Oakley Sound Systems

EFG & EFG-Deluxe
Envelope Follower & Gate Extractor

PCB Issue 3

User's Guide

V1.2

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

This User Guide for issue 3 EFG boards only. Your board should say 'EFG ISS.3' in the lower left hand corner of the PCB.

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.

Rather importantly the EFG 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 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 follower output signal. This allows you to create more flexible control signals to drive your filters and/or VCAs. The classic 'vactrol' based auto-wah pedals can be replicated this way with their fast attack and slow decay time.

The new issue 3 board differs only in few aspects to the original issue one and two layouts. The main change is that the board has been laid out for Spectrol 248 and 148 pots rather than the Omeg E16 style pots used on older Oakley boards. These high quality pots are longer lasting and have a nice smooth feel to them. Secondly, the layout has been optimised to make the board smaller.

Two new panel designs have also been introduced with this new issue board. The 2U wide EFG-Deluxe panel has had a little face lift. The whole Little-Lag section has been offset moving the pots off the standard MOTM grid. This presents a module that is instantly recognisable and therefore easier to use in a large modular.

The standard EFG has been changed too. This is in answer to a request from one customer who required the preamplifier output to be available from a 1U panel. The three LEDs have been moved left, and a little downwards, to allow the addition of a new switch. The switch selects whether either the slow **or** fast output is available from the CV output. The space made available from the loss of one of the two CV sockets, allows the preamplifier to have its own socket

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 Spectrol pots and Oakley pot 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 per rail, although operation of the LEDs will affect this value slightly. The addition of the Little-Lag adds an extra 7mA per rail. 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-3 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 which offers excellent performance. 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 via the GAIN pot. C15 keeps the gain for DC and very low frequency signals at near to one. This prevents any offsets within U1 from being amplified unnecessarily C12 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 original suggested layout of the EFG did not have a separate output for the pre-amp. However, I have now modified the design of the panel to allow for the pre-amp to be available. In the standard EFG, the output of the pre-amp also feeds directly 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 will be doing things slightly differently. In this form of the module you will have a separate input to the follower circuit. 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 three. 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 and associating circuitry form 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 five 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 in the gate extraction circuitry.

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 if you wish. The suggested panel layout has a switch to select which output. The EFG-Deluxe provides both outputs as standard. The 33Hz one 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.

Users 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 and takes the output of the FWR buffer directly to the Little-Lag circuit board via the pad marked 'LAG'.

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 (pins 1,2,3) 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 D8 from the FWR output. D8 allows the capacitor to be charged up, but not discharged, by the FWR's output. R52 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 R23. 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. The EFG 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 R30. 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, D1, has been placed in parallel with the LED pads to prevent damage to the LED.

Some of you may be wondering why the **LED-DRIVE** signal comes from the first stage 33Hz filter and not the final output? 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.

Buying the components

Most of the parts are easily available from your local parts stockist. Rapid Electronics, RS Components, Maplin and Farnell, are popular here in the UK. The EFG was designed to be built solely from parts obtainable from Rapid Electronics and Oakley Modular. 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 the Nordic countries 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 now Spectrol 248 series pots with 1/4" shafts. These are high quality sealed conductive plastic potentiometers. Rapid and Farnell sell these parts in the UK. The pot brackets are especially made for us, and are only available from Oakley Modular, who also sell the pots should you find it difficult to get them yourselves.

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!

All the electrolytic capacitors 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 all the values between 10nF and 680nF, 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. This is especially crucial for C28 and C29, the 680nF ones. This is a big value, and these capacitors can be physically large. If you chose a 63V polyester capacitor, they will fit with ease. Be wary if you buy any other type. Do check the size. In the UK, Rapid and Farnell can supply all the capacitors.

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.

L1 and L2 are radially mounted ferrite beads. These look a little like black resistors. They are usually in the EMC or Filtering section of your components catalogue. Farnell sell them as part number: 9526820 (latest RoHS compliant component). Rapid sell them as part number: 26-4860.

For the suggested layout you also need three LEDs. Feel free to use any colour. I use 5mm bipolar LEDs with suitable LED holders. I use bipolar LEDs in my ready made modules and pre-populated options. This is purely 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. For the LED holders Maplin still sell their excellent Cliplite clips.

The BC549 transistors can be pretty much any NPN transistor that corresponds to the same pin out. For example: BC550, BC548, BC547 etc. 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 ungraded BC550s throughout in my prototype.

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; TL074CN. The OP275 usually comes with a G or GP suffix, ie. OP275G. Do not use SMD, SM or surface mount packages. They do not fit at all.

If you are building the EFG with the switched output option you'll need a decent toggle switch. Its just a normal SPDT toggle type, but get a good quality one. Apem and C&K make good ones. Farnell part number: 1082301. There will be more detail on how to wire the switch to the board later on in this document.

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.

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 always correct. Please e-mail me if you find any discrepancies.

Resistors

Resistors 1/4W, 5% or better.

10R	R33, 29
22R	R36, 35
47R	R42, 31, 50
100R	R14, 56
820R	R26
1K	R28, 30, 9, 19, 20
1K5	R7
2K2	R1, 2
3K	R3
4K7	R8
7K5	R54, 55
10K	R10, 11, 52, 51, 32, 12, 53
12K	R6
15K	R47
22K	R5, 43, 37
27K	R46
39K	R22
47K	R44, 48
100K	R34, 49, 25, 17, 18, 15, 16, 4
220K	R24, 38, 41, 40, 39
470K	R21, 27
1M	R13, 45
3M3	R23

Capacitors

22uF, 25V electrolytic	C2, 15, 1, 14, 13, 23, 22, 11, 6
2u2, 50V electrolytic	C26, 27
22nF, 100V polyester	C18, 19, 20, 21
10nF, 100V polyester	C7, 8, 9, 10
100nF, 63V polyester	C5, 24, 25
680nF, 63V polyester	C16, 28, 29
22p Ceramic low-K	C12, 4, 17

Semiconductors

1N4148 silicon signal diode	D1-8
BC549 or BC550 NPN transistor	Q1-4
TL072 dual Bi-FET op-amp	U2, 3, 5
TL074 quad Bi-FET op-amp	U4
OP-275GP low noise audio op-amp	U1
5mm Yellow LED	Gate
5mm Red LED	Peak
5mm Green LED	F'low

Other

4-way 0.156" Molex/MTA connector	PSU
50K linear single gang variable resistor	Threshold, Response
10K linear dual gang variable resistor	Gain
SPDT switch	EFG only, not EFG-Deluxe

25cm of audio grade screened cable
1m of multistrand hook up wire
Three knobs
Three LED clips
Power lead MTA to MTA connector

You will also need a small power lead to supply the Little-Lag for the EFG-Deluxe version. This consists of a standard 4-way MTA socket with 10cm of wire leading from each terminal. This will need to be soldered into the PS1 pads on the EFG board, see later.

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. You will need four 8-pin DIL sockets and one 14-pin DIL socket.

Building the EFG

Warning:

Oakley Modular PCBs are now supplied with the RoHS compliant Lead Free HASL 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 tracks 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. The most common error with many of these was the wrong parts inserted into the board. 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 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 film capacitors are like little coloured boxes. Push the part into place up to the board's surface. Little lugs on the underside of the capacitor will leave enough of an air gap for the water wash to work. Cinch and solder the leads as you would resistors.

The electrolytic capacitors are very often supplied with 0.1" lead spacing. My hole spacing is 0.2". This means that the underside of these radial capacitors will not go flat onto the board. This is deliberate, so don't force the part in too hard. The capacitors will be happy at around 0.2" above the board, with the legs slightly splayed. Sometimes you will get electrolytic capacitors supplied with their legs preformed for 0.2" (5mm) insertion. This is fine, just push them in until they stop. Cinch and solder as before. Make sure you get them in the right way. Electrolytic capacitors are polarised, and may explode if put in the wrong way. No joke. Oddly, the PCB legend marks the positive side with a '+', although most capacitors have the '-' marked with a stripe. Obviously, the side marked with a '-' must go in the opposite hole to the one marked with the '+' sign. Most capacitors usually have a long lead to depict the positive end as well.

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 Spectrol Pots and LEDs

NOTE: This procedure is rather different to that of the Omeg 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
50K linear	M248 50K M	2 off
10K linear dual	149-DXG56 S103SP	1 off

Fit the pot brackets to the pots by the nuts and washers supplied with the pots. You should have two nuts and one washer per pot. Fit only one nut and its associating washer 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 PCB. 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 all three 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. Although, I usually fit the sockets at this point, and wire up the ground tags first. After this is done, I then mount the PCB to the front panel. Again, do not over tighten. You may want to add an extra washer at between the panel and the nut, although this extra washer is not provided in the pot kit.

The pots shafts will not need cutting to size. They are already at the correct length.

The pots are lubricated with a light 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 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 anode of the LED. It doesn't matter which lead goes into which hole of the LED pad if you have used bipolar LEDs.

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

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

EFG New Standard Configuration

There are four sockets, one switch and three LEDs in the new suggested EFG layout. The later versions of the suggested layout now makes the preamplifier output available, but looses the simultaneous slow/fast outputs in favour of a switch. The preamplifier still drives the envelope follower directly.

Link out the pads IN2 and OUT1 on the PCB with a small loop of wire. A resistor lead clipping is perfectly suitable for this. Make the loop of wire stand proud of the board's surface. You'll be soldering onto this loop later when you wire up the sockets.

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 inserted. This connection is automatically broken when you insert a jack. The tags are actually labelled in the plastic next to the tag. The signal lug is called 'T' for tip, the NC lug is labelled 'T/S' for tip-switched.

In this module we are going to 'common' the sockets' ground lugs. This means that the sockets' lugs are going to be joined together. I normally do this part of the wiring without the PCB or pots in place.

Fit all the sockets onto this module so that the bevel on the side of the socket is facing top left as you look at the rear of the panel. There are just four sockets in total.

The lugs we are connecting together will be the ground or earth tags on the two horizontal rows of sockets. I use 0.91mm diameter tinned copper wire for this job. Its nice and stiff, so retains its shape. Solder a length of this solid core wire right across the two earth tags on the top row. Trim off any excess that sticks out on either end. Then do the same on the lower row of sockets. What you have now done is common each row's earth tags together, but each row is still separate for now.

Fit the PCB against the front panel if you haven't done so already. Solder a piece of ordinary insulated wire to the earth lug on the socket furthest on the left on the top row. The other end of this wire needs to go to the pad on the PCB marked PN1. Now solder another piece of wire to the earth lug of the socket furthest left on the bottom row. This wire will be going to the pad PN2. Your earth tags are now commoned together.

Solder the three LEDs if you haven't done this already. Now fit the switch. Solder a piece of insulated wire to the top pin of the switch and connect the other end to the pad marked FAST on the PCB. Solder another piece of wire to the bottom pin, and then connect the other end of this wire to the solder pad labelled SLOW. The middle pin will be left until later.

Now its time to connect up the sockets' signal lugs. The input jack, labelled PRE-IN, 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 gate socket is next to be connected. Solder a wire so that its signal lug connects to the PCB pad marked GATE.

The PRE-OUT socket's signal lug must be wired to the little wire loop joining the pads IN2 and OUT1. If you use thin hook up wire for your interconnections you may find it neater to simply solder the wire into one of the solder pads that is also being used by the wire link.

The CV OUT socket's signal lug is the last wire connection to be made. This should go to the wiper of the switch, that's the middle pin of the switch.

You should find that the LAG pad has not been used. It is the direct output of the full wave rectifier and is used in the Deluxe version only.

EFG-Deluxe (Envelope follower with lag generator)

This dual unit uses a 2U wide panel to accommodate both PCBs and the eight sockets needed.

Your first job is to 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.

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 inserted. This connection is automatically broken when you insert a jack. The tags are actually labelled in the plastic next to the tag. The signal lug is called 'T' for tip, the NC lug is labelled 'T/S' for tip-switched.

In this module we are going to 'common' the sockets' ground lugs. This means that the sockets' lugs are going to be joined together. I normally do this part of the wiring without the PCB or pots in place.

Fit all the sockets onto this module so that the bevel on the side of the socket is facing top left as you look at the rear of the panel. There are eight sockets in total.

The lugs we are connecting together will be the ground or earth tags on the two horizontal rows of sockets. I use 0.91mm diameter tinned copper wire for this job. Its nice and stiff, so retains its shape. Solder a length of this solid core wire right across the four earth tags on the top row. Trim off any excess that sticks out on either end. Then do the same on the lower row of sockets. What you have now done is common each row's earth tags together, but each row is still separate for now.

Fit the EFG PCB against the front panel if you haven't done so already. Solder a piece of ordinary insulated wire to the earth lug on the socket furthest on the left on the top row. The other end of this wire needs to go to the pad on the PCB marked PN1. Now solder another piece of wire to the earth lug of the socket furthest left on the bottom row. This wire will be going to the pad PN2. Your earth tags are now commoned together.

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 already done with the stiff wire frame.

Make sure you solder the LEDs in place at this point if you haven't done so already. The LEDs will be difficult to solder once the Little-Lag PCB is put in later.

The power lead to the Little-Lag can now be fitted. This should be constructed as you would a normal power lead. But make it only 11cm in length and terminate only one end with a standard 0.156" MTA or Molex socket. The four bare ends can then be soldered into the EFG PCB at PS1. Solder the wires in so that pin one of the MTA socket will connect with pin 1 of the header on the EFG PCB.

You may wish to use two headers in your EFG board, that is, one in PS1 and PS2. This will make any connecting wire removable from both boards. However, take note that if you use standard Amp MTA headers this cannot be done. This is because MTA sockets have the wires leaving the socket at right angles to the body of the socket. These wires would then foul the outer MTA header. If you use Molex connectors this will not be an problem, since the wires head upwards. Personally, this is not an issue since I always solder the four ways into the EFG board directly, using an MTA header on the Little-Lag board only.

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. Also, connect the LAG IN socket’s NC lug to the ‘Lag’ pad on the EFG PCB.

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.

Wire up the LIN/EXP switch as follows. Connect your switch so that the two pads are shorted by the switch contacts when the switch is in the EXP position. For a standard toggle switch this will mean that top two pins of the switch are connected to these pads. It doesn’t matter which one goes to which pad.

I use a few cable ties to keep all the wires nice and tidy. You’ll find that there are two runs of wires that can be tied together. The ones going to the top of the EFG board can be bundled together, and the ones to the Little-lag board.

You should find that on the Little-Lag PCB the PAN pad is not used.

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 PN1 and PN2 pads 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.

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.17

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. If you have built the EFG-Deluxe, you have the lag generator built in. This is wired to the output of the full wave rectifier of the EFG circuitry. This allows very fast attack (rise) times to be achieved when the UP pot is set to its minimum value.

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*.

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.

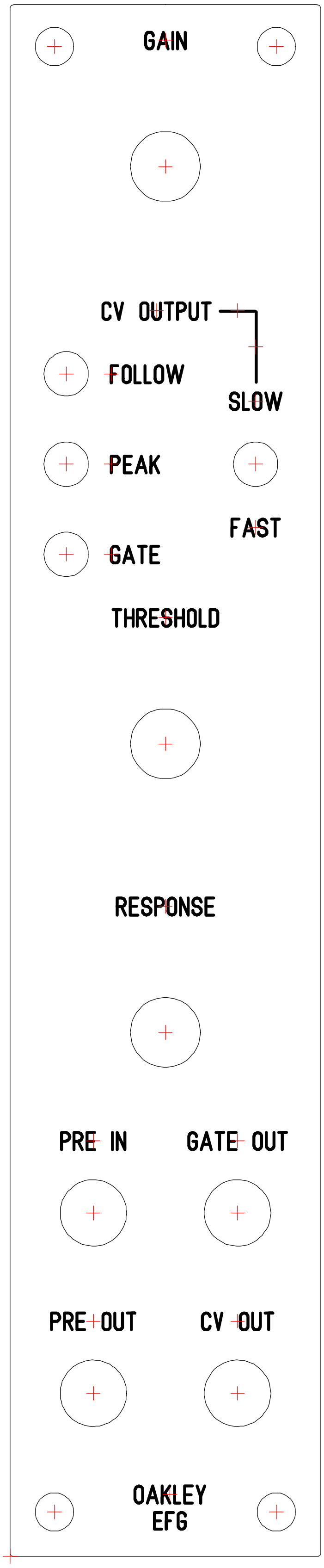
Tony Allgood

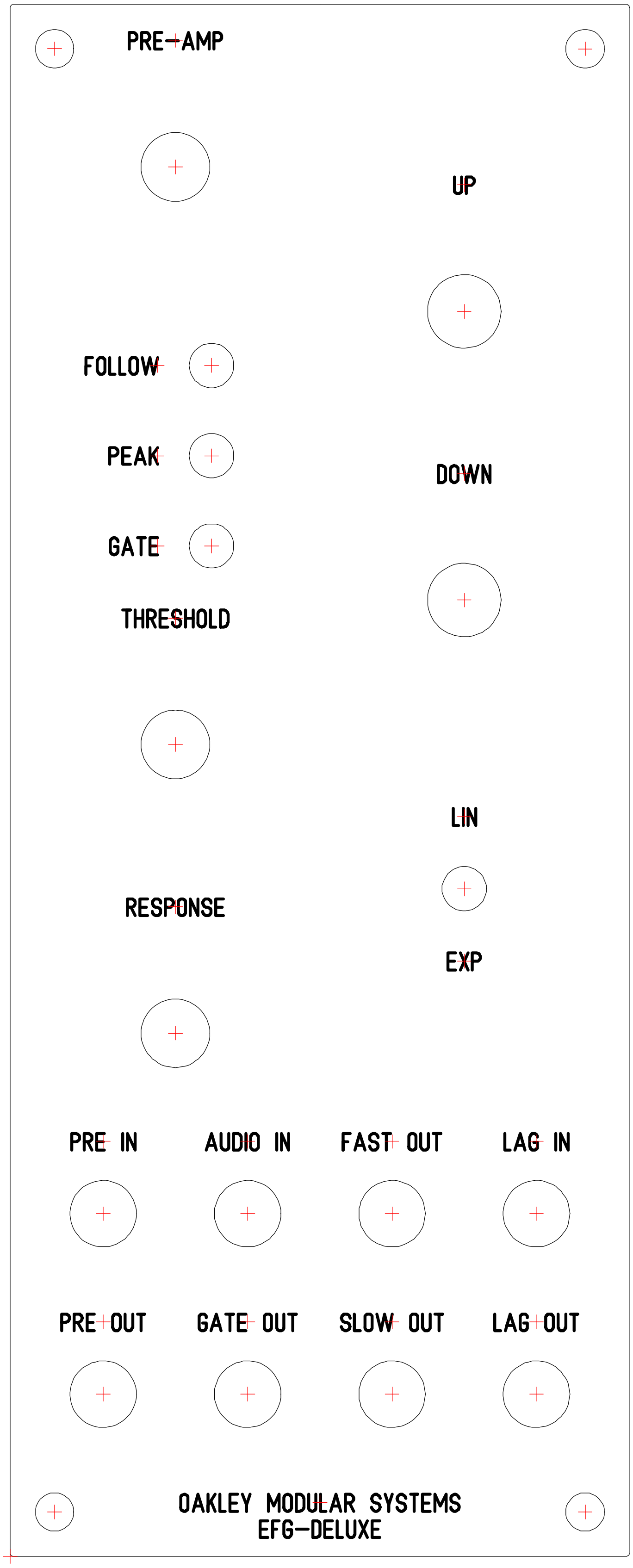
Cumbria, UK
May 2007

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.


GAIN
CV OUTPUT
FOLLOW
SLOW
PEAK
FAST
GATE
THRESHOLD
RESPONSE
PRE IN
GATE OUT
PRE OUT
CV OUT
OAKLEY
EFG


PRE-AMP
UP
FOLLOW
PEAK
DOWN
GATE
THRESHOLD
LIN
RESPONSE
EXP
PRE IN
AUDIO IN
FAST OUT
LAG IN
PRE OUT
GATE OUT
SLOW OUT
LAG OUT
OAKLEY MODULAR SYSTEMS
EFG-DELUXE