Oakley Sound Systems
Little-EG

Dual AD/AR Envelope Generator

User's Guide

V1.0

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

Introduction

This is a simple but effective envelope generator module primarily designed for use in modular synthesisers. Its simple but innovative design coupled with the use of PCB mounted pots makes this an ideal starter module for beginners to my modular synthesiser projects.

The envelope generator or EG for short, generates a rising and falling voltage at its output when triggered by a gate signal. The gate is derived from a keyboard, switch or a midi-CV convertor elsewhere in the synthesiser system. The speed of the rise in output voltage is determined by the Attack control. The speed of the fall of the output voltage is determined by the Decay or Release controls. The output of an EG is traditionally patched to the VCA control voltage input, to control the volume of the note when the keyboard is pressed; or to the VCF, where it dynamically alters the harmonic structure of the sound throughout the duration of the note played.

Traditionally there are three basic analogue EG types available: attack-decay (AD), attack-release (AR), and attack-decay-sustain-release (ADSR). Other types are available, especially on digital synthesisers, but the three analogue types still seem to the most musician friendly.

The AD type produces a rising voltage in the attack phase. This rises exponentially from zero until a predetermined value is reached. The decay phase then starts where the output voltage falls back to zero. If the gate is removed during the attack phase, the decay is prematurely started and the voltage output will fall without the peak ever being attained. Two pots are required for the control of an AD type EG.

The AR type produces a rising voltage in the attack phase, again rising exponentially, to the sustain level. The voltage will then remain at this level for as long as the gate is held high. When the gate is removed, the output will fall back to zero. This is the final release stage. Again, only two pots are needed for an AR-EG.

The ADSR type is controlled by four pots. The attack phase is initiated by the gate signal, the output will rise to a predetermined level, whereupon the decay phase takes place. However, the output voltage will fall to the level set by the sustain pot. It will stay there until the gate is removed, when the final release stage is initiated.

The ADSR-EG can perform both AR and AD operations simply by turning the sustain pot full up or down respectively. It is this that has made the ADSR the most common type of EG within a synthesiser.

The Juno-6 and its followers were severely hampered by just one ADSR. The MicroMoog had just two simple EGs and in my opinion performed much better. Each of its EGs can only be switched to be either AD or AR. I was amazed that this did not really degrade its sound palette in any major way. My other homemade synths had full ADSRs, but on reflection, I rarely used the sustain pot properly. For the VCA, I normally set the sustain pot full up. For the VCF, I normally set the sustain at minimum. Sometimes, it is true, a certain sound does demand use of the full ADSR, but there are other ways around this.

So back in late 1999 with the help of Steve Ridley, I had a rethink for my new EG design. The Oakley EG would be able to produce AD and AR simultaneously, so no need for a switch here. The unit could be wired up to have two pots, so that attack and decay/release phases could be controlled together for the both types of EG output. Or, you could build it so that it could have four pots to control the AD and AR parts separately.

If you went for the four pot version you could fit one EG into one 1U wide panel. If you used both outputs and mixed them together, you could get ADSR if you wished, but you could get much more. Imagine a sound that quickly swells up and falls, only to be built up again and held with sustain. I soon began to realise that even this simple 1U module was more useful in a modular set up than a single ADSR-EG.

So in January 2000, the Oakley 'Basic-EG' was born. It was 'basic' since it formed part of a planned series of simple to build modules that would fit into MOTM panels. The 'Basic-VCA' and 'Basic-LFO' followed shortly after. Afterwards I felt that the name 'basic' didn't do the modules any favours. They were more than basic. In 2001 the series was renamed the 'Little' range, starting with the Little-LFO and then the Little-Lag. This aptly described the size of the board but not their usefulness.

In April 2003, the new Little-EG was announced. This is quite simply a revised and updated version of the original Basic-EG. We now have 0.156" MTA power headers, and the solder pads have been increased for easier soldering. The gate input now has a 'schmitt' trigger action to ensure reliable triggering on slow moving CV inputs.

The PCB

The PCB is indeed quite little and is almost cute at just 7.4 x 6.1 cm in size. This is a tad bigger than the original issue board. As with all Oakley boards 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 matches well with the MOTM synthesiser as well as being the same as all other Oakley modules.

You have two suggested options on panel layout. Option 1 is the so called Dual-EG mode. This is a four pot design, that has separate attack and decay pots for the AD and the AR outputs. You will use single 470KB pots. You will only mount two of the pots to the PCB, the AR ones, and the other two will have to be connected by small lengths of flexible wire. If you are mounting the board vertically to go into a 1U panel, make sure the two board mounted pots are the bottom pair of pots to avoid fouling the mounting rails.

The option 2 design is the Dual-Output mode. Here you use two dual ganged pots and have the time constants of the AD and AR outputs linked. Both front panel layouts are shown at the rear of this User Guide.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 20mA per rail. Power is routed onto the PCB by a four way 0.156” header. You could, of course, wire up the board by soldering on wires directly. The four pins are +15V, ground, panel, -15V. The panel connection allows you to connect the metal front panel to the power supply’s ground and therefore earth without it sharing the modules’ ground line. More about this later.

Circuit Description

The EG 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 PWR. Power enters the board here, and is immediately filtered by a simple RC networks based around R8 and C4 for the positive rail, and R7 and C3 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 supplies to each IC is shown separately to avoid cluttering the main circuit diagram. Note that the TL072 requires both negative and positive supplies, while the LM556 requires only a single supply. However, the supply to the LM556 is heavily decoupled and roughly regulated. R20 provides the current to keep C8 ‘topped up’ with voltage. The zener diode, D4, limits this voltage to a maximum of 8.2V. When the output of the LM556 changes state, the device shorts out its own power supply for a very short length of time. This can cause all sorts of problems if it is not catered for. C8, provides the LM556 with a very large reservoir of current, and R20 effectively isolates it from the main +15V supply. The zener diode is chosen so that the maximum output of the EG is set at around 5V. This is a bit crude, but it works well enough.

A gate signal is a control voltage that simply goes from zero, when inactive, to a higher voltage when active. Typically the gate signal will rise to +5V or higher. The gate is normally derived from a keyboard or a midi-CV convertor. The midi-DAC puts out a +5V gate as standard. A positive voltage at the gate input will turn on Q3, and take its collector down to about zero volts. Q4 will then be turned off and its collector dragged up to 8.2V by R21. This positive going signal is a copy of the input gate signal, but it has been conditioned so that the following circuits will be able to use it. C7 differentiates this signal to cause Q6 to briefly switch on, puling its collector down to ground for a very short while. This is the trigger pulse that activates the main timer IC, the LM556.

R5 provides a bit of positive feedback forcing Q3 to switch on faster. This should allow slow moving CV inputs to trigger the EG more reliably. D1 provides protection against negative input signals damaging Q3.

The key to understanding this EG is to understand the operation of the LM556. This is a dual timer, and is in fact, two of the ubiquitous LM555 timers in one package. The topper most half of the LM556 is concerned with generating the AD output. This is the output that will rise to a certain level at a rate determined by the attack, A1, pot and then falls back to zero at a speed governed by the decay pot.

When the falling-edge of the trigger signal is received by the LM556 at pin 8, the output at pin 9 will rise rapidly to the IC's supply rail, namely 8.2V. Current will flow from this pin via D2 and through A1 into the timing capacitor, C6. The voltage across C6 will rise, quickly at first, then getting slower. The overall rate of increase will be determined by A1 and R16. R16 is set so as not to cause excess current flowing from pin 9 and damaging the LM556. One half of the FET op-amp, TL072, sniffs at the voltage on the capacitor and its output follows this voltage precisely.

The op-amp's output is the AD output we want, but it is also fed back into the timer IC at its threshold pin, pin 12. When pin 12 reaches two thirds of its supply voltage, about +5V, it will do two things. Firstly, pin 9 will go low, and current no longer flows through A1 into C6. The attack phase is now over. Secondly, the discharge pin, pin 13, is rapidly dragged down to zero volts. The decay pot is connected to pin 13 and so current now flows from C6 through the decay pot to ground. The voltage across C6 will fall, and the op-amp's output will follow this fall. D2 prevents any current from leaking into the now low pin 9 output. When the voltage finally approaches zero, our AD output cycle is now complete.

If at any time in this cycle the gate signal is removed, the reset pin, pin 10, will detect this and the discharge phase is invoked prematurely.

The AR output is generated in a similar to the AD, but there are some crucial differences. We will look at these later. Let us first consider what happens in an ordinary AR-EG, because the Oakley Little-EG is different. Normal AR-EGs produce a rising attack phase that levels off at the sustain voltage. This is normally set by the voltage at the 'hot' end of the attack pot. In other words setting the asymptote of the charging curve. However, in theory, the capacitor never actually gets there, it just tends to that value. What this means is that the voltage is still rising after a considerable time. When this sort of EG is used to drive a VCA, the output lacks the punch needed in fast attack sounds. My EG is different and in some ways is more similar to a full ADSR. I use a 8.2V charging voltage, but I enable the sustain portion of the signal when the output rises to just 5V. This gives a clean definable attack portion, and also allows the AR envelope to follow the AD output.

The LM556 is triggered with a sharp pulse at pin 6, the attack phase begins as before, with C1 charging through A2 and R18. When the threshold is reached, at around +5V, charging stops. However, C1 is not discharged while the gate remains high. Instead is kept 'topped up' to the threshold voltage. The threshold voltage is set inside the LM556 by three identical resistors. We cannot change these resistors, but we do have access to the junction of the top two at pin 3. The voltage at this point sets the threshold voltage to the 556's internal comparator. We use the 'discharge' output, pin 2, to control the base of Q1. It is not used to actually discharge the timing capacitor. When the base of Q1 is pulled low, the threshold voltage at pin 3 is passed via R14 and the release pot to C1. This keeps C1 at the threshold voltage during the time that gate is active but not during the attack phase. I think this is the first time that this technique has been used. When the gate is removed, Q5 turns on, and C1 is discharged to ground via the release pot.

In practice, the rather high output resistance, about 4K, of pin 3 creates a rather strange dip and rise in the output waveform when the sustain portion is initiated. However, it is a small perturbation, and is not audible in VCF or VCA applications.

An LED may be fitted to indicate the state of the gate input. Q2 performs the required switching operation. R9 is set to give roughly 6mA of LED forward current.

Components

All of the parts should be easily available from 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, CPC and Rapid Electronic sell the Omeg pots at a very good price. Omeg will also sell direct, but this is only viable for large orders. All three suppliers have web sites. The pots recommended are 1M log or 470K log. You will be ill-advised to get linear pots, these make adjusting the rates very difficult at the fast end.

For the four pot version we use four **single gang 1M log** types. For the two pot version we use **dual gang 470K log** types. This lower value in the two pot version will reduce the maximum attack and decay times to about 5 seconds. So to compensate, you could use a 15uF, 16V tantalum capacitor for C1 and C6 if you wished.

For the resistors 5% 0.25W carbon types may be used for all values. But I would go for 1% 0.25W metal film resistors throughout since they are very cheap these days, and are more useful for any other Oakley projects you may want to build.

All the electrolytic capacitors should be 25V or 35V. C8 can be 16V since it will only see 8.2V across it. All electrolytics should be radially mounted, ie. their legs stick out of the bottom of the component housing.

As with all new Oakley board designs, the pitch spacing of the nF valued non-polar capacitors is 5mm (0.2”). The old style 7.5mm types are now getting harder to find. The new types come in little plastic boxes with legs that stick out of the bottom. They come in a variety of styles and names, but they should be described as metalised polyester types. You can also use axial multilayer ceramic types too. I like these for non critical positions due to their small size. For either type try to get those with an operating voltage of 63V or at the most 100V. Higher operating voltages can be used but will probably be too large to fit on the PCB. 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.

For the suggested layout you also need an LED. Feel free to use any colour LED, but I will be using amber or orange in my pre-built and pre-populated units. I use 5mm bipolar LEDs with suitable LED holders. I use bipolar LEDs 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. I use yellow clips with the amber or orange LEDs for this module.

It is best to solder the LEDs in after you have fitted the board to the panel. 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 BC550 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). Likewise, the BC560 can be any PNP type that conforms to the same pinout. For example, BC559, BC558 etc.

D4 is a 8.2V zener diode. The parts list specifies 400mW, but 500mW devices can also be used.

All ICs are dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP, CN or a N in their part numbers. For example; TL072CN and NE556N. 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 one of the sockets per EG must have a normalising lug if you want to use the Oakley CV/gate normalising scheme. The Switchcraft 112 types have normalising lugs as standard.

I am now able to supply four way MTA power leads at either 40cm or 60cm lengths.

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, 4n7 is a 4.7 nF capacitor.

Resistors

All 0.25W 5% carbon or better.

22R	R7, 8
100R	R16, 18
390R	R20
1K	R2, 4
2K2	R9

4K7	R11, 21
10K	R22
47K	R10
100K	R1, 12, 15, 17, 13, 14, 19
220K	R6
1M	R3, 5

Capacitors

22uF, 25V elect	C3, 4
220uF, 16V elect	C8
100pF ceramic low-K	C7
100n polylayer or multilayer ceramic	C2, 5
10uF, 25V elect**	C1, 6

** Can be swapped for 15uF, 16V tantalum for greater maximum attack and decay times.

Discrete Semiconductors

1N4148	signal diode	D1
1N4001	rectifier diode	D2, 3
8V2, 400mW	zener diode	D4
BC550	NPN transistor	Q2, 3, 4, 5, 6
BC560	PNP transistor	Q1
Amber 5mm LED		LED (not fitted directly to board)

Integrated Circuits

TL072CN	Dual FET op-amp
LM556CN or NE556N	Dual 555 timer

Miscellaneous

4-way 0.1" lockable header	PWR
Pot bracket for Eco pots	(2 off)
1M log Eco pot	(4 off) **For four pot version**
470K log dual gang Eco pot	(2 off) **For two pot version**

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

Building the Little-EG Module

Occasionally people have not been able to get their Oakley projects to work first time. Some times the boards will end up back with me so that I can get them to work. To date this has happened only a few times across the whole range of Oakley PCBs. The most common error with most 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!

I always using water washable flux in solder these days for my board manufacture. In Europe, Farnell and Rapid sell Multicore's Hydro-X, a very good value water based product. You must wash the PCB as soon as you stop 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. I usually put the board above a radiator for a few hours. 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 new type of 'no-clean' solder.

I have 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. However, I do not recommend sockets for any of the transistors. Even turned pin types.

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.

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. If you have bought axial multilayer ceramics, these look like small yellow resistors. They can be soldered in any way round, but will need their legs preformed before placing them in the board.

The ceramic plate capacitor, C7, looks like a small flat plate of ceramic. These can go in anyway round. But be careful not to bend them flat when you are washing or drying the board.

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 at this stage. The mounting of the pots requires special attention. See the next section for more details.

Mounting the Pots onto the Board

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 pins that connect to the pot. This will be just three pins for the single gang pots, or six pins for the dual gang ones. I normally solder the middle pin first and then check if the pot is lying true. It is particularly sensible if you are using dual gang pots to check that the pot is properly placed before you solder the rest of the pins. If it is not, simply reheat the middle pin's solder joint to allow you to move the pot into the correct position. **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 other wise you will deform the pot bearing. If the pot feels rough when you turn it, the chances are that you have tightened the pot nuts too tight. Normally backing off the nut a bit is enough to free the shaft. 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 470KB is a 470K 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.

Connections

This module is easy to connect up in either of the two suggested panel design options.

Option 1: The Dual-EG or 'four pot' 1U panel.

The board is to be fitted to the lower two of the four pot holes. The top two pots, the 'EG1-Attack' and 'EG1-Decay' will be connected to the board with wires. The board must be fitted so that the writing on the board is upside down when the module is held vertically. In other words the A1 and A2 pot positions are fitted to the third pot hold down.

Solder the pot brackets now if you have not done so already.

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 three sockets can be all connected together with solid wire. I use 0.91mm diameter tinned copper wire for this job. Its nice and stiff, so retains its shape. A single piece of insulated wire can then be used to connect those connected earth tags to the PAN pad.

Connect, with a piece of insulated wire, each **signal tag** on each socket to the respective pad on the PCB.

Socket Name	***PCB pad name***
Gate	GTE
EG1-AD	AD
EG2-AR	AR

The **NC tags** on these sockets are left unconnected. However, if you are using the Oakley CV-gate normalisation system, you will want to add the appropriate tail to your module. The EG requires a gate signal, and this is to be found on pin 3 on the 0.1" molex connector on the Dizzy board. The gate signal connects via the NC lug on the GATE socket. See the Dizzy User Guide for more information on this handy way of controlling your modular.

The other two sockets have no connection to their NC tags.

The LED is to be wired up next. If you are using Maplin Cliplites to hold your LED, then fit this now. Be careful that all four lugs go into the hole before you try to force the lens into position. The LED must be wired to the board with wires. Use a little heatshrink tubing to prevent the LED's legs from touching after you have soldered the wires in place. The cathode of the LED goes to pin 1 of the LED header. This is the square pin. Pin 2, the round one, goes to the anode. If you have used the recommended bipolar LEDs, you can fit your LED any way around.

The two pots must be wired up next. Both types are 1MB. Fit them so that their pins face down. Remember not to tighten the pot nuts up too much otherwise you'll damage the pot shaft bearing. Remember also that the washer goes on the outside of the panel, under the nut.

The EG1-Attack pot is wired so that the middle pin of the pot goes to the middle pad on A1. The right hand pin, looking at the back of the pot, is wired to the top pad of A1. The EG1-Decay pot is wired similarly. The middle pin of the pot goes to the middle pad of DEC. The right hand pin goes to the top pad of DEC.

You can use some cable ties to keep your wiring tidy if you wish.

Option 2: The Triple Dual-output EG 2U panel.

You need three complete Little-EG boards to make this module. Provision is made on the new Little-EG PCB to allow for a 'module power buss'. The top board only needs to be fitted with a MTA header for the power. The other boards deriving their power from the top one via the three conductor power buss.

Fit the top board first, and solder the pot brackets now if you have not done so already. Then fit the second one, and then the lowest one.

The power buss is very simple. It uses 0.91mm diameter wire that connects all three boards together. When the boards are in place and not likely to be moved again, slide a piece of thick solid core wire in the hole marked -15 and thread this though to the other boards. Make sure the wire is tight, and solder in place. If your wire is too thin, it may bend and touch the neighbouring buss wire. Even with 0.91mm wire I still use a bit of heatshrink tubing to provide insulation just in case.

Repeat this for the +15 and GND pads as well. The only disadvantage with the ‘power buss’ system is removing the boards should a fault develop. However, the easiest way to remove the board is simply to cut the wires off and repeat the process when you need to reassemble. It doesn’t take that long.

The sockets can be fitted next. You need nine sockets in total. 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.

For the Triple-EG panel, I fit the sockets so that the bevel is facing to the bottom left of the module as you view it from the rear. The ground tags of all the sockets can be all connected together with solid wire. I use 0.91mm diameter tinned copper wire for this job. Its nice and stiff, so retains its shape. Connect the sockets by using three vertical pieces of solid core wire, one piece of wire for one column of sockets. Then a fourth piece of solid wire can bridge all three vertical pieces. A single piece of insulated wire can then be used to connect those connected earth tags to the PAN pad. I use the top right ground terminal for this connection.

Using another piece of wire to link all the NC lugs on the right hand row of sockets. These are the gate inputs. If you are using the Oakley normalisation, then this is sufficient for now. If your modules are completely standalone, then it is a nice idea to link this row of NC lugs to the top right signal lug too. This allows for all EGs to be fired from one gate input. Inserting any jack overrides this ‘master’ gate signal.

Do not connect any of the other sockets’ NC lugs to anything.

Connect, with a piece of insulated wire, each **signal tag** on each socket to the respective pad for each of the three PCBs.

Socket Name	***PCB pad name***
Gate	GTE
AD	AD
AR	AR

If you are using the Oakley CV-gate normalisation system, you will want to add the appropriate tail to your module. The EG requires a gate signal, and this is to be found on pin 3 on the 0.1” molex connector on the Dizzy board. The gate signal connects via the three connected NC lugs on the GATE sockets. You can fit your tail on the lowest socket’s NC lug for ease. Make sure your tail is long enough to reach the Dizzy board. See the Dizzy User Guide for more information on this handy way of controlling your modular.

The LEDs are to be wired up next. If you are using Maplin Cliplites to hold your LEDs, then fit these now. Be careful that all four lugs go into the hole before you try to force the lens into position. The LEDs must be wired to the appropriate board with wires. Use a little heatshrink tubing to prevent the LED's legs from touching after you have soldered the wires in place. The cathode of the LED goes to pin 1 of the LED header. This is the square pin. Pin 2, the round one, goes to the anode. If you have used the recommended bipolar LEDs, you can fit your LED any way around.

You can use some cable ties to keep your wiring tidy if you wish.

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.

At the rear of this user guide I have included 1:1 drawings of the suggested front panel. Actual panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £12 for the 1U panel and about £20 for the larger 2U panel. All you need to do is e-mail the chosen fpd file that is found on the Little-EG web page on my site to Schaeffer, and they do the rest. You can also use the FrontPlatten Designer program to initiate orders too. The panel is black with white **engraved** legending. The panel itself is made from 3 mm thick anodised aluminium. The fpd panel can of course be edited with the Frontplatten Designer program freely available on the Schaeffer web site.

Final Comments

I hope you enjoy building and using the Oakley Little-EG module. 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 £20 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 gift if its a real howler.

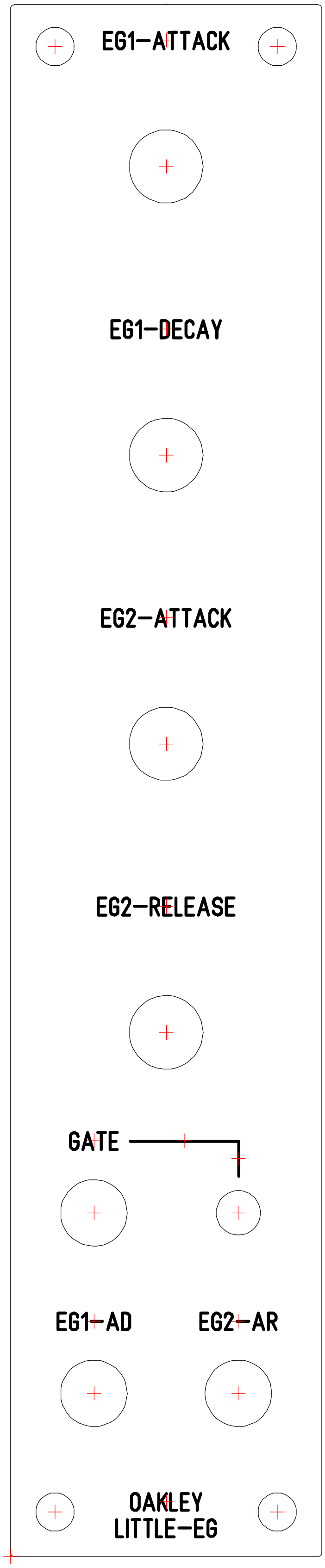
Please further any comments and questions back to me, your suggestions really do count. If you have any suggestions for new projects, feel free to contact me. You can e-mail, write or telephone me. If you telephone then it is best to do this on Monday to Friday, between 9 am and 6 pm, British time.

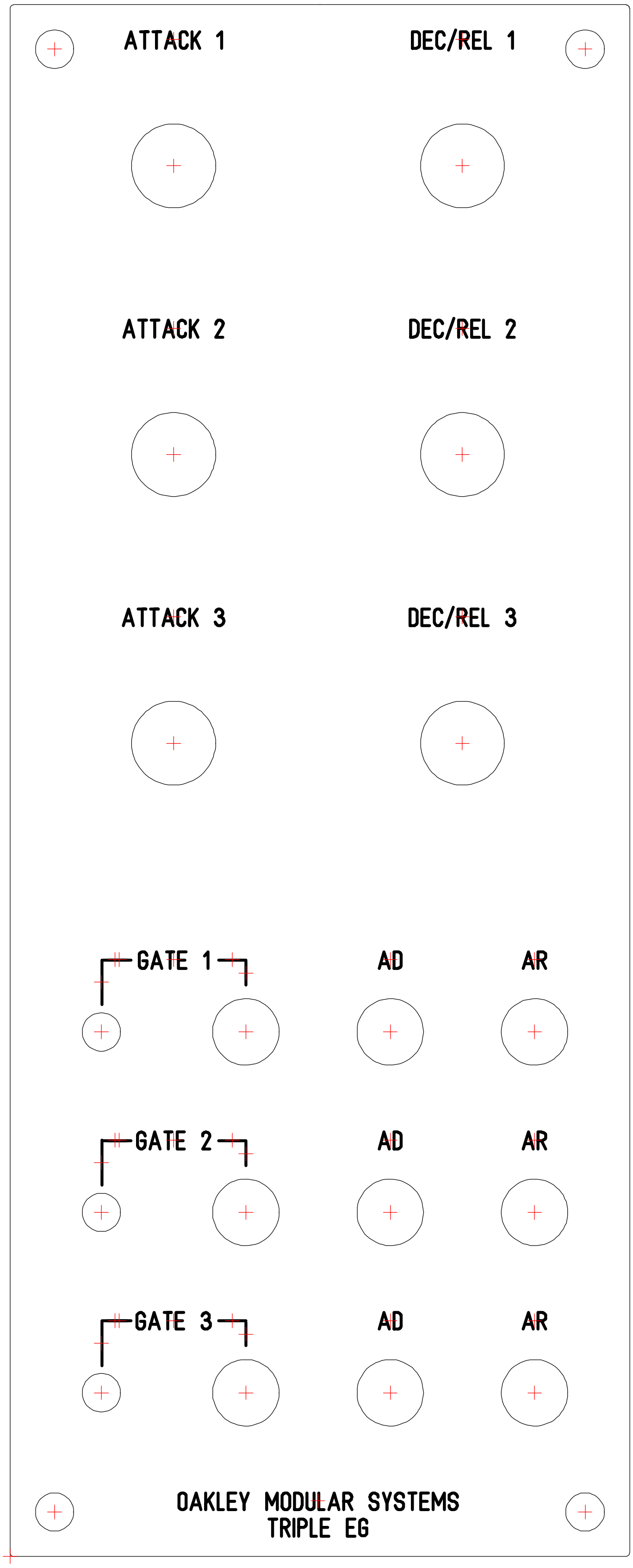
Last but not least, can I say a big thank you to all of you who helped and inspired me. Thanks especially to all those nice people on the synth-diy, Oakley-Synths and MOTM mailing lists.

Tony Allgood. April 2003

Version. 1.0

Formatted on Lotus Word Pro


EG1-ATTACK
EG1-DECAY
EG2-ATTACK
EG2-RELEASE
GATE
EG1-AD
EG2-AR
OAKLEY
LITTLE-EG


ATTACK 1
DEC/REL 1
ATTACK 2
DEC/REL 2
ATTACK 3
DEC/REL 3
GATE 1
AD
AR
GATE 2
AD
AR
GATE 3
AD
AR
OAKLEY MODULAR SYSTEMS
TRIPLE EG