Oakley Sound Systems

Little Low Frequency Oscillator
Board Issue 2

User's Guide

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

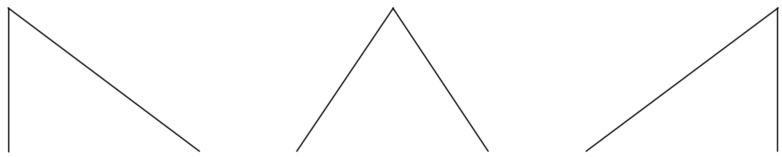
e-mail: oakley@techrepairs.freeserve.co.uk

Introduction

Please make sure you have the issue 2 Little-LFO PCB. It should have the words ‘LFO ISS.2’ written inside the outline for the DG403.

This is a simple but effective low frequency oscillator (LFO) module primarily designed for use in modular synthesisers. Its basic design and the use of PCB mounted pots makes this an ideal starter module for beginners to my modular synthesiser projects.

The design was based around the topology of the Korg MS-20’s LFO. This simple design had three op-amps and two selected FETs. The basic circuit was quite straightforward, but what made it special was the ‘Shape’ control. This enabled the ‘triangle’ and ‘pulse’ outputs to have a variable duty cycle without affecting frequency. For the triangle output, you could vary the rise-time to fall-time ratio. Thus the triangle would be able to go from a saw down waveform, to a saw up waveform. The diagram below shows this more clearly:



The Oakley ‘Little-LFO’ takes the basic topology of the Korg design but improves it with modern components. The selected FETs, the 2SK30A-O, which are difficult to get hold of, have been replaced by a single integrated FET switch, the DG403. This gives enhanced performance with faster edges for the sawtooth waveforms. We can also lose one of the op-amps. The DG403 has two pairs of FET switches inside. The second pair of switches allows us to have a synchronisation function. When a rising edge is applied to the sync input, eg. a gate signal from a midi-CV convertor, the LFO output waveform will be reset to zero momentarily. The LFO can now be used as a simple linear envelope generator.

The second issue is pretty much unaltered from the first one. However there are some differences:

1. The part numbering is different.

2. The I/O socket has now been replaced by individual solder pads for each output.

3. The LED is now controlled by the square wave output. This will give sudden changes in colour as the polarity changes. The old issue allowed the LED to fade to nothing before changing colour. This meant on very slow sweeps, it was possible for the LED to completely out. Both methods have their advantages. I prefer the new version.

4. A space for another timing capacitor and extra solder pads have been made to allow a range switch. This means you can now increase the frequency range of the LFO. Simply by flipping a switch you can go from low frequency to ultra low frequency. Please note: I have not included the switch on the suggested layout due to space restrictions.

5. The old 0.1” Molex connector has now been replaced by the standard Oakley/MOTM 0.156” MTA header.

6. The PCB is slightly bigger to accommodate the changes.

All the parts are easily obtainable, although the PCB mounted pots and pot brackets are available from me should you find any difficulty in getting these.

The PCB

The PCB is small and almost cute at just 6.5 x 4.6 cm in size. It uses double sided copper traces and has through plated holes. It has solder mask both sides for easier soldering, and has component legending on the top side for easier building.

I have provided space for the two control pots on the PCB. These can be mounted away from the PCB, but they do form part of the mounting process, as no holes are provided on the PCB for supporting the board. 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 horizontally, ie. in a 2U wide panel. I have included a suggested front panel layout at the back of this document.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 10 mA per rail, although this can rise considerably if you chose to use the bicolour LED. Power is routed onto the PCB by a four way 0.156" MTA header. You could, of course, wire up the board by soldering on wires directly. The four pins are +15V, ground, earth, -15V. The earth connection allows you to connect the metal front panel to the power supply's ground without it sharing the modules' ground line. More about this later. The connector used is now identical to my other larger module designs.

Circuit Description

The LFO circuit is quite simple, but let's run through the design carefully. Looking at the left of the schematic you can see the four way header, labeled PSU. Power enters the board here, and is immediately filtered by a simple RC networks based around R5 and C4 for the positive rail, and R6 and C2 for the negative rail. Two grounds are provided, one for the circuit itself, and one for the earthing of the jack sockets on the front panel.

There are just two ICs on this PCB, and each requires power. The power supply to each IC is shown separately to avoid cluttering the main circuit diagram.

The circuit is built from three parts. A changeover switch, an integrator and a schmitt trigger. The output from each feeds into the next one in the chain, and then right round again. We will start by looking at each bit in turn.

The first TL072 op-amp forms part of the integrator. Any positive voltage applied to the left of R4, will cause the voltage to fall at the output of the op-amp. The speed at which the voltage falls is controlled by C1 (and C6 if the range switch is set to low) and the size of the voltage applied to R4. If the applied voltage is negative the op-amp's output will rise. It is the integrator's output that will be used as the source for the triangle output.

The DG403 is configured as four electronically controlled switches. They are arranged in pairs, so that when one switch of the pair is closed the other is open. When a switch is closed the signal can pass through pretty much unaffected. The second half of the DG403, on the right, is wired so that the output of the integrator passes straight to the schmitt trigger. This connection can be broken when the sync pulse is applied, but more about this later.

The schmitt trigger is a simple circuit block based around the second half of the TL072 op-amp. It's output is either high at +13V, or low at -13V. If the output of the Schmitt is initially low, it requires +5V at the output of the integrator to make it go high. The integrator will need to produce an output of -5V to make the Schmitt go low again.

To make any oscillator you normally require an output to be fed back into the input. In a standard LFO, the integrator is fed by the output of the schmitt trigger. Thus, a low at the output of the schmitt causes the integrator to rise. When the integrator's output reaches a certain point, the schmitt switches state and the integrator's output falls. The schmitt trigger changes state once again, and the process repeats itself....

In the Little-LFO, the amount of signal applied from the Schmitt trigger to the integrator is controlled by two items. Firstly, the 'Rate' pot. This allows a only a controlled proportional of the schmitt's output voltage to reach the integrator. If the proportion is large, the voltage on R4 is large, and the integrator sweeps fast. If the proportion is small the integrator sweeps slowly. R1 sets the minimum speed. Don't be tempted to lower this value any more to get really slow sweeps. Input errors within the integrator op-amp will take over and your LFO won't oscillate any more. The second control for the speed of the integrator is the first half of the DG403 switch coupled to the 'Shape' pot. The switch is controlled by the schmitt trigger's output. When it is high, the top switch is closed, and the resistance between the top of the pot and the wiper will determine the rate of fall in the integrator output. When it is low, it is the bottom of the pot that is switched in, and this controls the rise of the integrator's output. If the rate pot is central, then both halves of the pot are equal, and the rise and fall times are also equal. If, however, the pot is moved towards one of its ends, then the rise and fall times of the output will be different, but the overall frequency will be roughly the same.

D1 allows only positive voltages to control the DG403. Negative ones will overheat the device, and must be removed from the control pin. The pulse output is taken from this positive only signal. R9 and R10 attenuate the signal to approximately 10V peak. The output impedance is about 1K3.

The triangle output is taken from the integrator's output via R13. The output level, is controlled by the Schmitt trigger and is set at +/-5V approximately.

When a pulse or gate appears at the 'sync' input, the pulse or gate is shaped by C5, R8 and D2 to give a short positive spike. This controls the second half of the DG403. Normally, as I mentioned earlier, this part of the DG403 allows the integrator output to pass straight to the Schmitt trigger. However, when a spike is present at pin 10 of the DG403, two things happen. Firstly, the integrator output is cut off from the schmitt trigger by the opening of the FET switch between pins 5 and 6. R2 will then force the schmitt trigger's input low and the Schmitt's output will be set low. Secondly, the integrator capacitor will be shorted out, by the closing of the FET switch between pins 8 and 9. This will set the integrator's output at zero volts. When the 'sync' spike fades away, the integrator will start to ramp positive due to the negative voltage from the schmitt trigger. If we did not force the Schmitt low, then the integrator may ramp down or up. A good LFO should have a predictable behaviour, and ramping up is the most sensible.

C1 controls the overall range of frequencies produced by the Little-LFO. You may want to try different values for this capacitor. With 100nF, you get about 0.2Hz to about 25 Hz. Lower values of C1 will give you a faster oscillator, although some problems will occur if you try to get frequencies above 500Hz. I tried 1uF, and I got some very slow sweeps of around a minute or so at the bottom end. A simple switch can be made to allow a higher value capacitor, C6, to be added in parallel to C1. S1 and S2 are the solder pads that need to be connected to your chosen switch. When the switch is closed the capacitors add together to get a single larger value capacitor. For C6 I have selected 470nF, this will give a frequency range of 0.04Hz to 4Hz. This gives a reasonable overlap between the two ranges. Either way, the range with the ‘rate’ pot alone is about 100:1.

A bicolour LED may be fitted too. This is simply fixed to a current limited version of the comparator output. R12 controls the maximum current, and hence, brightness, of the LED. The LED will be one colour when the square wave output is positive and another when the output is zero. This corresponds to one colour as the ramp output is rising, and the other colour as the ramp output is falling.

Components

All the parts are easily available form your local parts stockist. I use Maplin, CPC, Rapid RS-Components and Farnell, here in the UK. In North America, companies called Mouser, Newark and Digikey are very popular.

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, Maplin, CPC and Rapid Electronics sell the Omeg pots at a very good price. However, they do not sell the pot brackets. Omeg will also sell direct, but this is only viable for large orders. I am able to supply a pot kit with all three pots and associating brackets.

The dual op-amp is a TL072, but you can use any good dual op-amp.

I would go for 1% 0.25W metal film resistors throughout. But you could use 5% types with no problem in the rest of circuit.

The two electrolytics should be over 16V, except where stated, 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 the non-polar capacitors, C1, 3, 5, (6 is optional), is 7.5mm (0.3”). I think polyester film types are fine for the Little-LFO. I like the open frame Siemens polyester layer types, because they are very compact. They are normally called ‘poly-layer’ and are available in many different working voltages. Use 63V or 100V. But remember the pitch spacing. You could also use the 2222 368 series from BC components. These are metalised polyester types, but again do be sure you get low working voltages. They easily fit into the

7.5mm spaces provided. Around 100 to 150V is best. In the UK, Farnell can supply all the capacitors.

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

Please be careful with the orientation of the polarised capacitors, diodes and the ICs. **Please note:** The TL072 and the DG403 have pin 1 at the **top** left hand. Pin 1 is depicted on the board by a square pad for both ICs and the 0.1" headers.

Paul Schreiber of SynthTech has won me over to water washable flux in solder. The quality of results is remarkable. In Europe, Farnell sell Multicore's Hydro-X, part number: 629-431. This is 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, although, it is probably not a good idea to wash a board with trimmers and pots on. These can be soldered in after the final wash with conventional solder or the better new type of 'no-clean' solder. I use Farnell part number: 904-545. Make sure the board is fully dry before switching it on.

I would make the board in the following order: resistors, IC sockets, small capacitors, electrolytic capacitors. Then the final water wash. Then the pots can be soldered in with 'no clean' or ordinary rosin based solder. See later for more details on mounting the pots.

Finally, if you make a circuit change that makes the circuit better, do tell me so I can pass it on to others. Any updates are added to the current user guide, and posted on the 'Mods' page on my web site.

Parts List

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

Resistors: all 1% metal film

22R	R5, 6
100R	R1
1K	R13
1K8	R12
2K2	R9
4K7	R10
6K8	R4
33K	R2, 3
100K	R7, 8, 11

Capacitors

22uF,25V elect	C2, 4
100n polyester film	C1, 3
1n polyester film	C5
470n polyester film	C6 (optional)

Discrete Semiconductors

1N4148	D1, 2
LED Bicolour	Off board (optional)

Integrated Circuits

TL072 dual FET op-amp
DG403 Dual SPDT analogue FET switch

Miscellaneous

4-way 0.156" header	PSU
Pot bracket for Eco pots	(2 off)
1M lin Eco pot	Shape
10K log Eco pot	Rate

You may well want to use sockets for the ICs. I would recommend low profile turned pin types as these are the most reliable.

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 normally 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 completed the PCB, you can fit it to your front panel. Position the PCB at right angles to the panel. Now 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: 47KA or 100KB. Omeg uses the European convention of A = Linear, B = logarithmic and C = Reverse logarithmic. So a 47KA is a 47K ohm linear pot.

Connections

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

Power	***Pin number***
+15V	1
Module 0V	2
Earth/Screen	3
-15V	4

The SCR 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. 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-LFO, we have no use for the NC lugs.

The ground tags of each socket can be all connected together with solid wire to make a simple frame. A single piece of insulated wire can then be used to connect all the tags to the SCR pad on the PCB. Do not connect the ground tags to any other ground. If you are building the triple LFO panel, then connect all nine sockets up to the same ground frame, and take your wire to just one SCR pad.

At the rear of this user guide I have included a 1:1 drawing of the suggested front panel layout for a triple LFO panel. This layout uses three Little-LFO PCBs with nine sockets and three LEDs. Wiring them up is straightforward enough. Use multistrand hook up wire to connect each socket’s signal lug to the relevant pad on the PCB. There is no need to use screened cable for such short runs. Keep the wires short but not taut. Colour coding the wires makes it easy to connect, and gives an attractive finish. If you do insist on using screened cable then connect the screen to the socket’s ground lug. Leave the screen unconnected at the PCB end. Use a bit of heatshrink tube to keep the end from fraying.

The LEDs are wired between LD1 and 0V pads. I normally wire them up to give red for positive and green for negative.

If you are wiring a triple panel, then you need only fit the power connector to the top one. You then simply connect the other two LFO PCBs with insulated wire to the top one.

Actual panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £25 per panel. All you need to do is e-mail the fpd file that is found on the LFO 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 2.5 mm thick anodised aluminium. The fpd panel can be edited with the Frontplatten Designer program available on the Schaeffer web site. Please note, the mounting holes are not compatible with the MOTM mounting rails. However, it should be a simple matter to alter these as required.

Final Comments

I hope you enjoy building and using the Oakley Little-LFO. 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.

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.

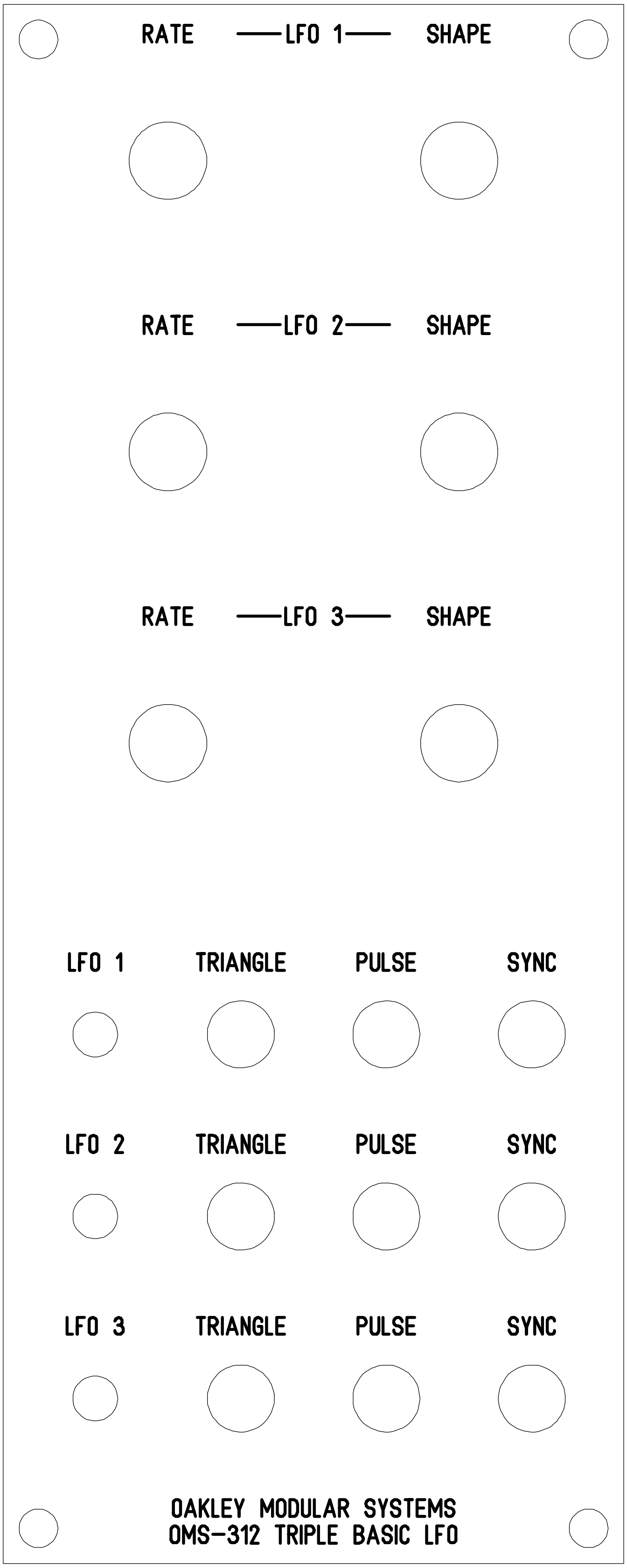
Please further any comments and questions back to me, your suggestions really do count. If you have any suggestions for new projects, feel free to contact me. You can e-mail, write or telephone me. If you telephone then it is best to do this on Monday to Friday, between 9 am and 6 pm, British time.

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, Synth-DIY and MOTM mailing lists.

Tony Allgood. May 2001

Version. 1.01

Formatted on Lotus Word Pro


RATE
LFO 1
SHAPE
RATE
LFO 2
SHAPE
RATE
LFO 3
SHAPE
LFO 1
TRIANGLE
PULSE
SYNC
LFO 2
TRIANGLE
PULSE
SYNC
LFO 3
TRIANGLE
PULSE
SYNC
OAKLEY MODULAR SYSTEMS
OMS-312 TRIPLE BASIC LFO