis. This is applied by R38 and C18. This form of hysteresis only acts at the transitions and doesn't affect the longer term DC accuracy.

And if that's not enough, we also use C9. This is a data sheet recommendation and its pretty much essential if you have very slow moving low level inputs to compare. Some LM311 devices are happy without this, but I found that 150pF is a good compromise between speed and stability on this particular PCB layout.

Unlike an ordinary op-amp, the LM311 has a single open collector transistor output. In this way it is more similar to logic gate. R45 supplies the current to drag the output positive when the transistor turns off.

There is some scope for playing with the values around the comparator. There is a trade off here with speed, stability and accuracy. I think I have found a good compromise, but if you do play around and get a faster, more stable circuit, please let me know.

To provide a cleaner output still further, a simple switching transistor circuit, based around Q5, is used. This is then fed to a non-inverting amplifier with a gain of two, and a reference of +5V. This turns our 0 to +5V input signal into a -5V to +5V output signal suitable for use with the rest of the modular. It must be noted that if want a unipolar output, ie. 0 to +5V, then all you need to do is not fit R44.

For a visual indication of output state an LED is provided. Q2 provides the current switching to drive the LED. The LED is driven from the +5V supply to prevent excess switching noise from reaching the sensitive +15V rail. D1 prevents Q2 from being damaged by the negatively going output of U7.

R42 provides the usual protection for the output. This stops the output amplifier from oscillating when driving long cables, and also prevents damage should you accidentally patch two outputs 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. Rapid Electronics, RS Components, Maplin and Farnell, are popular here in the UK. The Dual Comparator Module was designed to be built mainly from parts available from Rapid Electronics. 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 pots are Spectrol 248 conductive plastic types and are held onto the boards with specially made Oakley pot brackets. Two pot brackets are required. These are provided with the 'pot bracket kit' which also contains the two extra nuts required to correctly fit the panel to the boards.

The PCB is another Oakley board to allow you to incorporate axial 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.

The other ceramic capacitors should be 'low-K' ceramic plates. These are sometimes called C0G or NP0 types. The lead spacing is 0.2" or 5mm.

The BC549 devices are discrete low noise NPN transistors. You can replace any BC549 with a BC550. Quite often you see an A, B or C suffix used, eg. BC549C. This letter depicts the gain or grade of the transistor (actually hfe of the device). The Dual Comparator is designed to work with any grade device although I have used BC549C throughout in my prototypes.

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 or LM311N. 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: 9526820 (latest RoHS compliant component). Rapid sell them as part number: 26-4860.

For the suggested layout you'll also need two LEDs. Feel free to use any colour, but I have specified red on the schematic. I use 5mm bipolar LEDs with suitable LED holders. I use bipolar LEDs in my ready made modules and pre-populated options because you can fit them in any way around. This saves time and avoids confusion. If an ordinary LED were to be fitted in reverse it doesn't work and it can damage the LED. For the LED holders Maplin still sell their excellent Cliplite clips.

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 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 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*
Six way housing	1098714	640442-6
Six way header	588-611	640456-6
Five way housing	1098713	640442-5
Five way header	588-600	640456-5
Handtool	589-494	59803-1

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.

Resistors

5% 1/4W carbon or better.

47R	R16
100R	R37
470R	R1, R2
1K	R42, R30, R31, R32, R34, R33, R26
2K2	R15, R14
10K	R45, R48, R13
33K	R47, R23, R6, R21, R20, R44, R46, R49, R10, R9, R8, R19, R50, R43, R35, R36, R11, R27, R24, R18, R12
47K	R29
100K	R22, R5, R3, R38, R7, R17, R4, R40, R28
220K	R39, R41
1M	R25

Capacitors

150pF low-K ceramic plate	C9, C10
10nF, 63V polyester box	C18, C19, C1
100nF axial ML ceramic	C21, C5, C11, C12, C7, C4, C14, C15, C6, C3, C2
1uF, 63V electrolytic	C8
2u2, 63V electrolytic	C16, C17, C13
22uF, 25V electrolytic	C20

Discrete Semiconductors

5mm Red bipolar LED	LED1, LED2 ** See text for fitting procedure **
1N4001 rectifier diode	D4
1N4148 signal diode	D1, D2, D3
BC549 NPN transistor	Q5, Q2, Q1, Q6, Q4, Q3

Integrated Circuits

TL072 dual Bi-FET op-amp	U7, U3
TL074 quad Bi-FET op-amp	U1, U2
LM311 High speed comparator	U5, U6
78L05 5V regulator	U4

Miscellaneous

4-way 0.156" MTA header	PWR	1 off
5-way 0.1" MTA header	Lower (Main PCB and I/O PCB)	2 off
5-way 0.1" MTA housing	Lower cable	2 off
6-way 0.1" MTA header	Upper (Main PCB and I/O PCB), LED	2 off
6-way 0.1" MTA housing	Upper cable	2 off
Leaded Ferrite beads	L1, 2	2 off
Red LED clip/lens	LED	2 off
Sockets	Switchcraft APC112	7 off
50K lin Spectrol 248	LEVEL1, LEVEL2	2 off
Oakley-Spectrol Solder brackets	LEVEL1, LEVEL2	2 off

Around 2 m of insulated multistrand wire (26awg)

Populating the Main Circuit Board

Warning:

Oakley Modular PCBs are now supplied with the RoHS compliant Ni/Au 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 Ni/Au that 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.

These boards can be soldered with either leaded or lead free solder. However, you should be aware that lead solder is toxic. Always wash your hands after handling solder and never put solder, or any products containing solder, in your mouth

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 five 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 leaded solder. The quality of results is remarkable, although you should remember that boards made this way are not RoHS compliant and would fall foul of the law should you decide to sell your unit on a commercial basis. 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.

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 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.

The diodes can be treated much like 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. When all the diodes are in place, double check all are pointing the right way.

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 transistors and the 78L05 are in the same type of packaging and therefore look the same. Make sure you get both types in their correct places. Only the numbers on the side will allow you to tell them apart. Match the flat side of the device with that shown on the PCB legend. Push the transistor or IC 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 parallel to the flat side of the transistor.

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, 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 Boards

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 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 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
50K linear	M248 50K M	2 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.

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.

The pots shafts of the two 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 cotton wool bud.

The LEDs may be able to be soldered directly into the board if its leads are long enough. Preform the LED legs using a pair of fine nosed pliers, bending the leads close (around 3mm) to the body of the LED at right angles. The leads should be just long enough to reach the board when the LEDs are sticking through the panel. 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. Make sure you get the LEDs connected the right way if you are not using bipolar LEDs. Pin 1 which is the square pad must be connected to the cathode of the LED. The cathode 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.

If your LEDs do not have sufficiently long leads to reach to board from the panel hole, then you may have to wire it to the board with some small pieces of insulated wire. Keep the wires as short as possible without being taut. Use a little heatshrink tubing to insulate the LED's leads from rubbing together.

Connections

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 5-way jumper and one 6-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 LOWER on each board.

The second lead is a 6-way interconnect. This is made up to be 110 mm long. This should connect the UPPER headers on the I/O board and the main 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.

I like to use as many different colour wires as possible. It makes wiring easier and looks great. And it also helps to make sure that pin 1 on one header is connected to pin 1 on the other.

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.

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-Apparatebau of Berlin, Germany. The cost is about £20 per panel. All you need to do is e-mail the appropriate fpd file that is found on the Dual Comparator web page on my site to Schaeffer, 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 web site.

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

Final Comments

I hope you enjoy building and using the Oakley Comparator module.

If you have any problems with the construction of this module, the first place for support is the Oakley-Synths Yahoo Group:

http://launch.groups.yahoo.com/group/oakley-synths/

If you still 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 email 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 postal costs, any parts used and my time at £20 per hour. Most faults can be found and fixed within one hour. The minimum charge is £20 plus return postage costs.

Your comments are important to both Oakley Sound and Oakley Modular. However, please do not contact me or Oakley Modular directly with questions about sourcing components or general fault finding. I would love to help but I really don't have the time these days to provide any sort of detailed customer support.

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.

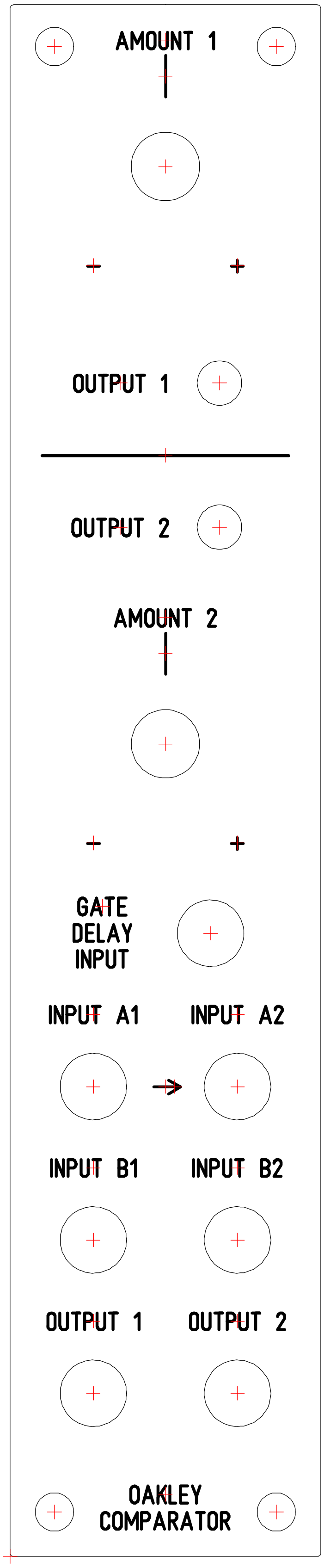
Tony Allgood

Cumbria, UK
May 2007

Version 1.2

Formatted on Open Office 2.0

Errors will always occur in the preparation of a document. Please forward any errors found to me so I can correct them.


AMOUNT 1
OUTPUT 1
OUTPUT 2
AMOUNT 2
GATE
DELAY
INPUT
INPUT A1
INPUT A2
INPUT B1
INPUT B2
OUTPUT 1
OUTPUT 2
OAKLEY
COMPARATOR