Oakley Sound Systems

Little Lag Processor

User's Guide

V1.3

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

e-mail: oakley@techrepairs.freeserve.co.uk

Introduction

This is a very simple but useful little module to introduce 'smoothness' to CVs and audio waveforms. It doesn't just have the usual single 'lag time' pot that some simple filters and lag generators possess, but two separate UP and DOWN controls. The UP control will affect the speed at which the output voltage of the module rises. The DOWN control affects the speed at which it falls. You can do a variety of signal processing tasks with this module. For example:

1. Drive the unit with a gate signal and the module becomes an effective AR envelope generator. The UP control is the attack, and the DOWN is the release.

2. Use it also to process the output of an envelope follower to create more natural filter sweeps when used with a VCF. You can use it to simulate ‘vactrol’ or opto-electronic devices.

A new panel design for the Oakley ‘Envelope Follower and Gate extractor’ (EFG) has incorporated the Little-Lag to create the EFG-Deluxe module. More details about this are found in the EFG User Guide.

You have a choice of linear or logarithmic output slopes. The logarithmic output allows for longish times of lag to be set up easily, roughly 8 seconds maximum lag time. The output moves quickly at first then slows to reach the final value. You get a more natural effect when using this mode. However, the disadvantage is that the unit is not sufficiently accurate to use for portamento applications in ‘log’ mode.

In linear mode, the output rises or falls in a straight line. It is classified in volts per second, as opposed to just time. The amount of time it takes depends on the pots positions AND the voltage change on the input. Generally, for small changes in voltage the linear output will appear to move quicker than its ‘log’ equivalent. The linear mode is very accurate and can be used to create linear portamento.

The difference in the circuitry between the two modes is one resistor and one capacitor.

Circuit Description

The circuit inspiration came from the development of the Filtrex rack mounted filter. The Filtrex required a **simple** lag generator circuit that smoothed off the bumpy output of the envelope follower. And to save on board space I wanted it to be also used as the core of the internal envelope generator. After all, the OMS/MOTM-820 combination proved that you could successfully marry lag and envelope generators together produce one powerful module. However, most simple lag generators have just one control. This affects the up and down times simultaneously. For use in an envelope generator, we have to have separate up and down times. So this required a new type of circuit. Of course, more complex ‘Up-down’ lag generators already exist. Two examples are the MOTM820 and the E-mu modular.

And the Oakley Little-Lag circuit is really quite simple.

The circuit can be divided down into three lumps. The input stage, the up/down control and the output buffer. The signal flow is generally left to right on the circuit diagram, but the all important job of R16 must not be forgotten. This one part is the key to the ‘magic’ of this circuit.

In the log mode circuit the input stage is based around a standard op-amp follower with compensation to allow for capacitive loads. R17, 18 and C11 provide this onerous task. A full explanation of how this actually works is beyond the scope of this user guide, but more information can be found on the Analog Devices website in their Applications Note AN-257.

In the linear mode circuit, the input section acts more like a comparator most of the time. It operates in very high gain while the lag capacitors are charging and discharging. R17 provides a little stability for the op-amp. C11 is omitted in this circuit.

R14 holds the input low with no connection made to the input socket. R15 provides a little protection for any overvoltages.

The output of the input stage is to be found at the junction of R17 and R18. Now without R16 the voltage at this point normally reflects exactly what is happening at pin 3 of the op-amp. But with R16, a proportion of the final output buffer is fed back into the input stage. This produces larger swings in the output voltage of the op-amp which produce the necessary charging and discharging currents for the next stage.

R16 is important, and essential if you are building the linear output version. This simple component provides feedback across all three stages of the design. Thus the input stage will respond not only to the main CV input, but also to the Little-Lag module’s output. This provides relative DC stability, reduces the drops associated with the two diodes, and controls the shape of the charge and discharge curves.

The output of the first stage is split, each path going through a diode and a control pot. The two paths then coming back together at the four charge holding capacitors, C7 to 10. The diodes point in different directions, so current is either allowed out of the input stage through one path, or back into the input stage though the other. Simply put, the capacitors can be charged up via D3 and the UP pot. And discharged through D4 and DOWN. The resistance of UP and DOWN determine the speed of the charge and discharging process. The fastest speed is limited by the internal resistance of the diode (pretty small) and the output capability of the input stage. The slowest speed is determined by the highest resistance of the UP and DOWN pots, namely 1 megohm.

Longer times may be created by using larger values of capacitors. However, the cost of capacitances greater than 1uF may make this idea uneconomic. The capacitors have to be non-polar, since we cannot be sure of what polarity the input voltage will be. If you can guarantee that the input will be only positive, say from an ADSR module or the EFG’s fast output, then you could replace all four capacitors with one larger electrolytic type.

By using four 1uF capacitances in parallel, we have a total of 4uF.

The voltage stored on the capacitors is fed directly to another op-amp based buffer, the third and final stage of the design. This circuit sniffs the voltage without affecting it, and its output is a copy of the voltage on the capacitors. Again, the use of compensated buffers allows you to draw reasonable levels of current from the output without affecting the voltage. In other words, there is no 1K output resistance to worry about.

The two options: Linear or Log?

In the standard log variation R17, along with R16, has been chosen to give the unit quasi linear up and down slopes. You can lower R17 to 4K7 to give even more exponential slopes. This will also give you longer slope times too. However, it will affect the DC accuracy of the unit still further. Errors of the order 250mV will be present. In many cases this should not affect you at all.

To create linear slopes the circuit changes are simple.

Do not use C11, leave this blank. Make R17 a 3M3 carbon or metal film resistor. All other components as they are in the standard unit. Use this option if your intended application is to control the pitch of VCOs. Linear slopes can be used very successfully with the output of the EFG too.

The disadvantage of them is that they tend to be give considerably shorter rise and fall times. In essence the UP and DOWN pots control the slew rate of the unit and don't directly determine the overall time. You might be able to get 5 volts per second from the Little-Lag with the pots at maximum value. This actually turns out to be around 250mV for 1V. So it will take a quarter of a second to jump an octave. That's quite quick.

If you are building two Little-Lags in one 1U panel, then one could be built as linear, the other as a log slope. I find that both units have their use.

It will also be possible to use a switch to change between the two modes. Use the 3M3 resistor in the R17 place on the PCB. Then connect across this a switch and a 22K resistor in series. When the switch is closed you will have a 'log' mode, and when open the unit will operate in 'linear' mode. You can leave C11 omitted.

Components

Most of the parts are easily available from your local parts stockist. I use Rapid Electronics, RS Components, Maplin and Farnell, here in the UK. The Little-Lag was designed to be built solely from parts obtainable from Rapid Electronics and myself only. Rapid's telephone number is 01206 751166. They offer a traditional 'paper' catalogue and take VISA card orders over the telephone.

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.

The pots are Omeg Eco types with matching brackets. You could use any type you want, but not all pots have the same pin spacing. Not a problem, of course, if you are not fitting them to the board. In the UK, CPC, Maplin and Rapid sell the Omeg pots at a very good price. However, getting hold of the pot brackets can be difficult. Although I have recently found that Maplin have once again started selling the brackets.

The resistors are generally ordinary types, but I would go for 1% 0.25W metal film resistors throughout, since these are very cheap nowadays. For the UK builders, then Rapid offer 100 1% metal film resistors for less than 2p each!

For the capacitors, there are no set rules. The two electrolytics (abbreviated to ‘elect’) should be 25V or over, and radially mounted. However, don’t chose too higher voltage either. The higher the working voltage the larger in size the capacitor. A 220V capacitor will be too big to fit on the board. 25V or 35V is a good value to go for.

The pitch spacing of all the non-polar capacitors is now 5mm (0.2”). This may differ from some of the older Oakley boards you have built. For the four big 1uF (1000nF) capacitors I use metalised polyester film types. These come in little plastic boxes with legs that stick out of the bottom. Try to get ones with operating voltages of 63V or 100V. If you get any larger voltage rating than this, there is a good chance they will not fit on the board.

You may be able to get larger capacitance than 1uF for these capacitors. I have seen compact 2.2uF types, and it would be worthwhile getting these if you can afford them. If you building the linear mode ‘Little-Lag’, then the bigger capacitance you have the more useful the slide effect will become.

The ceramic capacitors should be ‘low-K’ ceramic plates. The lead spacing is 0.2” or 5mm. Do not chose cheap and nasty ceramic types, usually ‘high-K’, obtainable from some surplus places.

The IC is a dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP or a N in their part numbers. For example; LF412N. You can use a standard TL072 in place of the LF412 if you wish. Do not use SMD, SM or surface mount packages.

Input and output sockets are not board mounted. You can choose what types of sockets to use. I use good quality metal sockets, like the excellent Switchcraft 112. You will need a small amount of wire to connect them to the board. The standard multicore hook up wire is sufficient, although some may prefer to use screened cable. The PCB does have space to connect the screen of this type of cable.

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

Parts List

The components are grouped into values, the order of the component names is of no particular consequence. The parts list is for the standard ‘log’ output option.

A ‘*’ indicates this component is affected by the linear/log option. For the linear mode output then make R17 a 3M3 5% resistor and do not fit C11.

Resistors: all 5% carbon or better. For example 1%, metal film

22R	R18, 21, 22
75R	R19
4K7	R15, 16
10K	R20
22K	R17*
100K	R14

Capacitors

22uF,25V elect	C13, 15
1000nF, 63V polyester film	C7, 8, 9, 10
100nF, 63V polyester film	C14, 16
22pF ceramic low-K	C12
220pF ceramic low-K	C11*

Discrete Semiconductors

BAT-42 Schottky diode	D3, 4

Integrated Circuits

LF412N or TL072CN dual FET op-amp

Miscellaneous

4-way 0.156" header	PSU
Pot bracket for Eco pots	(2 off)
1M log Eco pot	Up, Down

You may well want to use an 8-pin socket for the IC. I would recommend low profile turned pin types as these are the most reliable.

Building the Little-Lag Module

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 four times across the whole range of Oakley PCBs. The most common error with three 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, so it would be better if you don't make a mistake.

A few years ago Paul Schreiber of SynthTech has won over to water washable flux in solder which he supplies with his MOTM kits. 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 newer 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 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. The two diodes used in this project are BAT-42s. They look like small cylinders made from a blue glass. The band which denotes the cathode is painted on the surface of the device usually in black or white ink.

An IC socket is 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 capacitors are like little blue or red 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 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.

I would make the board in the following order: resistors, diodes, IC socket, small non-polar capacitors, electrolytic capacitors. Then the final water wash. Do not fit the pots at this stage. The mounting of the pots requires special attention. See the next section for more details.

Mounting the Pots

If you are using the recommended Eco pots, then they can support the PCB with specially manufactured pot brackets. You will not need any further support for the board. When constructing the board, fit the pot brackets to the pots by the nuts and washers supplied with the pots. Now fit them into the appropriate holes in the PCB. But only solder the three pins that connect to the pot. **Do not** solder the pot bracket at this stage. When you have soldered all the pots you can fit the board to your front panel. Position the PCB at right angles to the panel, the pot's own pins will hold it fairly rigid for now. Then you can solder each of the brackets. This will give you a very strong support and not stress the pot connections.

The Omeg pots are labelled A, B or C. For example: 100KA or 1MB. Omeg uses the European convention of A = Linear, B = logarithmic and C = Reverse logarithmic. So a 1MB is a 1 megohm log pot.

Connections

The power socket is 0.156" 4-way header in common with rest of the Oakley, Blacet and MOTM modules. Friction lock types are recommended.

Power	***Pin number***
+15V	1
Module 0V	2
Earth/Screen	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 power 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.

If you have used Switchcraft 112 sockets you will see that they have three connections. One is the earth tag on the bevelled edge. One is the signal tag which will be connected to the tip of the jack plug when it is inserted. The third tag is the normalised tag, or NC (normally closed) tag. The NC tag is internally connected to the signal tag when a jack is not connected. This connection is automatically broken when you insert a jack. In the Little-Lag, only the NC lug on the input socket should be connected.

You can use either screened cable or multicore hook-up wire to connect the sockets to the board. For the input socket, connect the signal lug to the pad on the board marked IN. Connect also the NC lug to the IN-G pad. If you want to use screened cable, then use the core of the cable for the signal and the screen for the NC to IN-G connection. On no account connect the NC lug or the screen to the grounding lug on the socket.

For the output socket, connect the signal lug to the OUT pad. With screened cable, you must only connect the screen to the pad marked OUT-G. At the other end near the socket, the screen must be trimmed back and insulated. You can use a bit of heatshrink tube to keep the end from fraying. The NC lug is left unconnected.

The ground tags of each socket can be connected together with solid piece of wire. A single piece of insulated wire can then be used to connect both the tags to the PAN pad on the PCB. Do not connect the ground tags to any other ground. If you are building the dual Little-Lag panel, then connect all four sockets up to the same ground frame, and take your wire to just one of the board's PAN pad.

Keep the wires short but not taut. Colour coding the wires makes it easy to connect, and gives an attractive finish.

At the rear of this user guide I have included a 1:1 drawing of a suggested front panel layout for a 1U dual Little-Lag panel. This layout uses two Little-Lag PCBs mounted vertically with four sockets.

If you are wiring a dual panel, then you need only fit the power connector to the top one. You then simply connect the other two Little-Lag PCBs with insulated wire to the top one. Or you can power both from their own four way power cable.

Another option, already mentioned, is to make the Little-Lag project as part of the EFG module. This is the so called EFG-Deluxe. This 2U panel will give access to all the EFG's outputs and also have room for the Little-Lag. The input of the Little-Lag is normalised to the fast output of the EFG. But this can be overridden by inserting a jack into the Lag input socket.

Actual panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £15 per panel. All you need to do is e-mail the fpd file that is found on the Little-Lag 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 3 mm thick anodised aluminium. The fpd panel can be edited with the Frontplatten Designer program available on the Schaeffer web site.

Testing the Little-Lag

There's not much to this module so it should be very easy to test. Firstly power the unit up, making sure you have the power connected correctly. The unit should not smoke...

Once you have passed the smoke test, make sure no component is getting warm. Gently, but quickly, place your finger on each component to make sure everything is running cool.

The method of testing depends on your hardware. However, let us assume you have a standard VCO, VCF, VCA modular synth in front of you. Set it up so that you have the standard or classic synth patch which plays a note when you use your controller keyboard. Connect a mult'ed version of the gate output of your keyboard into the IN of the Lag module.

Now the output of the Lag can go to the VCF's control voltage input. Pressing a note should make the filter's cut-off frequency rise. The UP control on the Lag module should control the speed at which it rises. Fully counter clockwise should make it jump upwards immediately. Fully clockwise should make it sweep up slowly. It should take about 8 seconds to get there if you have built the standard 'log' version. The DOWN pot should control the downward sweep once you remove your finger of the keyboard. Make sure that the VCA stays open to allow you to hear this. The Little-Lag should be behaving as a standard AR envelope generator.

If all is well, you have a working Little-Lag module. Try using audio as an input and see what affect the Up and Down pots have on the sound. If both pots are moved together you have a simple one pole low pass filter.

Final Comments

I hope you enjoy building and using the Oakley Little-Lag. Please feel free to ask any further questions about construction or setting up. If you cannot get your project to work, do get in touch with me, and I will see what I can do. 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 with £10 and I will fix it for you. You will also have to pay for the postage both ways, and for any replacement parts needed. Make sure you wrap it carefully and include a full description of the fault. It may be easier to just send the populated PCB back since the weight of the front panel can add to the cost of postage.

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. You will be credited on the 'Updates and Mods' page, and you may get a free PCB if its a real howler.

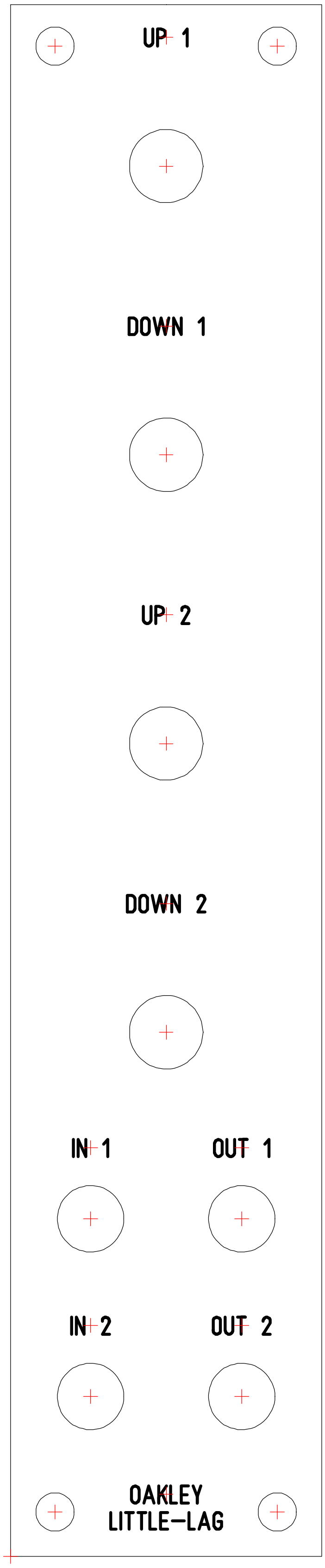
Please further any comments and questions back to me, your suggestions really do count. If you have any suggestions for new projects, feel free to contact me. You can e-mail, write or telephone me. If you telephone then it is best to do this on Monday to Friday, between 9 am and 6 pm, British time.

Last but not least, can I say a big thank you to all of you who helped and inspired me. Thanks also to all those nice people on the 'Oakley-synths' mailing list.

Tony Allgood. July 2002

Version. 1.3

Formatted on Lotus Word Pro


UP 1
DOWN 1
UP 2
DOWN 2
IN 1
OUT 1
IN 2
OUT 2
OAKLEY
LITTLE-LAG