Oakley Sound Systems

Power Supply

PCB issue 1

+/-15V Regulated Power Supply

Builder's Guide

V1.5

Oakley Sound Systems
CARLISLE
CA4 9QR
United Kingdom

Safety Warning

This supply has been designed to work with isolated low voltage AC inputs only. Connection to any other supply is done at your own risk. Low voltage is classified as being less than 25V with respect the ground potential. Voltages above this level can, and are, lethal to living creatures.

Oakley Sound Systems will not advise on building or modifying this board to allow for connection to the mains or other high voltage sources. Please do not ask us for information pertaining to direct mains connections or using internally mounted transformers as we will not give it.

We do not recommend powering this board from any other supply than those listed in this document.

Oakley Sound Systems are not liable for any damages caused by the misuse of this product. It is your responsibility to use this product safely. If you have any doubt about making a safe power supply, then do not attempt to build one.

Introduction

The power supply board will allow the conversion of a suitable low voltage alternating current (AC) to be rectified, smoothed and regulated for operation with the Oakley Modular.

It requires a split or centre tapped AC output and this obtained from a external power supply pack available from Yamaha. Other sources of AC can be an appropriate transformer mounted internally to the modular. However, details of this are not given in the document and for safety and legal liability reasons we do not recommend this method of connection.

The board has four mounting holes for stable placement onto the metal plate used for heat dissipation. The two power devices used in the regulation circuitry are connected directly to the board but their housings should be mounted onto the metal plate. In the standard fitting the metal plate is actually the 3U front panel, called the ‘Master Panel’, that will also house an optional jack multiple and power switch. A Schaeffer FPD file for the panel is available for download on the PSU webpage.

The module has been designed so that construction is simplified as much as possible. The hardest thing is mounting the two power devices which need a special insulation washer and bush. It is also possible to mount the power devices on a remote heatsink and have the regulator board fitted elsewhere.

The unit features active current regulation on its two outputs. The standard design has a current limit of around 650mA per rail and is fully short circuit proof. The current limit can be upgraded if you know what you are doing. Changing the value of one resistor on each rail will alter the current limit.

The power supply has two integral fuse holders in case of a problem with the power supply circuitry itself. Two on-board LEDs provide a quick visual reference that all is well with the supply outputs.

The standard circuit provides two outputs, one at +15V and one at -15V. Both output voltages can be finely adjusted with one trimmer.

The unit can be modified to supply +/-12V if the need arises.

Power input and outputs are available on screw terminals mounted directly on the board.

The PCB

The is 140mm x 80mm in size. It is double sided and features solder mask on both sides. The new issue board is RoHS friendly and is therefore lead free.

There are detailed instructions later in the document about how to build the board.

The Power Supply to the Power Supply

The standard method of powering the unit is from a low voltage tapped AC source. The safest available option is to use a ready made 'line lump' supply. A line lump is a half brick sized plastic box that has two leads attached to it. One end goes to your mains socket, the other will attach to the item to be powered.

The particular line lump supply we have chosen is different from a standard AC adapter. This one gives us two AC outputs with centre tap or mid point reference voltage. So unlike the usual AC adapter output with two leads, this one has three.

The one we are using is made for Yamaha and they are available from Yamaha spares departments. You should ask for the power pack that goes with the MG12/4 mixer. These are CE approved and connect to the mains via your local mains connector. They will be different types depending on the country you need them for. The output is rated at 17.5-0-17.5V AC at 0.94A maximum. It comes with a handy three way plug at the low voltage end that you can use with an appropriate socket. If you wish you can ditch their connector and use your own.

In the UK the line lump's part number is V9812300 and the cost is around £20 and postage. We do have permission from Yamaha-Kemble in the UK to use this particular part for the Oakley system, but in other countries this may be not so clear cut. The liability issue once again rears its ugly head and they may not want to sell power items for third party use. The best thing is to do is just to ask for a spare supply for your own MG12/4 mixer.

Circuit Description

The design is based around some very traditional methods so there are no claims for originality with this board.

Power enters the board via the screw terminal, IN. Pins 2 and 3 are commoned on the PCB although in the standard connection to the Yamaha supply we will only use one of the pins, pin 2. Note that if you are using an internal transformer, pin 2 and 3 can be used for commoning the two secondary windings.

The ‘centre-tap’ output of the line lump supply, ie, 0V, will connect to pin 2. Pins 1 and 4 will be connected to the two 'hot' AC outputs of the line lump. Or if you have fitted an internal transformer then pins 1 and 4 would go the ends of any connected secondaries.

The line lump outputs an AC signal with a peak of nearly 25V with respect to its centre tap output. The difference between the two phases is that one is completely out of phase with the other. Its a bit like an audio balanced output in that respect. Both phases should be present for the power supply to work correctly.

Each phase is separately fused. A value of 2A has been chosen for the fuses although a higher value should be used if you are planning to ‘upgrade’ the power supply’s current capability. The standard design when run from the Yamaha supply is designed to supply no more than 650mA or so. If you intending to draw more current than this, you will need to use a different source of low voltage AC.

Note that the fuses are anti-surge, or ‘slo-blo’. Either way, the standard convention is for a ‘T’ in front of the value, where T is for time lag. These fuses have a higher thermal mass than fast blow fuses to prevent them blowing when they see a short burst of current. The inrush current when the supply is first switched on is completely normal and we don’t want our fuses blowing at that point.

It is possible to run the Oakley PSU module on a single phase. This means that the unit could be connected to a single output AC adapter, ie. one with just two leads to pins 1 and 2. However, the regulators and the AC adapter will have to work very hard in this application. It is therefore not recommended to run the PSU in single phase unless the output current is going to be less than 250mA or so.

The raw AC is fed to a bridge rectifier based around D6, D7, D8 and D9. This is the classic bridge rectifier circuit. Although it is drawn somewhat differently in the schematic than the usual ‘bridge’ style. If you think about a diode as passing current through it just one way, you should be able to work out why the voltage across C9 ends up as only positive, and across C10 as only negative.

The small capacitors across the diodes act as little noise suppressors and they are sometimes called snubbers. Diodes tend to burst into oscillation as they switch from passing and then blocking current. The capacitors help keep things stable.

I have used 1N4001 diodes in this place. You should use ones with a higher current capability if intend to draw more current from your supply than the standard 650mA.

The outputs of the rectifiers supply current to the two smoothing capacitors. These act as reservoirs of charge when the AC voltage dips below its peak output. You can think about the rectifiers as merely topping up the reservoirs 100 times a second (120 in North America), whilst the capacitors actually provide the power to keep the modular powered.

The smoothing capacitors are generously rated both in terms of voltage and capacitance. It is essential that you use good quality components here and that they have sufficiently high ripple current rating. Since this PSU will be supplying 650mA, the capacitors will need to have twice that to be on the safe side. A cheaper and less rugged capacitor will overheat and won't last as long.

C7 and C8 are there to reduce any high frequency noise on the line. The bigger smoothing capacitors tend to stop behaving like capacitors at high frequencies and the little fellas can take over at that point.

The voltage across the smoothing capacitors is fairly constant but its not stable enough to drive a modular synth. Modular synths need to have a regulated supply so that they stay in tune and behave in a predictable fashion whatever patches are selected. The regulators smooth out the voltage across the big capacitors even more and will produce a constant, low ripple +15V and -15V output.

Both of the regulators work in a similar way. You can think of them like a person controlling the speed of a water wheel. If the wheel runs too fast, the water flowing over the wheel is cut back by closing the sluice gate. If it runs too slowly, then the water flow is increased by opening the gate. The person uses his own eyes to monitor the speed of the wheel, and will make adjustments to the sluice accordingly. Now the wheel speed will depend on two things primarily, firstly the water rate, but also the load on the wheel itself. If the wheel needs to do more work it will slow down unless the water rate is increased.

The regulated power supply must also control its output with changes of load [ie. the number of synth modules in your system] and the input voltage [ie. fluctuations in the line lump's output].

Before we talk about the circuit in more detail, I ought to say why I haven't used the more common and I might add cheaper 7815 and 7915 regulators. The main reason is down to controllable current limiting. The 78XX and 79XX series do not feature this. In fact, if you were to short out a 7815 to ground, it would take around 1.7A from the supply and would heat up until the inbuilt thermal limiter kicked in. 1.7A may well be high enough to damage any connected transformer or line lump, to say nothing of the rectifiers and smoothing capacitors that would have to supply this current. By using the LM723 and a few external components we put the control back in the hands of the builders of the circuit.

Let us look at the positive regulator first since this actually also controls the negative rail too. The circuit is built around the venerable LM723 chip. It contains pretty much everything you need to make a small regulator, including precision voltage reference, feedback control systems and a current limiter. However, it doesn't have a very big output current capacity so to use it for anything greater than 100mA, one needs to use an external pass transistor, Q1. The pass transistor is analogous to our sluice gate. The resistance of the pass transistor is effectively changed by the voltage at its base (pin 1). Decrease the voltage at its base with respect to its emitter (pin 3) and the transistor will increase its effective resistance and the output voltage of the PSU will drop.

The base voltage is controlled by the internal electronics of the LM723, but this in turn responds to the feedback from the output of the power supply. The feedback path is analogous to the sluice gate man's visual record of the water wheel's motion. In this case a fraction of the output signal is fed back into the LM723 at pin 4. The electronics will determine whether to make the base drive for Q1 higher or lower depending on what it 'sees'. The ADJ trimmer adjusts the fraction of the output voltage pin 4 will see and thus controls the overall output voltage.

R2 forms an important part of the current limiting circuit. When current travels through this resistor a voltage develops across it. When the voltage approaches 650mV, ie. 650mA through the resistor, the internal electronics in the LM723 starts to pull down the base voltage on Q1. This effectively lowers the output voltage to make sure the current doesn't climb much above that value.

The value of R2 will determine the actual value of the current limit. A 1R resistor will set the current limit to be around 650mA. A 0.5R resistor will make it around 1.3A. Simply chose a value of resistance that will develop 650mV across it when the current limit would be going through it. However, there are several things to be taken into consideration should you decide to do this:

Firstly, you need to ensure your transformer will be able to handle the additional current you are asking from it.

Secondly, the amount of current needed to drive the pass transistors is directly related to the amount of current draw on the supply lines. Thus to ask for more from the supply would be to ask more from the 723 on the positive rail and the op-amp on the negative rail. The actual relationship between load current and drive current is related to hfe or the gain of the pass transistor. Choosing a pass transistor with too little gain will mean that you cannot obtain your required current. The recommended TIP35 and TIP145 devices have been tested to work with a supply that can supply up to 1.1A, ie. R2 is 0.56R.

Thirdly, the printed circuit board's copper tracks are not thick enough to pass huge amounts of current. I have designed this board for currents no greater than 1.5A.

Now let us look in more detail at the negative part of the power supply. The negative supply works in a similar way to the positive regulator but uses an op-amp, U2, and discrete components. The reference for the negative rails comes not from a precision reference but from the output of the +15V regulator. This means that the negative output voltage will track the positive one, but not vice versa I should add.

The op-amp is wired as a simple inverting amplifier although its difficult to see this at first glance. The input is via R9 which is connected to the +15V output. The feedback is provided via R8 which is connected to the negative supply output. The output of the op-amp drives the base of the pass transistor, Q2, via R11, which in turn controls the level of the negative output. Q2, if using the recommended TIP145, is a PNP darlington transistor. This is actually two transistors in one enclosure – but it can treated as one device with a larger than normal current gain (hfe) and twice than normal base-emitter voltage of 1.2V. Having a high gain requires only a small amount of drive current from the op-amp via R11 even when the load on the power supply is relatively high.

U2 will act so that its output will force all the current flowing through R9 to be passed onto R8. This is one of the op-amp golden rules; that is, no current shall flow into the input pins so long as feedback is maintained. To do this the op-amp must force the pass transistor to pass enough current though it to establish exactly -15V at the output.

Q3 and R3 form part of the current limit circuit. If the voltage across R3 exceeds 650mV then Q3 will turn on dragging current away from Q2's base and thus lowering the output voltage of the negative rail accordingly.

D1, D2, D4 and D5 protect the +15V and -15V output rails from a variety of naughties. On power down any excess current left in the modular will be shunted to the smoothing caps and not damage the now unpowered power supply components. Also, they also prevent the negative rail from going positive should the negative rail die for some reason. And vice versa too.

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 power supply module was designed to be built mostly from parts obtainable from Oakley Modular and Rapid Electronics.

The resistors are either 5% carbon 0.25W types or the better quality 1% 0.25W metal film resistors. Personally I would go with the metal film types throughout, since these are very cheap nowadays.

The exception to this are the two current limit resistors. These are both 1R and should be rated at least to 0.5W or larger. I use 1W metal film types as they are small and compact. The pitch spacing of the board's pads is 18mm with a 1.1mm hole diameter.

The two large smoothing capacitors should be adequately rated for ripple current. In the standard build, you are expecting to pull over 650mA from each one, so get caps that have a ripple rating of over 1.5A. The working voltage should be sufficiently higher than the 25V the caps will probably see, so get ones at 35V or above. Bear in mind though, the bigger the working voltage the bigger the capacitor. The board has been laid out to accept some fairly big components, but be on your guard, power supply caps can get very big. The lead spacing is 10mm with a 1.1mm hole diameter. This means the board should accept most types of radial capacitor, although you may have splay the legs out a little on some types of capacitor.

Unfortunately, the PCB holes are not big enough to take the clip-in types. I used Rapid part number 11-0765 in the prototype which is a 2200uF, 35V. Alternatively, I could have chosen a 1000uF, 63V type which would have been equally as good.

The smaller electrolytic capacitors are radial types. These are the types that are cylindrical in shape with their legs sticking out of the bottom. I use bog standard 10uF, 35V types for this non critical situation.8

The PCB is another Oakley board to allow you to incorporate axial multilayer ceramics for the diode snubbers and power supply decoupling. These are good components with an excellent performance. Various types exist but I tend to use the better quality C0G types from Farnell. They look like small yellow or orange beads with wires that stick out on either side like a resistor. They normally come delivered on tape. You can use 63V or 100V types.

The two other small capacitors, C4 and C5, are ceramic plate types. These can be bought is various qualities, but you are looking for low-K. They are sometimes called NP0 or COG. The pitch (spacing) of the leads is 0.2” or 5mm.

The multiturn trimmer is the type that have the adjustment on the top of the box. Spectrol and Bourns make these. Some types are 22 turns, while others are 25 turns. Either will do. They should have three pins that are in a line at 0.1” pitch. Don’t chose the 10-turn ones with the adjustment on the end, they won’t fit on the PCB.

The two 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 the two devices used in the prototype the full component identification was LM723CN and OP177GPZ. Do not use SMD, SM or surface mount packages. You can also use the OP-07 or OP-77 for U2.

The two off board transistors are awesomely chunky devices in a TO-3P package. The devices I strongly recommend are TIP35C for the NPN and TIP145 for the darlington PNP. I buy the MOSPEC ones available from Farnell.

Originally when I designed this module I specified the Sanken 2SC4388 and 2SA1673 devices. These are similarly specified to the currently recommended devices but they also had insulated mounting surfaces. This means that you can bolt them directly to the metal heatsink or panel without any additional insulation or mounting hardware. Unfortunately, both of these devices have been made obsolete with the introduction of the RoHS laws in the EU.

If you are using the TIP devices you may also need two TO-3P mounting kits. This comprises usually of an insulating pad or plate and an insulating bush. The kit may also have a sachet of some heatsink compound if the insulation material is made from mica. Newer kits contain a pad made from a fibre glass cloth which is usually grey, but I have seen white and pale green ones too. This pad has two functions: Firstly it insulates the metal tab of the housing, which is internally connected to the collector of the transistor, from the panel or heatsink. Secondly it improves thermal transfer because it squashes itself into the two surfaces so it doesn't normally need any heatsink paste. The insulating bush is like a little top hat made from a very brittle plastic. This will sit in the hole of the device and insulate the mounting screw from the metal tab. The MOSPEC TIP35 and TIP145 devices need no insulating bush, only the insulating pad, since the mounting hole is already insulated as part of the device.

A small amount of heatsink compound will be needed if you are using the Sanken power devices to ensure that they transfer any dissipated heat to the panel. You can pick up small amounts of this at most computer places, but most of the mail order companies will sell larger tubes of it.

Alternatively, you can use any large power NPN and PNP device you fancy, but just make sure you have wired the correct pins to the right pads on the PCB. Remember, for the PNP device you need to ensure that the minimum hfe is sufficiently high. As we have seen this is because there is a direct relationship between the current draw on the power supply and the current that is needed to drive the transistor. I would recommend devices with hfe of 80 or more. Using a Darlington device for the PNP transistor will ensure that you have enough gain – although too much gain could lead to instability. Remember also that older TO-3 packages will need special mounting kits with bushes, solder tags and insulating pads.

The little zener diode is the standard 500mW device in an axial package, ie. BZX55. 400mW devices can also be used, as can the larger 1.3W types.

The two LEDs are standard 5mm types and they are designed to be board mounted for service use only. You can fit them both to the panel so they can be viewed externally. In that case you will also need some LED clips to hold them in place on the panel.

The board features three 4-way screw terminals. These are same types I use on the Oakley Dizzy board. This is a 5mm screw terminal connector. For example, Rapid part number: 21-0116. The connecting wires can be then fed into the appropriate hole and screwed into place.

The two fuse holders are 20mm fuse holders and you should get the ones designed to be soldered to the PCB. The lead pitch is 22mm. Rapid part number: 26-0164. You can also get little plastic covers for these to prevent accidental shorts to the fuse bodies. However, this is not really worth it, and there is a possibility of the plastic case fouling C15 and C14.

For mounting the PCB to the panel you will need some hardware. You will need six M3 countersunk screws, four of them at 20mm length, and two at 10mm. These will be fixing the board to the panel and the two transistors. Also, four M3 threaded hex spacers at 10mm in length. Six M3 nuts and ten M3 star washers to hold it all together.

If you are building the full Master Panel module you'll also be a decent power switch and a standby bipolar 5mm LED with suitable clip.

The Yamaha power supply features a 3-way locking power connector. The sockets to fit these into seem to be quite difficult to come by and you may wish to simply replace the supplied connector with one of your own choice. In the UK the only place I could find a suitable connector was Maplin. This once great mail order company is not the best place to get stuff these days, but if you ask for FM51F then you will have the perfect connector for the Yamaha line lump supply. Please make sure that if you do replace the socket on the Yamaha supply that you make careful note which of the now bared wires are which. Failure to do so will result in some unfortunate events.

The pre-built Master Panel module also features a jack multiple and this requires twelve Switchcraft 112A sockets as well as two resistors.

Finally, if you make a change that makes the circuit better, do tell the 'Oakley-synths' mailing list or myself directly. Any updates are added to the current user guide as quick as possible.

UK builders should know that there is a 'Oakley Preferred Parts List' online which is updated periodically by myself. This can be found at www.oakleysound.com/parts.pdf.

Parts List

A quick note on European part descriptions. To prevent loss of the small '.'as the decimal point, a convention of inserting the unit in its place is used. eg. 4R7 is a 4.7 ohm, 4K7 is a 4700 ohm resistor, 6n8 is a 6.8 nF capacitor.

Resistors

5% 1/4W carbon

1K	R1, R11
4K7	R10, R5, R4

1% 1/4W metal film

10K	R7, R9, R8
12K	R6

1W metal film (5% or better)

1R	R2, R3

Capacitors

18pF low-K ceramic plate	C4
330p low-K ceramic plate	C5
10nF axial ML ceramic	C14, C12, C13, C15
100nF axial ML ceramic	C8, C1, C7, C2
10uF, 35V electrolytic	C3, C6, C11
2200uF, 35V or higher	C9, C10

Integrated Circuits

LM723 100mA voltage regulator	U1
OP177 single precision op-amp	U2

Discrete Semiconductors

10V, 500mW zener diode	D3
1N4004 silicon diode	D8, D7, D9, D6, D5, D1, D4, D2
TIP35C or 2SC4388 NPN transistor	Q1
TIP145 or 2SA1673 PNP transistor	Q2
BC560 PNP transistor	Q3
5mm red LED bipolar	+VE
5mm green LED bipolar	-VE

Trimmer

10K multiturn trimmer	ADJ

Miscellaneous

2A antisurge 20mm fuse	F1, F2
20mm fuseholder PC mount	F1, F2
4-way screw terminal 5mm	IN, OP1, OP2

Heatsink compound if you are using the 2SC and 2SA power devices, or the TIP devices with a clear mica insulating pad.

TO-3P mounting kits (2 off) for mounting the TIP power devices.

Connecting wire of your choice. I use 22AWG (0.5 sq.mm) appliance wire for all power connections.

Mounting hardware

M3 20mm Countersunk screws	4 off
M3 10mm Countersunk screws	2 off
M3 star washers	10 off
M3 hex threaded spacers	4 off
M3 hex nuts	6 off

Optional Extras

Power switch DPDT	1 off	Power on
22K, 5% or better, 0.25W resistor	1 off	LED dropper resistor
5mm bipolar LED red	1 off	Standby Power
3-way connector	1 off	Connector to the line-lump

Jack Multiple, CV/gate and Attenuator Option

Switchcraft 1/4” sockets 112A	12 off
47K 5% or better, 0.25W resistors	1 off
4K7 5% or better, 0.25W resistors	1 off
3-way Molex/MTA housing	1 off
20AWG tinned copper wire	Approx. 1m

Populating the Power Supply Board

The Power Supply PCB is flashed with solder around the pads. This helps the soldering process and keeps the board solderable for many years. This flashing is with solder that contains lead. You should therefore wash your hands after handling the boards and do not place the boards in your mouth. It is also recommended that for best results this board is soldered with lead-tin solder.

Oakley Sound 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 errors are 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 use water washable flux in solder. The quality of results is remarkable. In Europe, Farnell and Rapid sell Multicore's Hydro-X, a very good value water based product. You must wash the PCB at least once every couple of hours 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 axial multilayer ceramics can be treated like resistors. Simply bend their legs to fit the 7.5mm (0.3") spacing holes.

The ceramic plate capacitors are like small, er... ceramic plates. They need to be treated with a little respect. Don't bend them to much once you have soldered them in. And do trim down the leads with wire cutters, even if they don't have that much to chop off.

IC sockets are to be recommended for the dual in line chips, 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.

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.

The big smoothing capacitors can now be fitted. Hopefully, they'll just slide into the board like the smaller caps. If you have to splay the legs a little this is fine, but don't force the cap too far down if the lead spacing doesn't match the board exactly. By the way, you really don't want to get these chaps in the wrong way.

The two LEDs can be fitted at this point. Simply drop them into the board and solder the leads. If you have used bipolar LEDs you will not need to worry about which direction the leads go in. If you have used conventional LEDs then make sure the cathode and anode go into the board the correct way.

The fuse holders and the three four-way screw terminals can now be fitted.

For the small BC560 transistor, 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 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.

Now you can do the final water wash if using a solder with water washable flux. Once the board is dry you can now solder in the trimmer. It doesn't matter which way around it goes.

Do not fit the two power transistors at this point this needs to be done with the panel assembly.

Front Panel

At the rear of this user guide I have included a 1:1 drawing of the suggested 3U Master Panel layout. Actual panels can be obtained from Schaeffer of Berlin, Germany, or Front Panel Express in the US. The cost is about £25 per panel. All you need to do is e-mail the appropriate fpd file that is found on the PSU web page on the Oakley Sound site to Schaeffer or FPE, and they do the rest.

The panel is black with white **engraved** legending. The panel itself is made from 3mm thick anodised aluminium. The fpd panel can be edited with the Frontplatten Designer program available on the Schaeffer or FPE web site.

Panel Assembly and Attaching the Power Devices

This section assumes you have purchased a 3U Schaeffer panel or similar.

For the four PCB mounting holes, insert a 20mm countersunk screw through each of them. Fit a star washer over each of the exposed screws. Now fit a hex spacer over each of the screws and tighten firmly. Check that the PCB fits over all four screws without too much of a fight, but don't fit the PCB permanently in just yet.

Now you need to prepare the leads on the two power transistors. The three legs need to be bent upwards so that the PCB can be fitted over them. Note that the top surface of the device is marked with the name of the component and it is the flat side on the bottom of the device that will be in contact with the panel. You should be able to see that the leads have a thicker section close to the body of the device. Make a 90 degree bend upwards at a point 1mm away from this thicker section. Do this for all three legs of the device.

For the TIP devices only:

Check that the mounting bush fits through the two holes in the panel that the power devices will be attached to. If it doesn't fit, don't force it in but widen the hole slightly with a reamer or suitable twist drill. Make sure all the swarf is cleaned away. Now place the bush into the hole of the transistor, with the flange of the bush lying on the top side of the device. Take one of the insulating pads and place it against the rear of the TIP35C. Match up the hole in the pad with the bush that is sticking out from the underside of the tab. Now place the power device, bush and pad flat against the rear of the panel so that the bush fits into the right hand side hole in panel. Make sure the pad does not slip out of place when you do this. Insert a 10mm countersunk M3 screw into the hole from the front, and fit a washer and nut onto the screw but do not tighten fully. Do the same for the TIP145 in the left hand hole.

Now if you have done all this correctly, you should find that the when the power supply PCB is fitted back onto the four screws, you can coax the power devices' legs through the respective pads on the board.

If you have MOSPEC TIP devices you do no need the insulating bush as the case itself has insulation around the mounting hole. You do still need the insulating pads.

For the Sanken devices only:

Using a little heatsink grease, gently smear the underside of both transistors. Place the 2SC4388 device flat against the rear of the panel so that the hole in the transistor corresponds to the right hand side hole in panel and the transistor's legending is the right way up. The grease should hold the device in place. Insert a 10mm countersunk M3 screw into the hole from the front, and fit a washer and nut onto the screw but do not tighten fully. Do the same for the 2SA1673 in the left hand hole.

Just double check that you have the TIP35C or 2SC4388 in the Q1 position and the TIP145 or 2SA1673 in the Q2 position.

Tighten up all the screws, and solder all the power devices legs into the board. You should snip off any excess lead lengths on the two transistors.

You might have noticed that one of the PCB mounting holes is connected to ground of the power supply. This ensures that the connected front panel is grounded.

Applying the Power

The Yamaha power supply comes with a three way connector which you can use or swap for a connector of your own choice. For the purposes of this document we will assume you will be using the supplied connector. The connector's pins are labelled 1, 2 and 3. Pin 1 and 3 are the main outputs of the supply and should eventually connect to pins 1 and 4 on the power supply PCB. Pin 2 is the centre tap and this will be connected to pin 2 on the PCB.

I mount the power in socket on the back of the modular or at the side. I do not favour having any sort of power connector on the front of the modular. It is unsightly to have large power cables entering the front of your modular and it's probably not that useful either.

The inlet socket will then pass power to the 3U master panel. The Oakley Master Panel incorporates a power switch. Note that this is actually called a standby switch since it does not control the power to the main line lump. The line lump is always on the moment it is plugged in. The standby switch simply allows the raw low voltage AC to be passed through to the power supply PCB. The standby switch is not to be used as the only method of switching your modular off for long periods. You must disconnect the line lump from the mains supply to ensure that all power is removed from your modular and power supply.

To utilise this sort of standby switch we will need to break the connection between the power in socket on the rear of the modular and the input screw terminal block on the power supply PCB. We need to use a dual pole switch. Pin 1 from the 3-way connector should go to the wiper of the left side of the switch. Pin 3 will go to the wiper on the right hand side of the switch. Pin 2 of the inlet socket can go directly to pin 2 of the IN terminal block on the PSU board bypassing the switch entirely.

The two spare lugs on the switch now need to be connected with a short length of wire to the PCB. The left hand lug should go to pin 1 on IN terminal of the PCB, and the right hand lug should go to pin 4.

To fit a standby LED on the panel is a little fiddly. You need to connect a **series** combination of bipolar red LED and 22K resistor across pins 1 and 4 of the IN connector. Use a little heatshrink sleeving to prevent any wires from shorting. You must use a bipolar LED here, a normal LED will not last long with AC running through it.

Jack Multiple and Attenuator Option

Fit twelve Switchcraft sockets into the spaces on the Master Panel. Align them so that the bevel faces to the top right as you look at the back of the unit. With a single piece of 20SWG tinned solid core wire connect all the earth lugs together on the right hand column. Now do the same on the left hand column.

Now connect all the signal lugs, marked as T on the body of the socket, on the first four of the right hand column of sockets. Then do the same on the left hand row.

You should now have four vertical wires, two running down the top four sockets, and two running down over six. Your multiples are now complete.

The KEY CV and GATE sockets now need to be connected to the Oakley Dizzy buss. Use a three way molex or MTA, as described in the Dizzy User Guide, to connect the signal lugs on these two sockets to the CV and Gate buss.

The 10:1 attenuator is simply made. Connect the signal lug on the right hand socket (IN) to the signal lug of the left hand socket (OUT) with a 47K resistor. Then solder a 4K7 resistor from the signal lug of the left hand socket to the earth lug of the same socket. And that's it!

Testing, testing, 1, 2, 3 ...

Note all testing must be done with the heatsink or panel attached to the power devices. Fit the T2A fuses into the fuse holders and power up the unit. Both LEDs should be lit and neither should be too bright or too dim.

Check to see if any components are getting warm. They shouldn't at this stage. If you can smell burning then switch off immediately.

With a multimeter, check the DC voltage across the +15V and -15V output terminals. You should get around 30V or so. Adjust the trimmer so that you get exactly 30.00V. Now measure the voltage across the +15V and one of the 0V terminals. This should read exactly 15.00V or very close to it. Check that the voltage across -15V and 0V is -15V.

If all is well, then its time to check the current limit. Put your multimeter into current mode. Use the 2A or 10A setting. Now put your probes across the +15V and 0V terminals. The meter should read 650mA or thereabouts. The red LED should have gone out. Now do the same for the -15V and 0V terminals. This time we should get a slightly higher reading of around 700mA. This time, the green LED should go out.

While doing the current limit check you may notice the power devices getting warm. This is perfectly normal.

If all is well, everything is probably working and you can now connect your new supply up to your modules.

Final Comments

I hope you enjoy building and using your Oakley PSU.

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. I am on this group, as well as many other users and builders of Oakley modules.

If you 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 e-mail 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 postage costs, any parts used and my time at 25GBP per hour. Most faults can be found and fixed within one hour, and I normally return modules within a week. The minimum charge is 25GBP plus return postage costs.

If you have a comment about this user guide, or have a found a mistake in it, then please do let me know. But please do not contact me or Paul Darlow directly with questions about sourcing components or general fault finding. Honestly, we would love to help but we do not have the time to help individually by e-mail.

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 Analogue Heaven mailing lists.

Tony Allgood at Oakley Sound

Cumbria, UK

Updated July 2009