Oakley Sound Systems
&
Modulus Electronics

polyDAC issue 1

4 channel Polyphonic midi-CV convertor

User's Guide

V1.51

Paul Maddox & Tony Allgood
Oakley Sound Systems
PENRITH
CA10 1HR
United Kingdom

Introduction

The Oakley-Modulus PolyDAC is a powerful and flexible module that will find many uses in your modular system. It uses commonly available parts, the only exception being the pre-programmed CPU. The whole project has been designed and programmed by Paul Maddox, with help from Tony Allgood.

The PolyDAC is a 3U wide module, this was done to accommodate all the outputs that would be needed.

Whilst the original intention was to design a 4 note polyphonic Midi to CV convertor, two extra modes were added to increase its flexibility:

Mono Mode 1:	All 4 'voices' have the same values for note, velocity and gate.
Mono Mode 2:	Each voices operates on a separate channel, allowing you to use your modular like a 'multi-timbral' synth via MIDI, without the need for 4 monophonic MIDI to CV modules.

The PolyDAC has the following outputs:

All Voices (1, 2, 3 and 4):

Gate (scaleable from 5V to 15V), Note (1V/Oct) Velocity (0-10v)

Individual outputs:

Name	***Range***
Pitchbend	-10v through 0v to +10v
Mod Wheel	0v through to 10v
After touch	0v through to 10v
Sustain	0v, 10V when on
CV A,B,C and D	0v to 10v

Also two knobs are available, 'Tune' and 'Bend range' these do exactly as it says, the TUNE is a global tune for all 4 NOTE CV outputs and the bend range controls how far the pitch bend voltage will go.

We also have 4 switches:

- Retrigger ;- When set to on and a new note comes in whilst an old one is held the gate will 'dip' briefly.
- Mode ;- Mono1, Poly , Mono2
- Poly 1 / 2 ;- (Poly mode only) This allows you to select what happens to incoming notes when you have 4 held down. Poly1 new notes are ignored, Poly 2 the oldest note is replaced.
- Assign ;- This allows you to assign MIDI CC's to the 4 'CV' outputs.

To use the ASSIGN; hold for approximately 2 seconds, the MIDI light will come on solid for a moment and then flash once. This will signify that it is waiting for the first MIDI CC (CV-A) to be assigned. Next send a MIDI CC command from your controller, the MIDI light will light solid for a moment, and then flash twice, it will now wait for the second MIDI CC to be assigned.

If you don't send anything for a while, the unit will exit the assign mode and wait for Midi data as per normal, without any changes being made to the current MIDI CC.

You have 4 assignable MIDI CCs, by default they are as follows A = 74 (cut-off) B = 7 (Volume) C = 5 (portamento) D = 2 (breath controller).

We also have a MIDI in and MIDI thru, along with 5 LEDs, one for each voice (lights when a voice is active) and a MIDI LED, this flashes whenever a command is received that the PolyDAC acts upon (note data/pitchbend/controller/etc)

The PolyDAC PCB is not only a large PCB (90mm x 120mm) but its also fairly dense as far as the number of components go. Because of this care needs to be taken when assembling the unit.

As with all Oakley projects the PCB is double sided with through plated holes, has a high quality solder mask both sides, and has component legending for ease of construction. The boards are electrically tested for defects. Please note that the PCB has been designed by me and not Tony Allgood. Thus it will have some different features to the usual Oakley PCB you may have used in the past.

Of Pots and Power

There are two control pots that are directly mounted on the PCB. If you use the specified pots and brackets, the PCB can be held firmly to the panel without any additional mounting procedures.

The board also had five LEDs which are mounted away from the board on the front panel. Flying leads are used to connect the LEDs to the board.

The design requires plus and minus 15V supplies. These should be adequately regulated. The current consumption is about 80mA per rail. Power is routed onto the PCB by a four way 0.156" Molex type connector. This unit will not run correctly from a +/-12V supply. There is a small 5V regulator on board and it should adequately handle the load needed, I have also taken great care to ensure that no 'digital' noise makes it way onto the main supply rails.

Circuit Description

The PolyDAC circuit is spread over four pages.

Page 1 shows the 'Digital' portion of the PolyDAC, it shows the CPU, crystal, MIDI interface, DAC and 5v regulator.

Page 2 shows part 1 of the analogue section. It has the Multiplexor, DAC Scale/Offset, 4 S&Hs and pitch bend circuit.

Page 3 (analogue section part 2) has the 4 Note S&Hs and summing.

Page 4 (analogue section part 3) shows the remainder of the S&Hs and the 4 'GATE' output sections.

Without going into too much detail about the CPU code and handling, the MIDI arrives via J7 and is fed into U1 (6N137) which is an opto-coupler, the output of this is fed into the CPU and also a 74HC00 which acts as a buffer to provide MIDI thru.

Once the micro receives the signal from the MIDI interface it decodes it and acts upon it accordingly. Only messages on the correct channel are responded to. The channel is set by SW1 (4way DIP switch) and a table for these can be found on the next page. Note that when in MonoMode2 this is the 'BASE' channel (the channel used by Voice 1), so if you have it set to channel 12, then Voice 1 will be on Channel 12, Voice 2 on channel 13, voice 3 on channel 14 and voice 4 on channel 15.

Some of you may now be asking "What if I set the switch to channel 16 and then use MonoMode2?", well the answer is simple, the channel 'wrap around' so voice 1 will be channel 16, voice 2 will be channel 1, voice 3 will be channel 2 and so on.

Channel select switch '1' is 'ON' and '0' is 'OFF', you should find on your switch that it has the word 'ON' written on it, move the switch towards this to turn it ON.

0000	= Channel 1
0001	= Channel 2
0010	= Channel 3
0011	= Channel 4
0100	= Channel 5
0101	= Channel 6
0110	= Channel 7
0111	= Channel 8
1000	= Channel 9
1001	= Channel 10
1010	= Channel 11
1011	= Channel 12
1100	= Channel 13
1101	= Channel 14
1110	= Channel 15

1111 = Channel 16

The CPU controls the operation of the MUX via the four 'DEMUX' lines and the 'MUX_EN' line. It also controls the transistors which generate the gate output (GATE0-3). It also controls the Serial DAC (AD8300). This is a 12 Bit DAC, and was chosen for the best price/performance compromise, we could have had a 16 Bit one but the cost would have been much greater.

Onto the Analogue section, we have a 4067 analogue mux/demux sending out the DAC's output to the various sample and hold circuits. The scale of the DAC is adjusted via the 'SCALE' preset, it's a 10 turn preset and in conjunction with R21 should provide adequate sensitivity to get 1V/oct without much effort.

Each sample and hold is made up from a 10nF hold capacitor and a TL074, the refresh rate of each sample hold is around 11 kHz, so it's a pretty nippy little refresh cycle, which means there are no noticeable delays or latency when you send MIDI data to the PolyDAC.

The only other point of interest on page 2 is the 'PITCH ZERO' preset, this sets the ZERO point for the Pitch Bend CV (as this CV is bipolar).

The Pitch Bend CV connector has a 3 pin connector, the idea behind this is to allow normalisation of the pitch bend CV back into the NOTE CV. Use a stereo socket for this connection; the Tip for the CV out (pin1) the Ring for CV in (pin2) and pin3 is ground. Connect the two NC tags of the socket together so that when no jack is inserted the CV is fed back into the circuit.

In a similar fashion, the Mod Wheel CV has the same option, (this means one LFO/VCA for modulation on all 4 voices) I would suggest normalising the ring to 'ground' this time though.

Page 3 of the schematic has the NOTE CV Sample and holds and also some output circuitry. These outputs differ from the others as we want to ensure that no loading effects occur and no oscillations with long lengths of coaxial cable attached. It wouldn't be much good as a MIDI to CV converter if we weren't careful of these things!

Notice that I've used 6 pin resistor packs (5 commoned resistors), this allows for the CV summing to all be done on matched resistors (the resistors inside the pack are well matched) it also takes up less board space than 5 separate resistors.

You can see the Pitchbend CV and mod wheel CV inputs coming in on the bottom of the page.

Page 4 shows the remainder of the Sample and hold circuits. It also shows the GATE circuits, the clever part here is that you can 'select' your gate trigger voltage by changing R55, R59, R63 and R67, *why* you ask? Modular's GATE voltages can very, I have seen triggers range from +5V to +10v.

With it set to 10K you get 7.5V for a GATE on pulse (assuming 15v supply rails) change this to 5K1 and you'll get 5V for a GATE on. Similarly, 20K will give you a 10v gate pulse. Bare in mind that if you choose to use a +/-12V power supply that you will need to change these resistors accordingly.

The LEDs are ‘active low’ which means that when GATE0 is at 0V the LED will come on. whilst this might seem strange at first, its common in digital electronics and most digital ICs can ‘sink’ more current than they can ‘source’. So the transistor (Q6, Q7, Q8, Q9) invert this active low to give the more commonly used active high suitable for triggering ADSRs and so on.

This might seem like a bit of a quick run through, but there really isn’t much to tell. I'm sure most of you are familiar with Sample and Hold circuits. The DAC, well, it does just that, convert Digital to Analogue. I chose a serial DAC for space reasons. Also the AD8300 has a very stable built in reference voltage so things shouldn’t start drifting over time.

Sourcing your Components

Most of the parts are 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. 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-16 or Piher PC16 types with matching brackets. You could use any type you want, but not all pots have the same pin spacing. In the UK, Maplin, CPC and Rapid sell the Omeg pots at a very good price. But note that Rapid or CPC don’t sell the pot brackets. RS sell the Piher range. The pot kit that supplied contains the two pots and the two pot brackets.

For the resistors 1% 0.25W Metal film types should be used for all values, we want things to be as accurate as possible!

All the electrolytic capacitors should be 25V or 35V, and radially mounted. Don’t chose too high a working voltage like 63V. The higher the working voltage the larger the size of the capacitor. A 220V capacitor will be too big to fit on the board!

The pitch spacing of the 10nF and 100nF polyester capacitors is 2.5mm (0.1”). Whilst these aren't as common as the 5mm pitch ones the obvious saving in board space makes the extra hunt worth while.

Rapid Electronic part numbers are as follows:

10nF Sub-miniature polyester - 10-1065
100nF Sub-miniature polyester - 10-1095

The small ceramic capacitors are also only 2.5mm pitch. You can use either multilayer types or ceramic plates. The Rapid part numbers: 33pF Radial multilayer ceramic - 08-0930 or 33pF Ceramic Disc - 08-0050.

L1 and L2 are axial mounted ferrite beads. These look a little like black resistors. They are usually in the EMC or Filtering section of your components catalogue. Farnell make a good one, part number: 108-267

All op-amp ICs are dual in line (DIL or DIP) packages. These are generally, but not always, suffixed with a CP or a CN in their part numbers. For example; TL074CN. Do not use SMD, SM or surface mount packages. They do not fit at all and you'll probably loose them behind your bench during building.

The op-amps are standard parts, the AD712, TL074 and LF412 should be available from anywhere.

The two multiturn trimmers are the Bournes 3296Y type. These have 3 pins with centre pin being offset. These are adjusted from the top and, are called 'Multiturn Cermet'. These are different to ones normally used on most Oakley projects which have all their pins in a single line.

The four switches you need are not all the same, although all are SPDT types. I use Apem flat toggle switches. These are attractively priced from Farnell and others, and are of good quality.

Assign:	on-mom	Farnell pt. number: 607-381
Re-trigger:	on-on	Farnell pt. number: 147-772
Poly1/2:	on-on	Farnell pt. number: 147-772
Mode:	on-off-on	Farnell pt. number: 147-774

I would recommend that you use IC socket for the CPU, even if you don't use sockets for any of the other chips. This is a 40 pin DIL (0.6" spacing) type. You can then still insert the pre-programmed chip in case any updates need to be made. If you're confident of your soldering then I would suggest soldering in the other ICs, if not, use sockets.

D3 is a LM329, a 6.9V reference voltage chip. This should come in a plastic TO92 package, ie. it will look like a small NPN transistor. They are available with different accuracies and temperature coefficients. Just get the cheapest one that you supplier sells. It will still have a perfectly good and accurate output for our needs. Please note that this device is not correctly depicted on the PCB layout. The board was laid out for an ordinary zener diode, but the LM329 is the recommended device. Details on mounting this device is given later in this document.

I use 5mm LEDs, with Cliplite LED holders. The LEDs are available in most places, although Rapid sell them cheapest. The cliplites are available from Maplin.

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. All of the sockets must have normalising lugs if you are building the suggested layout. The Switchcraft 112 types have normalising lugs as standard.

Also don’t forget that **two** of the sockets should be stereo to get the best from the pitchbend and mod wheel normalisation.

Finally, if you make a circuit change that makes the circuit better, do tell myself (Paul) or Tony 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.

NOTE: There may be some discrepancy between this document and the schematic. This User Guide takes precedence and should be considered as the most up to date version.

Resistors

1 of 10R
R73

15 of 75R
R20,R27,R29,R34,R35,R36,R37, R38,R39,R40,R41,R42,R43,R75,R76

3 of 100R
R7,R8,R9

3 of 220R
R2,R11,R12

5 of 1K
R10,R57,R61,R65,R69

10 of 3K3
R3,R4,R5,R6,R13,R56,R60,R64,R68,R88

7 of 4K7
R89, R19, R18, R17, R16, R15, R14

19 of 10K
R1,R28,R30,R31,R32,R33,R46,R47,R48,R50,R54,R55,R58,R59,R62,R63,R66,R67, R90

1 of 11K
R22

1 of 12K
R21

1 of 22K
R23

4 of 47K
R24, R25, R26, R91

4 of 10K x 5 commoned resistor network (6 Pin SIL)
RP3, RP4, RP5, RP6

2 of 1K x 8 commoned resistor network (9Pin SIL)
RP1, RP2

Capacitors

22 of 100nF polyester 63V (2.5mm pitch)
C1,C2,C5,C6,C7,C8,C13,C14, C20,C21,C22,C23,C24,C25,C26,
C27,C28,C39,C40,C45,C47,C48

16 of 10nF polyester 63V (2.5mm pitch)
C9,C10,C11,C12,C15,C16,C17, C18,C29,C30,C31,C32,C33,C34,C35,C36

6 of 33pF ceramic plate or multilayer
C19,C41,C43,C44,C46,C49

4 of 22uF, 25V electrolytic (Radial mount)
C3,C4,C37,C38

Semiconductors

1 of 1N4148 silicon signal diode
D1

1 of BAT-42 schottky signal diode
D2

1 of LM329 6.2V reference chip
D3

9 of 2N3904 NPN transistors
Q1, Q2, Q3, Q4, Q5, Q6, Q7, Q8, Q9

Integrated Circuits

4 of TL074CN
U9, U11, U15, U16

5 of LF412CN
U10, U12, U13, U17, U18

1 of AD712JN
U8

1 of 6N137 (Logic level Opto-Isolator)
U1

1 of AT-MEGA16 pre-programmed Microcontroller
U2

1 of AD8300 (12Bit Serial DAC)
U4

1 of 78L05 (100mA 5V regulator)
U5

1 of 4067B (Analogue demux)
U7
1 of 74HC00 (quad 2 input NAND gate)
U6

1 of 16MHz Crystal Oscillator Module (8 pin DIL format, but only four pins are used)
U3

Switches

All SPDT (single pole double throw) switches.

Assign:	on-mom	Farnell pt. number: 607-381
Re-trigger:	on-on	Farnell pt. number: 147-772
Poly1/2:	on-on	Farnell pt. number: 147-772
Mode:	on-off-on	Farnell pt. number: 147-774

Other

47KA linear potentiometer	Two; Tune, Bend Depth
Omeg pot brackets	Two: Tune, Bend Depth
1K multiturn Bourns 3296Y style preset	Two; Scale and Pitch zero
Leaded axial ferrite beads	L1, L2
4-way DIP switch in 8 pin DIL	Midi channel select
5mm Orange LED and clip	five (Gate 1,2,3,4, MIDI)
1/4” sockets, mono	Eighteen
1/4” Sockets, Stereo	Two (Pitch Bend/Modwheel)
Two knobs	
1m of multistrand hook up wire in a variety of colours	
5 pin 180 degree sockets	TWO (MIDI In and Thru)

Building the Oakley/Modulus polyDAC

Occasionally people have not been able to get their Oakley projects to work first time. Some times the boards will end up back with Tony so that he 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 four of these was parts inserted into the wrong holes. Please double check every part before you solder any part into place, especially the ICs as they do not all have the same orientation. 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 use 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 at least once every two or so 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. 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.

Tony has 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' or some types of 'no-clean' 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. Tony uses an ordinary Antex 240V 25W iron with a Variac power supply running at 200V. This seems to work well for Tony.

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.

For D3 you need to take special care. The board was laid out incorrectly and a normal diode shape has been used. As you will see the LM329 looks more like a transistor than a diode. It has three legs and is in a TO92 style housing. But don't worry it will fit in the board OK if you follow these instructions carefully.

Take the device and hold it with the legs upwards, and the flat of the body away from you. With a pair of wire cutters, snip off the left leg. Make your cut about 2mm away from the body of the diode. Now you should have a two legged device. (Its cruel, I know, but its better this way). The middle pin of the device is the cathode, and this should be **gently** positioned into the bottom pad of D3. The right hand leg, the anode, should go in to the top pin of D3. Make sure that neither leg touches the stump of the snipped leg.

IC sockets are to be recommended, 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.

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

Make sure you get the electrolytic capacitors 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 ICs have different orientations on this PCB, this was in order to save space, so you *MUST* be careful when inserting the ICs. Pin 1 is also the one with the square solder pad.

The SIL packages for the Resistor packs, should have a dot at one end of the package. This represents pin one of the pack, and is shown with a square pin on the PCB and also an extra line on the legending.

I would make the board in the following order: resistors, multilayer ceramics, diodes, IC sockets, polyester and ceramic plate capacitors, transistors, electrolytic capacitors. Then the final water wash. You can now fit the two trimmers and potentiometers. The mounting of the pots requires special attention. See the next section for more details.

.

Mounting the Pots

If you are using the recommended Omeg ECO or Piher/Meggitt PC16 pots, then they can support the PCB with specially manufactured pot brackets. You will not normally need any further support for the board. When constructing the board, fit the pot brackets to the pots by the nuts and washers supplied with the pots. Now fit them into the appropriate holes in the PCB. But only solder the three pins that connect to the pot. I normally solder the middle pin first and then check if the pot is lying true. 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 47KA is a 47K lin 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 Omeg 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. If I do see any grease near the top of the bush, I tend to wipe it clean with a bit of kitchen paper before I mount the pot.

Connections

There are a lot of connections in this module, but it is easy to wire up if you take your time. If you have used Switchcraft 112 sockets you will see that the single pole ones (mono types) have three connections. One is the earth tag. One is the signal tag which will be connected to the tip of the jack plug when it is inserted. The third tag is the normalised tag, or NC (normally closed) tag. The NC tag is internally connected to the signal tag when a jack is not inserted. This connection is automatically broken when you insert a jack. The tags are actually labelled in the plastic next to the tag. The signal lug is called 'T' for tip, the NC lug is labelled 'T/S' for tip-switched.

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

For the Stereo sockets there is also a ‘RING’ lug and its corresponding ‘NC’ lug as well.
Fit all the sockets onto this module so that the bevel on the side of the socket is facing top left as you look at the rear of the panel. There are twenty sockets in total, so take your time.

Note, that aside from the two stereo sockets, we will not be using the NC lugs.

The first lugs we are connecting together will be the ground or earth tags on the two bottom rows of sockets. We will ignore the upper set of 12 sockets for now. I use 0.91mm diameter tinned copper wire for this job. Its nice and stiff, so retains its shape. Solder a length of this solid core wire across all the earth tags on the bottom row. Trim off any excess that sticks out on either end. Then do the same on the lower row of sockets. What you have now done is common each row’s earth tags together, but each row is still separate for now. Now repeat this for the upper set of 12 sockets.

Finally run another piece of wire between all of these and join them all together.

Fit the switches, I find this easier to do on the bench, but don’t forget to leave enough lead spare to reach the point on the PCB.

Fit the two 5 Pin sockets for the MIDI, again I find it easier to solder the wires on before mounting them.

Fit the PCB against the front panel if you haven't done so already.

There are not enough pads for the ground connections to all the sockets directly, but there are two available, for the pitchbend and Mod Wheel CVs, so if you have all the grounds of the pots connected then once you connect the Pitchbend and Mod Wheel sockets to their PCB pads all should be well.

Switch Name	***PCB pad name***
Assign (momentary)	J1, Pin 1 and 2(2 is Ground)
Mode (2 pole, centre off)	J2, Pin 2 goes to the centre lug on the Switch
PolyMode	J3, Pin 1 and 2(2 is Ground)
ReTrigger	J4, Pin 1 and 2(2 is Ground)

Socket Name	***PCB pad name***
MIDI IN	J7 , Left most pin goes to Pin 4 on the 5 pin socket, the rightmost goes to pin 5 of the MIDI socket
MIDI Thru	J8, Left most pin goes to pin 5 of the Midi, socket, pin2 (centre) goes to pin 2 of the Midi socket and the rightmost pin connect to pin4 of the MIDI socket
Pitch Bend	J9, Leftmost pin goes to ‘TIP’, middle pin goes to ‘RING’ and rightmost to

	ground lug. Connect the two NC lugs together with a bit of wire.
Mod Wheel	J10. Uppermost pin to ‘TIP’, middle pin to ‘RING’ and lowest pin to Ground lug.
Sustain	J11
Aftertouch	J12
Voice 1 NOTE CV	J13
Voice 2 NOTE CV	J14
Voice 3 NOTE CV	J15
Voice 4 NOTE CV	J16
CV-A	J17
CV-B	J18
CV-C	J19
CV-D	J20
Voice 1 VELO CV	J21
Voice 2 VELO CV	J22
Voice 3 VELO CV	J23
Voice 4 VELO CV	J24

Socket Name	***PCB pad name***
Voice 1 GATE	J29
Voice 2 GATE	J30
Voice 3 GATE	J31
Voice 4 GATE	J32

LED Name	***PCB pad name***
MIDI	J6, Leftmost pin to the Cathode Rightmost pin to Anode
Voice 1 GATE	J25, Uppermost pin to Anode Lower to the Cathode
Voice 2 GATE	J26, Uppermost pin to Anode Lower to the Cathode
Voice 3 GATE	J27, Uppermost pin to Anode Lower to the Cathode
Voice 4 GATE	J28, Uppermost pin to Anode Lower to the Cathode

The LEDs are simple to wire up, but they need to connected the correct way around, otherwise they don’t work and they may be damaged.

I always twist the two wires that connect LEDs together. This has two purposes; one, it keeps the connections tidy, and two, it minimises any LED switching noise from reaching the audio lines. In any case, keep your LED leads away from the audio leads.

Power connections

The power socket should be a 0.156" Molex/MTA 4-way header. However, due to a slight cock-up the holes in the PCB were made to small. So the only way of putting power into this unit is to wire it directly with 22 or 24AWG wire. The pin out is compatible with MOTM.

Power	***Pin number***
+15V	1 (square pad)
Module GND	2
Module GND	3
-15V	4

If you are wiring it up to a standard Oakley power bus you will not want to connect pin 3 on the polyDAC to pin 3 on the Oakley power bus. Pin 3 on an Oakley power bus is reserved for chassis ground or earth, and not module ground or 0V. In this case, Tony recommends that you connect pins 1, 2 and 4 only. The pin 3 connection is left unconnected.

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

Apply power to the unit making sure you are applying the power correctly. Check that no device is running hot. Any sign of smoke or strange smells turn off the power immediately and recheck the polarity of the power supply, and the direction of the ICs in their sockets.

The next thing to do is to make sure that we can receive MIDI data, I would suggest starting with the unit set to MIDI Channel 1 (all 4 switches on SW1 off).

Send it some notes and you should see the MIDI light flash, if you don't then the most common fault is that pins 4 and 5 have been connected the wrong way round, just swap them over and try again.

If you have the mode switch set to 'Poly' then as you press a note you should see the LEDs cycle, 1 then 2, then 3, then 4.

Next try Monomode 1, if you press a key you should see all 4 gate lights come on at once. Try monomode 2 and you should see only GATE light 1 flashing as you press notes.

Trimmers and scaling!

This is the part that most people fear with Midi to CV convertors, but as we have only two trimmers and two pots, its really simple, quick and easy to set-up.

Turn OFF the PolyDAC and then back on. Why? Just in case you have any sent any data that could upset the calibration.

Move your meter probe to Voice 1 NOTE CV. Adjust the tune knob so that it reads 0V.

Now, you’ll notice an unused jumper J5 (near the top of the PCB), put a wire temporary link in this and you’ll see that ALL the voltages go to their maximum (excluding pitchbend).

Then simply adjust the ‘SCALE’ trimmer until the Voltage reads 10.667V.

Set the BEND DEPTH pot to maximum and attach a meter to the pitch bend ‘TIP’ (you can do this with a plug or by probing the pin directly). Adjust the ‘PITCH ZERO’ preset until you have a reading of zero volts.

NOTE: If you have an existing modular set-up with four perfectly tuned VCOs, then you may want to try a different way of adjusting the scaling. In mono mode, tune all four VCOs to the same pitch with their own fine tune controls. Then in poly mode, play two notes an octave apart. Simply adjust the ‘SCALE’ trimmer to get minimum beating between the two VCOs.

That’s it, you’re done!

Final Comments

I hope you enjoy building and using the Modulus PolyDAC 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 myself (Paul) or Tony, and we will see what we can do. Sometimes, it can be the simplest things that can lay out a project. Tony does offer a ‘get you working’ service. Send your completed non-working module back to him with £25 and he will fix it for you. You will also have to pay for the postage both ways and any parts he has to replace. Make sure you wrap it carefully and include a full description of the fault. If you are sending the item from outside the EU, then be sure to say on the customs label ‘item being sent for repair only’.

If you feel the fault is related to a bug in the OS, then please e-mail me at Paul.Maddox @ Synth.net (No spaces next to the '@') . Please give a full description of the bug, and ideally a MIDI file that demonstrates it, I will then look into it and if needed send a new version of the OS to you.

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. If you do notice any error, please get in touch with either of us.

Please further any comments and questions back to myself and/or Tony; your suggestions really do count. If you have any suggestions for new projects, feel free to contact myself or Tony. You can e-mail me or you can e-mail, write or telephone Tony. If you telephone Tony 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 and persuaded me to get this puppy moving. Thanks especially to Tony Allgood for his help and patience with my e-mails and Steve Thomas for my constant changing around of the PolyDAC panel layout.

Paul Maddox April 2004

Edited by Tony Allgood August 2004