Oakley Sound Systems

EFG & EFG-Deluxe
Envelope Follower & Gate Extractor

PCB Issue 1

User Guide

V1.03

Tony Allgood B.Eng PGCE
Oakley Sound Systems
CARLISLE
United Kingdom

Introduction

An envelope follower takes an audio input and converts it into a control voltage. This control voltage will rise and fall with the overall volume of the input signal. The louder the input, the bigger the output of envelope follower. The Oakley EFG has two CV outputs available simultaneously. One reacts fast, the other reacts slower. The former is useful for tracking fast moving cymbal and hi-hat patterns. The slower one is more useful in processing single instruments.

Another feature of the Oakley EFG, is the provision of a gate extractor. This part analyses the peaks in the music and creates a gate type signal in time with these peaks. This can be used to trigger envelope generators in time with an external drum loop or click track. Two rotary controls enable you to get clean gate signals from all sorts of input material.

The Oakley EFG comes with a wide range two stage input preamplifier. This allows guitars and other external instruments to be processed simply. The output of this section is available separately for driving other modules if you wish.

And most importantly it has three LEDs. One showing input overload, one for gate status, and one for a visual indication of the CV outputs.

The EFG may be used with the optional extra of the ‘Little-Lag’ project to form the EFG-Deluxe module. This little PCB fits next to the EFG in a new 2U wide panel design. The Little-Lag is a lag generator that can slew control voltages. This allows you to control the speed at which CVs can rise and fall. The EFG-Deluxe normalises the input of the Little-Lag so that you can control the rise and fall times of the EFG’s fast output. This allows you to create more flexible control signals to drive your filters. The classic ‘vactrol’ based auto-wah pedals can be replicated this way with their fast attack and slow decay time. The additional space on the panel also allows the pre-amp output to become available.

Of Pots and Power

There are three main control pots on the PCB. The pots are preamplifier **Gain**, **Threshold** and **Response**. If you use the specified pots and brackets, the PCB can be held firmly to the panel without any additional mounting procedures. The pot spacing is on a 1.625” grid and is the same as the vertical spacing on the MOTM modular synthesiser. The PCB has four mounting holes, one in each corner should you require additional support which you probably won’t.

Three LEDs can be fitted. The board has been laid out to allow the LEDs to be soldered straight onto the board, with the suggested front panel layout.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 30mA. Power is routed onto the PCB by a four way 0.156” Molex type connector. Provision is made for the two ground system as used on all new Oakley modular projects, and is compatible with the MOTM systems. See later for details. This unit will run from a +/-12V supply with a slight reduction in dynamic range.

Circuit Description

Like many complex analogue circuits the EFG circuit can be split up in to little bits. The first bit we will look at is the preamplifier stage.

The pre-amp is built around U1. I have specified the low noise audio op-amp, the OP-275 or NE5532. This is only really necessary if you are using the pre-amp's output for driving other external modules. If you are just using the pre-amp just to boost a signal for use with the EFG, then you can substitute an TL072 for U1.

The preamplifier is a two stage design. The first stage is a non-inverting amplifier whose voltage gain can be varied from 1 to 12. C14 keeps the gain for DC and very low frequency signals at near to one. This prevents any offsets within U1 from being amplified unnecessarily C7 provides a little bit of high frequency roll-off to keep the amplifier stable.

The second stage of the pre-amp is an inverting amplifier. The GAIN pot is used in a slightly offbeat way. It is in both the feedback and the input resistors. This way we can control the gain over a wide range from -0.4 to -10.1. The minus in these numbers shows the inverting properties of the amplifier.

The voltage gain of the two preamplifier stages in tandem can be varied from -0.4 to -122. A gain of 0.4 means that the output of the pre-amp is only 40% of the input level. While a gain of 122 means that the output level is 122 times bigger than the input. In audio circles this would normally defined in dB. This preamplifier will give you a gain from -8dB to +42dB. Because the preamplifier is made from a inverting and non-inverting stage, the overall behaviour is inverting. This means the output is completely out of phase with the input. This shouldn't bother most people in the normal operation of the pre-amp, and doesn't affect the performance of the envelope follower at all.

The suggested layout of the EFG does not have a separate output for the pre-amp. Instead it goes straight into the next part of the follower. So the signal jumps from **OUT1** to **IN2**. The pads are close by on the PCB, so they can be easily linked together with a loop of wire. If you are building the EFG-Deluxe you can utilise these pads more by bringing out the pre-amp to its own front panel socket. See later for more details on this.

IN2 leads straight into some more amplification based around U2a. This is a non-inverting amplifier of around 2. The pre-amp is expected to produce a maximum output of around 5Vp-p in normal use, so U2a boosts this signal up to 10Vp-p. The amplified signal is now full wave rectified by the circuitry based around U2b. Full wave rectification can be described by the mathematical 'absolute' function. In other words, the output of the full wave rectifier (FWR) is always positive. If you present +10V to the input, you will get +10V. But if you present it with -10V you will also get +10V. Likewise, -5V turns into +5V, -3V into +3V. Now if you put an audio signal into this circuit, you will get a series of positive bumps that correspond to the up and downs of the audio signal.

U3a forms a buffer circuit. This configuration, allows the op-amp to drive medium to high capacitive loads without instability. The output of the full wave rectifier is therefore protected by the odd load presented by the next set of circuits.

Now, no real time system can recover envelope information without some disadvantage of some sort. Some systems employ the *peak and droop* method. These are fast to respond to sudden changes in loudness or envelope. They work by simply charging a capacitor from the FWR through a diode. The capacitor is then discharged through a resistor, sometimes variable, causing the stored voltage to droop at a determined rate. However, they are often plagued by ripple. Ripple is the bumps from the FWR creeping through to affect the required output. This tends to manifest itself in a 'buzz' to the output CV. If you increase the discharge resistor, you can reduce the bumps but this tends to not allow the CV to drop quick enough when the sound ends.

Another method involves low pass filtering of the FWR output. This leads to less bumps if the correct filter cut-off frequency is chosen, but does lead to longer attack times. There are more complex ways too. Involving sample and holds and other clever methods. But a decision needed to be made. Some three years ago when I designed my VCF-1 Filter rack, I sat down and compared the many different circuits. I didn't want a complex circuit, I didn't have the board space to do that. So I stuck to either filtering or the *peak and droop* topologies. The one that most excited me was the low pass filter, but only if you got the frequency right. I decided to use a four pole filter with the cut-off frequency at around 33Hz. I chose the best sounding and the most natural with all sound sources. This method is not as quick at responding to fast attack signals as the *peak and droop* but it does react equally to rises and falls in signal level. However, the *peak and droop* wouldn't be forgotten, I needed that for later.

The standard EFG provides two filters, one set at 33Hz for 'slow' response, and the other set at 160Hz for 'fast' response. Both are four pole active types, and both filter outputs are available simultaneously. The 33Hz is probably the best for general music input, while the 160Hz one is better dealing with hi-hat or percussion patterns. More on the uses of this later.

Incidentally, people who have a MOTM VC-Lag or an Oakley Little-Lag module can use the EFG's fast filter output to drive it. In this way, you can replicate *peak and droop* behaviour quite easily should you require. The EFG-Deluxe provides this route as standard.

U4 is a quad op-amp, and I use each stage in identical fashion. Each one is configured as a 2-pole Sallen and Key low pass filter. Two stages are cascaded to form a simple four pole filter. Notice the values of the resistors and capacitors are different in each pair of filters. It is these that set the cut off frequency.

To create a gate signal we do need a very fast response. In an ideal world this signal must go high the moment the signal arrives and goes low the moment the signal dies away. In this case I have used the *peak and droop* method. This does give us a fast as response as possible, but what about the ripple. Well, ripple is not *that* important here. Remember the gate output only goes high or low. What we have to do is make sure our gate doesn't 'rattle' when it picks up the ripple. In other words, we need our gate to come cleanly on and off with no spurious states as the signal rises and falls.

U5 is a comparator. This is a device based around an op-amp in this case, that determines whether a signal is higher than a pre-selected threshold voltage. The threshold voltage is controlled by the user, and is set by the **Threshold** pot. The threshold voltage can be set between 12V and 0.7V. C28 is charged via D7 from the FWR output. D7 allows the capacitor to be charged up, but not discharged, by the FWR's output. R53 allows the capacitor's stored peak voltage to droop at a controlled rate. Most gate extractors provide a gate signal when the voltage on the capacitor is above a certain value. The EFG is different. Once the gate does go high, a certain proportion of the opamp's high level output is fed back to keep the input higher. This forces the comparator to stay high longer than it would normally do. This allows more ripple to be present before ‘rattling’ occurs, giving us a cleaner edge to our gates. You have control over this amount of positive feedback, and I call it the **Response**. Now many good comparator designs have a little positive feedback anyway, its called hysteresis, and in our case its provided by R22. But the **Response** pot offers a type of one way hysteresis that can be controlled for better high to low gate transitions.

This system is still not perfect, but with careful selection of the **Response** and **Threshold** pots it should be able to meet most requirements.

Q3, Q4 and associating circuitry provide the necessary buffering to provide a +7V5 gate signal. If you require a +15V gate, then remove R55. Q1 provides the current switching for the **gate** LED. When the op-amp output is high, this transistor turns on allowing the LED to light. Current through the LED is set to be around 6mA.

Another comparator circuit, based around U5b, is used to operate the **peak** LED. This is normally achieved more simply using just a transistor, but I had the spare op-amp half available and the results from this sort of circuit are more predictable. The LED is designed to turn on when the output of the FWR reaches around 10V. An op-amp running off a +/-15V rail will be able to output around 13V maximum, so enabling the **peak** LED to turn on at 10V gives you just the right amount of headroom. You should normally operate the EFG so that the peak LED just flickers with the peaks in the music. It will work very well below this, but you will not be getting the maximum output possible from the CVs.

A visual indication of the CV outputs is available from the **F’low** LED. This is driven from a current source provided by op-amp U3a. The LED in the feedback loop will have a current that is determined solely by the voltage presented to the end of R28. A 5V CV output, will produce 5mA in the LED. Although in normal operation the LED is always forward biased, it may be subjected to odd negative swings on power up and power down. A normal diode should have been placed in parallel with the LED pads to prevent damage to the LED but it was missed out in the layout. However, all is not lost, just make sure that the **F’low** LED is a bipolar LED or a bicolour LED. A bipolar LED contains two LEDs in parallel but facing the opposite way. Not only, does it mean you can put it in any way and it will still work, but it each LED will protect the other from reverse bias. A normal LED can be used with the proviso of fitting a 1N4148 diode in parallel with the LED. Bipolar LEDs are only a few pence more than ordinary LEDs, so I often use them anyway to make wiring simpler.

Some of you may be wondering why the **LED-DRIVE** signal comes from the first stage 33Hz filter and not the final output? No you’re not? Well, you should be! Because its a bit clever. Op-amps only have a limited current driving capacity, about 15mA is really about the most you would want to pull or push from a normal op-amp. The op-amp output driving **LED-DRIVE** must supply the same current as the LED is taking. This can rise to a whopping 10mA peak when the CV is at its highest output. So rather than cripple the main CV output with this horrible burden, I will let the first stage deal with it. This way the main CV output is happily able to supply its full capacity into the CV output itself.

Components

Most of the parts are easily available form your local parts stockist. I use Rapid, RS components, Maplin and Farnell, here in the UK. 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 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 and Rapid sell the Omeg pots at a very good price. But note that Maplin no longer have the pot brackets or the dual gang pot required. And Rapid don't sell the pot brackets. The pot kit that I supply contains all three pots and the three pot 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% resistors for less than 2p each!

For the capacitors, there are no set rules. All the aluminium electrolytics, marked as 'electrolytic' in the parts list, should have a working voltage of 25V or 35V. Don't chose too high a working voltage. 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. They also should be radially mounted and not axial types. Radial types are the ones with the leads sticking out the bottom of the device.

The low capacitance (values in pF) ceramics have 5mm (0.2") lead spacing. For these three ceramics use low-K types, these are the better quality ones with higher stability and lower noise. They are sometimes described as NP0 or C0G types.

For the non ceramic types I think polyester types are fine for all decoupling, coupling and filter uses. On my more recent boards I have switched over to 5mm box capacitors but for this board you need to use 7.5mm lead spacing. The ones I recommend are normally called 'poly-layer' and are available in many different working voltages. Use 100V for values above 100nF, and 400V for the smaller values. But remember the pitch spacing.

You can also use metalised polyester types, but again do be sure you get low working voltages. Around 63V to 150V is best. This is especially crucial for C28 and C29. These both are 680nF. This is a big value, and these capacitors can be physically large. If you chose a 100V poly-layer capacitor, they will fit with ease. Be wary if you buy any other type. Do check the size. In the UK, Farnell and Maplin can supply all the capacitors.

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

For the suggested layout you also need three LEDs. Feel free to use any colour. As stated earlier, the F'low LED must be bipolar. I use 5mm LEDs with suitable LED holders. Maplin still sell their excellent Cliplite clips. I use the clear type with water clear LEDs from Rapid. If you need a bipolar water clear LED, the best thing to do is use a bicolour LED. After all a bicolour LED is a bipolar LED of two different colours. The PCB is laid out so that the LEDs will fit into the PCB and poke through the suggested front panel layout. It is best to solder the LEDs in after you have fitted the board to the panel if you do this. You can test the board without soldering the LEDs just by placing them in their respective holes. This won't harm the LEDs or the circuitry.

The BC549 transistors can be pretty much any NPN transistor that corresponds to the same pin out. For example: BC550, BC548, BC547 etc. However, I recommend using BC549 or BC550 only as these are low noise devices. Quite often you see an A, B or C suffix used, eg. BC549B. This letter depicts the gain or grade of the transistor (actually hfe of the device). The EFG is designed to work with any grade device although I have used BC549B throughout in my prototypes.

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; TL074CP. Do not use SMD, SM or surface mount packages. They do not fit at all.

Input and output sockets are not board mounted. You can choose whichever type of sockets you wish. I use the excellent Switchcraft 112 as used on the Moog and MOTM modulars. At least two of the sockets must have normalising lugs if you are building the EFG-Deluxe. The Switchcraft 112 types have normalising lugs as standard.

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

Parts List

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

Resistors 1/4W, 5% or better.

10R	R31, 27
22R	R33, 34
47R	R30, 43, 51
100R	R15, 56
1K	R26, 28, 41, 42, 10
2K2	R1, 2
3K	R6
3K9	R24
6K8	R3
7K5	R52, 55
10K	R13, 29, 36, 12, 54, 53

12K	R7
15K	R48
22K	R8, 11, 35, 47, 44, 5
39K	R21
47K	R45, 49
100K	R32, 16, 17, 18, 19, 9, 50, 4
220K	R23, 37, 38, 39, 40
470K	R25, 20
1M	R14, R46
3M3	R22

Capacitors

22uF, 25V electrolytic	C4, 14, 16, 2, 1, 6, 10, 13, 15, 22, 25, 26, 27
22nF, 400V polyester	C18, 19, 20, 21
10nF, 400V polyester	C8, 9, 11, 12
100nF, 100V polyester	C5, 23, 24
680nF, 100V polyester	C28, 29
22p Ceramic low-K	C3, 7, 17

Semiconductors

1N4148	D1-7
BC549	Q1-4
TL072	U2,3,5
TL074	U4
TL072 or OP-275GP	U1
5mm Orange LED	Gate
5mm Red LED	Peak
5mm Bipolar Green LED	F’low

Other

4-way 0.156” Molex/MTA connector	PSU
47K linear dual gang variable resistor	Gain
47K linear single gang variable resistor	Threshold
1M log single gang variable resistor	Response
25cm of audio grade screened cable	
1m of multistrand hook up wire	
Three knobs	
Three LED clips	

For the standard EFG you need four decent quality jack sockets, eg. Switchcraft 112. For the EFG-Deluxe you will need a total of eight sockets.

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

Building the EFG

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!

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, 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 recently 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 200V. 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.

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.

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 polyester layer capacitors are like little silver sandwiches. Push the part into place up to the board’s surface. 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 sockets, small non-polar capacitors, transistors, electrolytic capacitors. Then the final water wash. Do not fit the pots or the LEDs at this stage. The mounting of the pots and the LEDs requires special attention. See the next section for more details.

Mounting the Pots and LEDs

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.** Now remove all the nuts and washers from the pots and fit the board up to your front panel. Refit the washers and tighten the nuts, but not too tight. Now carefully position the PCB at right angles to the panel. The pot’s own pins will hold the PCB 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: 47KB or 100KA. Omeg uses the European convention of A = Linear, B = logarithmic and C = Reverse logarithmic. So a 47KC is a 47K reverse log pot.

The pots shafts may be cut down with a good pair of pliers, or a junior hack saw. Try not to bend or rotate the shaft as you are cutting.

The pots are lubricated with a thick clear grease. This sometimes is visible along the screw thread 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 three LEDs are probably best not soldered until the board has been mounted to the front panel. Preform the LED legs using a pair of fine nosed pliers, bending the leads close 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 **Gate** and **Peak** LEDs connected the right way. Pin 1 which is the square pad must be connected to the anode of the LED. The **F'low** LED must be a bipolar type, and can be fitted anyway.

Connections

This module is very easy to connect up either in its standard form or the slightly more complex EFG-Deluxe.

EFG Standard Configuration

There are just four sockets and three LEDs in the suggested EFG layout. The suggested version does not make the preamplifier output available separately. The preamplifier drives the envelope follower directly. This is achieved by linking between IN2 and OUT1 on the PCB with a small loop of wire.

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.

Connect, with a piece of insulated wire, each signal tag to the respective pad on the PCB. You should have Slow, Fast and Gate. Leave the NC tag unconnected.

The Input jack is best connected with a piece of screened cable. There is provision on the PCB for screen and the core, with a solder pad for each one. Solder the core of the cable to the signal tag of the input socket, and the screen of the cable to the NC tag. Do not solder the screen to the earth tag of the socket at all. The earthing is done with a separate lead.

The ground tags of each socket can be all connected together with solid wire. A piece of insulated wire can then be used to connect the tags to the PAN pad. Do not connect the GND or SCR pads to the earth tags at all.

You should find that the GND pad has not been used. This connects to the module GND and is not normally used.

EFG-Deluxe (Envelope follower with lag generator)

This dual unit uses a 2U wide panel to accommodate both PCBs and the eight sockets needed. Fit the completed Little-Lag PCB to the front panel. Position the board at right angles to the panel and solder the pot brackets. Now remove the board and put to one side. Now attach the EFG PCB to the front panel. Solder the pot brackets and LEDs in place if you haven't done this already. The LEDs will be difficult to solder once the Little-Lag PCB is put in later.

Fit all eight sockets to the front panel. 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.

The ground tags of each socket can be all connected together with solid wire. A piece of insulated wire can then be used to connect the tags to the PAN pad. Do not connect the GND or SCR pads to the earth tags at all.

The PRE IN socket is best connected to the IN1 pad with a piece of screened cable. There is provision on the PCB for screen and the core, with a solder pad for each one. The core going to IN1, and the screen going to SCR. Solder the other end of the core of the cable to the signal tag of the input socket, and the screen of the cable to the NC tag. Do not solder the screen to the earth tag of the socket at all. The earthing is done with a separate lead.

The PRE-OUT and AUDIO IN sockets are the next to be connected. Connect the OUT1 pad to the signal lug of the PRE OUT socket, but do not solder the wire onto the signal lug just yet. Just loop it around the solder tag. Also around the same tag connect a wire and take it to the NC lug on the AUDIO IN socket. You should have two wires coming from the signal lug of the PRE OUT socket and these can both be soldered together. Now solder a wire from the IN2 pad to the signal lug of the AUDIO IN socket. What this has done is to normalise the pre-amp output to the input of the envelope follower circuitry. This input will be overridden once a jack is inserted into the AUDIO IN socket.

Connect with insulated wire the signal lugs of the SLOW OUT, FAST OUT and GATE OUT to the 'Slow', 'Fast' and 'Gate' pads on the PCB. Connect also to the FAST OUT signal lug a wire to the NC tag of the LAG IN socket. This will pass the fast output of the EFG to the Little-Lag circuit automatically.

Now fit the Little-Lag circuit to the front panel again. Connect a wire to the IN pad on the PCB to the LAG IN signal lug. Connect a wire from the OUT pad to the LAG OUT signal lug.

You should find that the GND pad on the EFG PCB has not been used. This connects to the module GND and is not needed. On the Little-Lag PCB, three pads are not used. These should be the IN-G, OUT-G and PAN.

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 module’s 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 built the EFG-Deluxe, then both PCBs must be powered separately. You will therefore need two power connectors, one for each board. You could, of course, hard wire the power lines from one board to another. However, this may hamper fault finding should a problem occur.

At the rear of this user guide I have included 1:1 drawings of the suggested front panel and the dual EFG-Deluxe. 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 chosen fpd file that is found on the EFG 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.

Using the EFG

Like many modular synthesiser modules the best way to find out what it does is to actually use it. However, here’s a quick tutorial on using the Oakley EFG.

Fire up a drum machine or some other music input, and connect the output to the EFG’s input. Adjust the **Gain** pot until the **Peak** LED just flickers gently in time to the music. Now connect the **Slow** output CV to something you would like to control. Try listening to a VCO with the EFG output controlling VCO pitch. Mmm, its a bit of a racket, but quite interesting. Listen as the loud parts of the music make the VCO’s pitch go really high. Right, enough of that, my ears are hurting!

Set up the classic synth patch, that is: VCO to VCF to VCA, and use your keyboard to control an ADSR to gate the VCA. Now use the EFG to control the cut off frequency of the filter. When you play notes on the keyboard, you should find that the notes will bounce in time to the music. Adjusting the **Gain** will control the depth of the effect.

Now remove the slow CV from the VCF, and connect the VCA's control voltage input to the EFG's **Gate** output. Adjust the **Threshold** and **Response** pot to their central positions. If you are using a drum machine to drive your EFG, you should notice that the notes you hear are now being 'gated' by the beat. The **Gate** LED will also be flashing with the music too. The Threshold pot will adjust what input level is required to make the sound heard. Try adjusting it and see what effect it has. Depending on your choice of input material, you should find that a lower Threshold will cause the notes to be heard for longer periods of time. If you find the notes have a raspy edge to them as they turn on and off, try adjusting the response pot until you get cleaner transitions.

If you use the gate output to drive the an ADSR which in turn drives the VCA, you will find you have even more control over the way the sound is gated on and off.

Now try using the Slow output of the EFG to control a VCA. If you connect the same program input to the VCA and the EFG and listen to the output of the VCA, you have made a very simple audio expander. Loud sounds are made very loud and quiet sounds are made even quieter. By changing the Initial level and CV depth on the VCA you will effect the depth of the effect. Now, invert the EFG's slow output with a MultiMix or other inverting module before putting it into the CV input of the VCA. Now, loud sounds are made quieter, and quiet sound are made louder. You have now made a compressor. This works very well on rhythmic inputs like that from a drum machine.

If you have a VC-Lag module you can use this to process the **Fast** output to generate different CV responses. When used in conjunction with a suitable VCF, you can recreate the effects of some guitar envelope follower filters like the Q-tron. Set the Rise time to fast and the fall time to around 0.5 seconds to replicate the *peak and droop* method of envelope detection.

This hopefully will have given you a few ideas. However, feel free to experiment with your new module and create some really wild sounds.

Final Comments

I hope you enjoy building and using the Oakley *Envelope Follower and Gate Extractor*. 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. 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 especially to all those nice people on the synth-diy, Oakley-Synths and MOTM mailing lists.

Tony Allgood. May 2002

Version. 1.03

Formatted on Lotus Word Pro