

Oakley Sound Systems

OMS-410

PCB issue 3

**A companion module to the MOTM-410 Triple
Resonant Filter**

User's Guide

V3.0

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

Introduction

Please note: This User Guide is for boards marked Issue 3 only.

The OMS-410 is a companion module to the MOTM-410 triple resonant filter module. It adds powerful new features to the superb Synthtech module. No component changes are needed to the MOTM module, and the two modules are connected together by one, two or three removable connectors hidden behind the front panels.

The OMS-410 allows you to add five new features to your MOTM-410

- 1.** You now have three individual CV inputs to control each Vactrol element of the MOTM-410. Each input is sent through a reversible attenuator, with a control knob on the front panel. This allows incredible tonal sweeps and modulations to be created. Its like having three separate voltage controlled filters in one module.
- 2.** The MOTM-410's two internal VC-LFOs can now be accessed from the front panel. The OMS module shapes the internal LFO's outputs to give smooth sine waves. Two separate VCAs allow these LFO waveforms to be level controlled individually or together. You can now use the MOTM-410 as a twin VC-LFO, whilst still using the MOTM-410 as filter module.
- 3.** The three audio outputs of the MOTM-410 are combined to produce an outstanding stereo output. These outputs are especially designed to drive long cables and can be tailored to produced ordinary line levels for direct connection a to a mixer.
- 4.** A pad switch can be added to select audio line signal or modular signal levels to be used with the MOTM-410.
- 5.** An inverting amplifier is included on the board to allow for additional signal processing. This could be used to invert LFO2's output as on the MOTM-410.

The module is designed to fit into a 1U MOTM style panel. It requires the usual MOTM or Oakley power supply.

The OMS-410 and the MOTM-410 are connected together by one, two or three connectors. The number of which is determined by the options chosen. The connectors are 0.1" Molex or MTA and can be simply removed if the modules need to come apart at anytime. The wires from the connectors are soldered to special key locations on the MOTM-410's PCB. No component changes are required to the MOTM-410, unless you are fitting the pad switch, where one resistor needs to be snipped out.

Of Pots and Power

There are three main control pots on the PCB. These are the reversible input attenuators. 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.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 20mA. 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.

Circuit Description

As with many electronic circuits the OMS-410 can be split up in several little chunks. Grab hold of the schematic now and let's run through each bit in turn.

The reversible attenuators are three identical circuits, each based around one dual op-amp. Since they are all the same, we will concentrate on just the first one.

IN1 is the signal input coming directly from the input socket on the front panel. R34 provides a high resistance pull down to prevent static damage. The signal now goes two ways. The first is through U6 (pins 1, 2, 3). This is configured as our old friend the voltage follower. The circuit sniffs the input voltage, and presents the same voltage at its output, pin 1. Current may be taken out of this output without affecting the input. It acts as a buffer to the signal.

The second path is through the other half of the op-amp. This part is wired as an inverting amplifier with a gain of -1. The minus sign indicates an inversion. For example, an input signal of +2V is turned into a voltage of -2V at pin 7.

The IN-1 pot simply allows the wiper to be moved to either the output of the inverting amp or the output of the buffer amp. When it is turned towards the inverting amp, the signal at its wiper is derived solely from the inverted signal. When it is turned towards the buffer, the signal at the wiper is merely a copy of what is at the input. At the middle of the pot's track the outputs from the two op-amps will meet and completely cancel. Therefore, when the wiper is turned so that it sits in the middle of its travels, there is no output signal at all.

R44 turns the voltage at the wiper to a current to then be sent to the Vactrol controlling node on the MOTM-410. Each of the three reversible attenuator's current outputs are sent to the MOTM module via a five way interconnect, labelled '410-CV'. The remaining two poles on this header are used to transport the MOTM's LFO outputs to the sine shapers and VCAs in the OMS-410.

To create the stereo output we need a stereo mixer. U3 performs this simple function very well. The top half of U3 deals with the left hand signal, while the bottom half of the op-amp deals with the right hand side. Of course, there is no difference between the two sides, but we will stick to this nomenclature to avoid wiring confusion.

The individual band pass outputs of the MOTM-410 are taken via a three way interconnection to the '410-AUD' header. 'OUT 1' goes to the left hand side and is mixed with a proportion

of 'OUT 2'. The voltage gain of the mixer is set to be -0.1 for 'OUT 1'. This will reduce the very hot MOTM and Oakley Modular levels to something more like ordinary line levels. Change R22 and R23 to 100K if you wish to retain the 'hotness' of your signals. Note that mixer is configured to invert the signals. The inversion can be tolerated since the band pass outputs are already inverted compared with the audio input to the MOTM-410. Thus the further inversion merely turns all signals the right way around again.

The right hand signal mixes 'OUT 3' with a smaller proportion of 'OUT 2'. Thus, 'OUT 1' is panned hard left, 'OUT 3' panned hard left and 'OUT 2' is sent equally to left and right. To keep the overall volume levels the same for each output, 'OUT 2' must be about 70% lower than the other two, since it is being sent to both outputs.

C5 and R23 (and C4 and R22) form a compensation circuit to allow the op-amp to drive fairly high capacitive loads. It has no effect on the audio performance, but reduces any chance of high frequency ringing in long cable runs. A full explanation of this is beyond the scope of this user guide, but more information can be found on the Analog Devices website in their Applications Note AN-257. I was first introduced to this method when I worked at Soundcraft, and have since used it on other Oakley projects notably the VCF-1 filter rack's main output with great success.

The MOTM-410 incorporates two voltage controlled low frequency oscillators (VC-LFO). These are controlled by one control voltage (CV), but are essentially separate in every other way. The OMS-410 uses the two triangle output waveforms of the MOTM's VC-LFOs to create two sinewave outputs that can be controlled in level by another CV.

Each sine shaper and voltage controlled amplifier (VCA) is identical, although the suggested panel layout actually combines the level CV for both VCAs. The design is almost identical too to that found already within the MOTM-410.

LFO1 is the input signal derived from the triangle output of the VC-LFO. It is first attenuated by R41 and R42 before going into U7 the VCA. The amount of attenuation is chosen carefully so that it deliberately overdrives the VCA's input circuitry by a small amount. This overdrive produces a rounding off of the triangle's sharp edges creating a sine-like waveform. It is not perfect, but good enough for control purposes in a synthesiser.

The VCA is not really a VCA as such, its an operational transconductance amplifier (OTA). Its a sort of very basic op-amp with some crucial differences. The output signal is a current and it can be fed into the inverting input of an op-amp inverter (U5) to get a voltage. R33 sets the transimpedance of the inverter. In other words how much current produces how much voltage. The output is sent via the usual 1K output resistor to the front panel socket.

The gain of the OTA can be changed and is set by the current into its Iabc pin. In this case, pin 1. R14 turns the input CV into a current which is level shifted by Q1 to drive the OTA. D1 prevents any negative going CVs from damaging Q1. R15 and R16 allow a set amount of current to flow into Iabc, when there is no jack connected to the CV in. This means that you will always get a LFO output from the OMS-410, unless you deliberately force the input CV to zero volts with an external signal. This disadvantage of this system is that the input resistance is quite low, about 10K or so. This shouldn't bother most people. If you are

worried by this you can omit R16 and make R15 a 62K resistor, which should still work. If you are fitting the second CV control, then do the same for R18 and R19.

The second VCA and sine shaper can be controlled by its own CV input circuitry, based around Q2. However, in the suggested layout only one CV level input is available due to limited front panel space. Here a wire link, LK, is fitted and the two VCAs operate together. The CV2 circuitry should not be fitted if this is the case. If you are fitting the second CV input, then make R32 and R33 22K resistors. This will prevent the LFO outputs being too large with Iabc not being shared by the OTAs.

Also included on the OMS-410 board but not on the suggested front panel is an additional inverter circuit, based around U1. This will allow you to create a third LFO output that will mimic the inverted LFO in the MOTM-410. However, if you have the panel space you could also use it as a separate inverter circuit for any other CV or audio signal.

There is one bit of circuitry left in the OMS-410 to explain. Its just a simple resistor which, in conjunction with a front panel switch, allows you to select the input pad on the MOTM-410. The resistor is R5 and is connected to the switch via solder pads SW1 and SW2. The 2-way header 'PAD' is the interconnect between the MOTM and OMS modules. If this is to be used you must snip, or not fit, the optional resistor on the MOTM-410. The exact connection details will be given later in this document.

Feel free to make any changes to the circuit to improve or alter the performance. If you do, do drop the Oakley-Synths mailing list an e-mail and tell us all what you have done.

Buying the Components

Most of the parts are easily available from your local parts stockist. I use Rapid Electronics, RS Components, and Farnell, here in the UK. The Equinoxe was designed to be built from parts obtainable from Rapid Electronics and Oakley Modular. Rapid have an online ordering service that is very good.

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.

The PCB is designed for Spectrol 248 series pots with 1/4" shafts. These are high quality sealed conductive plastic potentiometers. Rapid, CPC, RS and Farnell sell these parts in the UK. The pot brackets are especially made for us, and are only available from Oakley Modular. We 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.

The two electrolytic capacitors (abbreviated to 'elect' in the parts list) should have a working voltage of 25V or 35V. They should be radially mounted.

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 100nF capacitors I would use 63V metalised polyester film types. These types of capacitors come in little plastic boxes with legs that stick out of the bottom.

The other capacitors are low value ceramic capacitors. They have a lead spacing of 5mm. 0.1" types can be used if the leads are pre-bent with care before putting them into the board. Try to get Low-K or C0G ceramic plates if possible. Do not use those horribly cheap high-K types. These will result in a noisier audio output.

L-1 and L-2 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. Rapid sell them as part number: 24-4860.

The BC560 transistors can be any general PNP transistor that has the same EBC pin out. Another example is the BC559. The suffix A, B or C may be used. For example, BC559B. The OMS-410 is able to use any grade, so feel free to use any suffix or ones with no suffix at all.

All ICs are dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP or a CN in their part numbers. For example; TL072CP. Do not use SMD, SM or surface mount packages. You can use OP-275, TL072 or NE5532 for U3. The OP-275 is specified for low noise and small DC offset.

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 three of the sockets must have normalising lugs. The Switchcraft 112 types have normalising lugs as standard.

U7 is specified as a LM13700, but you can also use the older LM13600 or the now defunct Phillips' part, NE5517. All will behave identically in this application.

UK builders should know that there is now a 'Oakley Preferred Parts List' online. This gives the part codes of our most used parts. It can greatly speed up ordering times. This can be found at www.oakleysound.com/parts.pdf.

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

VERY IMPORTANT: This is the parts list for the suggested 1U panel design. Some components will seem to be missing from your board once you have finished populating your board. This is intended, since these parts are used in the 2U panel version. Please see later for the additional parts required for this particular module. However, builders of the 2U panel should note that R32 and R32 are 22K and not 47K as in the suggested module.

For builders of the 1U Stooge panel OMS-410, you do not have the stereo output mixer section. You can either build this version and simply leave this part of the circuit unconnected. Or you can omit the following parts:

U3, C4, C5, R24, R25, R10, R11, R12, R13

The components are grouped into values, the order of the component names is of no particular consequence. Also note that I have standard rules for describing part values, ie. 4R7 is 4.7 ohms. 4K7 is 4 700 ohms.

Resistors

Resistors 1/4W, 5% or better.

4R7	R29, 37
100R	R24, 25
1K	R20, 21
2K2	R42, 43
10K	R26, 35, 6, 22, 23
22K	R14, 16, 30, 31
33K	R15
47K	R32*, 33*
100K	R13, 27, 36, 38, 10, 7, 9, 28, 45, 41
150K	R11, 12, 39, 40, 44
220K	R1, 8, 34

* Suggested 1U panel design only.

Capacitors

22uF, 25V electrolytic	C7, 10
100nF, 63V polyester	C1, 3, 8
100pF ceramic low-K	C6, 9
33pF ceramic low-K	C4, 5

Discrete Semiconductors

1N4148 or 1N914	D1
BC560	Q1

Integrated Circuits

TL072	U2, 4, 5, 6
OP275	U3
LM13700 or LM13600	U7

Other

4-way 0.156" Molex/MTA connector	PSU
5-way 0.1" header and plug	410-CV
3-way 0.1" header and plug	410-AUD
Wire link, eg. resistor lead clipping	LK for suggested 1U panel design only.
50KA linear single gang variable resistor	3 off
Three pot brackets to suit	
Leaded or taped ferrite beads	L-1, L-2
1m of multistrand hook up wire	
Three knobs	
Eight decent quality jack sockets, eg. Switchcraft 112	

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

Additional Parts needed for the OM-410 2U panel design

Do not fit LK1, leave space blank.

Resistors

100R	R2
10K	R5
33K	R19
47K	R3, 4
22K	R17, 18, 32, 33

Capacitors

22pF ceramic	C2
--------------	----

Semiconductors

1N4148 or 1N914	D2
BC560	Q2

Integrated Circuits

TL071	U1
-------	----

Other

SPDT switch for PAD	
2-way 0.1" header and plug	PAD
8 pin DIL socket	U1

Building the OMS-410

Warning:

Oakley Modular PCBs are now supplied with the RoHS compliant Ni/Au 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 Ni/Au 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. 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!

For construction of the PCB I now use 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.

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. Also, make sure that any water drops left between the pins of the sockets are fully dried up before switching the board on.

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 capacitors are like little blue or red boxes. Push the part into place up to the board's surface. Little lugs on the underside of the capacitor will leave enough of an air gap for the water wash to work. Cinch and solder the leads as you would resistors.

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

I would make the board in the following order: resistors, diodes, IC 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

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	3 off

Fit the pot brackets to the pots by the nuts supplied with the pots. You should have two nuts and one washer per pot. Fit only one nut 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 itself. Do this for all three pots.

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. The washers should go on the pot's bush at the front of the module and the second nut on top of this. Again, do not over tighten.

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.

Connections

There are several options for this module. At the back of this document I have included three examples of front panels. The first one is the suggested layout and is the one shown on the website photographs. It has the three reversible attenuators, two LFO outputs, but with only one VC level control input, and includes the stereo mixer. The second one, the Stooze variant, has MOTM standard socket spacing. This means it has two sockets less than the suggested

layout in which we lose the stereo mixer. The final one has every option available and is 2U wide.

All the options have things in common. They all need power, and they all use the '410-CV' header.

The power socket is 0.156" Molex/MTA 4-way header. Friction lock types are recommended. This system is compatible with MOTM systems.

<i>Power</i>	<i>Pin number</i>
+15V	1
Module GND	2
Earth/PNL	3
-15V	4

The PNL pad on the PCB has been provided to allow the ground tags of the jack sockets to be connected to the power supply ground without using the 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 used Switchcraft 112 sockets you will see that they have three connections. One is the earth tag. One is the signal tag which will be connected to the tip of the jack plug when it is inserted. The third tag is the normalised tag, or NC (normally closed) tag. The NC tag is internally connected to the signal tag when a jack is not connected. This connection is automatically broken when you insert a jack. In the OMS-410, we have no use for the NC lugs.

The ground tags of each socket can be all connected together with solid wire to make a simple frame. A single piece of insulated wire can then be used to connect all the tags to the PNL pad on the PCB. Do not connect the ground tags to any other ground.

The suggested layout uses eight sockets, and wiring them up is straightforward enough. Use multistrand hook up wire to connect each socket's signal lug to the relevant pad on the PCB. There is no need to use screened cable for such short runs. Keep the wires short but not taut. If you do insist on using screened cable then connect the screen to the socket's ground lug. Leave the screen unconnected at the PCB end. Use a bit of heatshrink tube to keep the end from fraying.

The PCB uses slightly different nomenclature for the sockets and pads. The following table shows which one is connected to which.

PCB pad	Front panel socket
IN1	CV1
IN2	CV2
IN3	CV3
LFO1	LFO OUT 1
LFO2	LFO OUT 2

CV1	LFO MOD
L	MIX L
R	MIX R

Right you have now completed the OMS-410. Now to wire the interconnects. You need a 5-way 0.1" female plug. These are normally designed to be crimped but they can be easily soldered with care. Make each wire to be 25cm long. I normally strip back the wire by just 2 to 3mm. Place all the bare wire into the crimp on a heatproof surface. I use 12mm MDF board to protect my bench top, which although not at all burn proof will take plenty of heat from a soldering iron without damage. Rest a pair of pliers on top of the wire to hold it in place and then solder it in. Do not use the water washable flux solder in this application. You must use either good old fashioned ersin or rosin flux based solders or the newer so called 'no-clean' types. I actually prefer the rosin based ones for this because I find they flow better. Once you have soldered it, wait a bit for it to cool, and then push it into the housing until it clicks. If it doesn't go in, then take it out and bend the crimp slightly backwards. Now try again.

I like to use as many different colour wires as possible. It makes wiring easier and looks great. A good source of cheap insulated multistrand wire is 'def-stan' 8-way multicore cable. You'll have to strip off all the outer casing, but its actually cheaper, than buying separate reels of each colour. The type I use is the one with eight 7/0.2mm insulated wires.

For the 410-CV interconnector you need to make the last two wires, pin 4 and 5 of the connector slightly shorter. Cut them down to 20cm only. Each wire now needs to be soldered onto key points on the MOTM-410 PCB. Trim back the loose end of each wire to leave 3mm of uninsulated wire. Tin the end to make sure they are going to solder well. Solder each end onto the top side of the MOTM PCB by adding just a slight amount of extra solder to secure the wire. The wires must be put into the correct positions for your module to work properly. Hold the MOTM module so that the panel is on the right, and the PCB is facing you. We will deal with each pin of the 0.1" socket in turn.

Pin 1: connect to the right hand end of the 2K7 resistor that is next to Q7. Its R54 but its hard to see.

Pin 2: connect to the right hand end of the 2K7 resistor that is next to VR3 and below R56.

Pin 3: connect to the right hand end of the 2K7 resistor that is above VR4. It is labelled R60.

Pin 4: connect to the right end of the 33K resistor next to U5. It is labelled R33

Pin 5: connect to the left hand end of a 33K resistor next to U5. It is not labelled, but it is below R20 and above R34.

That is all the CV inputs and LFOs sorted. Now we need to connect up the stereo mixer. You need to connect up the three way 0.1" Molex, '410-AUD'. Again use 25cm long wires for each wire.

Pin 1: connect to the right hand end of R3 down the bottom.

Pin 2: connect to the right hand end of R6.

Pin 3: connect to the right hand end of R9.

That completes the wiring of the interconnects for the standard suggested panel. You are now ready to go and try it out.

For the second panel design, we are not using the stereo mixer. Thus you can omit the following parts on the PCB; U3, C4, C5, R24, R25, R10, R11, R12, R13. We have no need for the 410-AUD interconnect either, and only use the 410-CV interconnect.

For the third design, everything that can be added has been. It needs a 2U wide panel, and features the pad switch and inverter circuit. Don't forget all component locations are filled on the PCB, except LK which is not needed. Use 22K resistors for R32 and R33.

The interconnects are wired as the suggested layout but you also need to wire up the PAD interconnect too. This is a simple two way header. Make up the wires as before, again, 25cm long wires should be ample. This time twist the wires together tightly to form a twisted-pair cable.

On the MOTM module you need to snip out R68 if you have fitted it. Now connect one of the wires to the one of the pads where R68 used to be, and connect the other wire to the other end of where R68 used to be. It doesn't matter which one is which.

Panels can be obtained from Schaeffer-Apparatebau of Berlin, Germany. The cost is about £15 per 1U panel, and about £20 for the 2U one. For the suggested layout all you need to do is e-mail the fpd file that is found on the OMS-410 web page on my site to Schaeffer, and they do the rest. The panel is black with white **engraved** legending. The panel itself is made from 2.5mm thick anodised aluminium. The fpd of the panel can be edited with the Frontplatten Designer program available on the Schaeffer web site.

Final Comments

I hope you enjoy building and using your OMS-410.

If you have any problems with the module, an excellent source of support is the Oakley-Synths Group that can be found at <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 and questions are important to both Oakley Sound and Oakley Modular. In the first instance, please use the Oakley Synths Group where a wealth of experience resides.

Please do not contact me or Oakley Modular directly with questions about sourcing components or general fault finding.

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 Oakley-synths, Synth-diy and MOTM mailing lists.

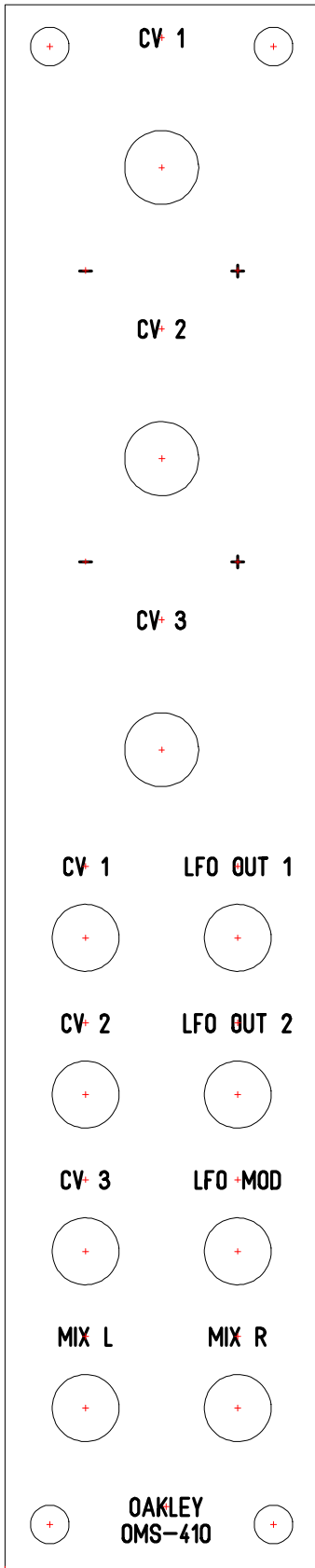
Tony Allgood
Cumbria, UK January 2007

Version 3.0

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.

Copyright belongs to Tony Allgood. No part of this document may be copied by any means for whatever purpose other than to build this unit.



CV 1



CV 2



CV 3



CV 1



LF01 OUT



CV 2



LF02 OUT



CV 3



LFO LEVEL



OAKLEY
OMS-410

