Magwitch Modular
Slewlim

Eurorack Module

Build Guide

Specification

Width	6 HP
Depth	25mm
Power	+12V 30mA, -12V 25mA, +5V 0 mA
Channels	1
Inputs per channel	2 CV, 1 Signal
Outputs per channel	1
Input Signal	-10V to +10V
CV Signal	0V to 5V
Output Error	< 0.5% into 100K impedance

Components

Suggested Suppliers

- uk.farnell.com
- www.mouser.com
- www.thonk.co.uk – for pots and jack sockets
- www.toby.co.uk – for shrouded power sockets (J1)
- www.accu.co.uk – for M2 screws and nuts

Component Notes

Over the page is the complete bill of materials, listing all components, values, packages and recommended parts.

Other variants of the given op-amps and voltage regulators will likely work. Googling the supplied part numbers should produce results with more information and links to suppliers.

It is important to use OPA4196 for U1, as it has the required rail-to-rail input and output characteristics to operate as a slew limiter. Others that can work are OPA4191, OPA4192, OPA4197 – but these are typically more expensive, and some are more power hungry.

Before assembly, it is recommended to adjust each of the trimmers so that the resistance between the outer pins and the middle pin is the same for each pair – i.e. in the middle of the range. This can make calibration quicker, or may suffice as basic calibration.

Not present on board: D18, D19, D20.

Bill of Materials

Parts	Description	Count	Package	Recommended
C1 C2	10uF	2	4mm can	EEEFP1E100AR, 25V+
C10	33nF ceramic X7R	1	0805	16V or higher
C11	2u2 ceramic X7R	1	0805	16V or higher
C3 C4 C5 C6 C7 C8 C9	100nF ceramic X7R	7	0805	16V or higher
D1 D2	Schottky diode	2	SMD	STPS1L30U
D16 D17	6V8 Zener diode	2	SOD-123	BZT52C6V8
D21	3mm R/G bipolar LED	1	Thru-hole	SSL_LX30FT14IGW
D3	5V reference	1	SOT-23	LM4040-5V
D4 D5	2.5V reference	2	SOT-23	LM4040-2V5
D6 D7 D10 D11 D12 D13 D14 D15	Signal diode	8	SOD-123	1N4148
D8 D9	Schottky diode	2	SOD-123	BAT48
J1	Power pin header	1	Shrouded	302-S-10-D1R1
J2 J3 J4 J5	Thonkiconn jack	4		PJ398SM or PJ301M-12
JP1	2 pin header	1	2.54mm	M20-9994046
L1 L2	Ferrite bead	2	0805	BLM21PG221SN1D
Q1	PNP transistor pair	1	SOT-363	DMMT3906
Q2	NPN transistor pair	1	SOT-363	DMMT3904
R5 R12 R20 R21 R24 R22 R38 R41 R42 R47 R48 R49 R50 R51 R53	100K 1% resistor	15	0805	ERJ-6ENF
R4 R6 R9	10K 1% resistor	3	0805	ERJ-6ENF
R23 R25 R26 R28	130K 1% resistor	4	0805	ERJ-6ENF
R54 R55	17K4 1% resistor	2	0805	ERJ-6ENF
R1 R7 R8 R17 R36	1K 1% resistor	5	0805	ERJ-6ENF
R10 R11	1M 1% resistor	2	0805	ERJ-6ENF
R15	220 ohm 1% resistor	1	0805	ERJ-6ENF
R43 R44 R45 R46	27K 1% resistor	4	0805	ERJ-6ENF
R19 R27	2K2 1% resistor	2	0805	ERJ-6ENF
R16 R18 R52	470 ohm 1% resistor	3	0805	ERJ-6ENF
R31 R34 R35 R37	47K 1% resistor	4	0805	ERJ-6ENF
R2 R3 R32 R33 R39 R40	4K7 1% resistor	6	0805	ERJ-6ENF
R29 R30	51K 1% resistor	2	0805	ERJ-6ENF
R13 R14	680 ohm 1% resistor	2	0805	ERJ-6ENF
R56 R57	82K5 1% resistor	2	0805	ERJ-6ENF

Parts	Description	Count	Package	Recommended
RV1 RV2	100K linear trimmer	2	Multiturn trimmer	3296W or PV36W
RV3 RV4 RV5 RV6 RV7 RV8	100K linear potentiometer	6	Thru-hole, vertical	Song Huei tall pot or Alpha 9mm T18 shaft
U1	OPA4196	1	SOIC-14	OPA4196IDR
U2	TL074	1	SOIC-14	TL074IDR
U3	TL072	1	SOIC-14	TL072IDR
	M2 15mm cheese head screw	1		
	M2 nut	3		
	2 pin 2.54mm jumper	1		

Assembly

This module requires both surface mount and through-hole soldering. You can use various methods for the surface mount soldering, such as hand soldering, heat gun or hotplate method. If you haven't attempted these before, it is highly recommended to study them and practice first.

- Hotplate method by Hobbytronics – this shows the basics
- Surface Mount Soldering – this shows the heat gun method, both with and without a stencil.

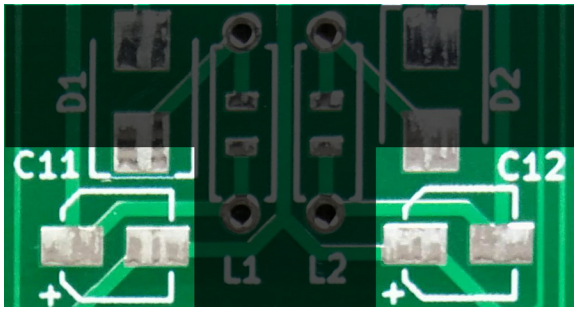
The steps to follow for this module using the heat gun method are as below, along with some hand soldering for both surface mount and through-hole. If you want to entirel hand solder, the order is similar. It is recommended to build up the board step by step, checking the joints and connectivity after each step.

Some hints for the hot plate method are provided at the end.

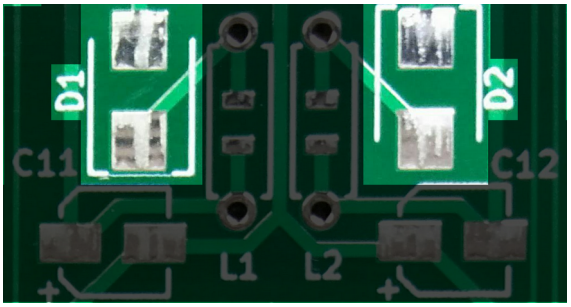
Orienting Components

Pay attention to the silkscreen printing around component footprints; this gives an indication of how certain polarised components (capacitors, diodes) and ICs should be placed.

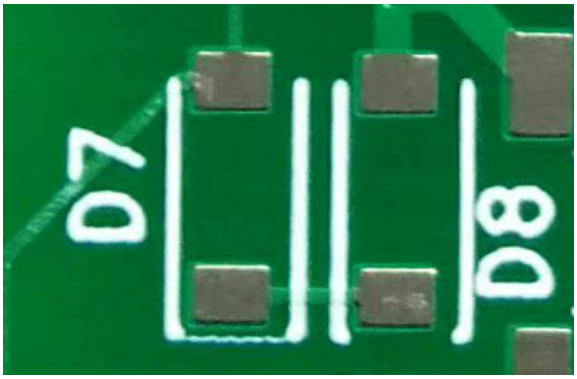
Orient electrolytic cap bases with the outline on the PCB, as highlighted here:



Orient all diodes correctly; the cathode is the closed end of any diode outline on the PCB, and should line up with the cathode marking on the diode itself, usually a line). The closing line may not be fully present on the silk screen – in that case, place the diode with the cathode on the pad where the silk screen lines outside it extend furthest.

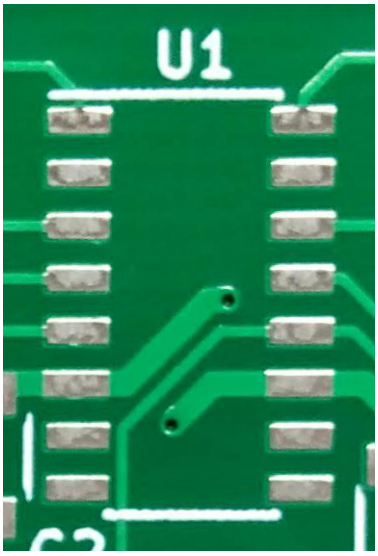


This applies to all types and sizes of diodes – the large rectifier diodes on the power inputs (D1, D2) and the various signal diodes (e.g. D6 and above).



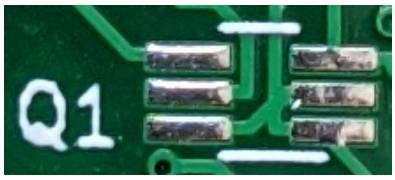
Orient all ICs properly – pin one should be on the pad with the adjacent extended line on the silkscreen.

The ICs in this circuit will have pin one marked by one or more of the following:



- A short line across the IC body indicating its top, where pin 1 is located.
- A circle, dot or dimple at the corner where pin 1 is located.
- A chamfer on the body (sloped edge) which marks the side on which pin 1 is located.

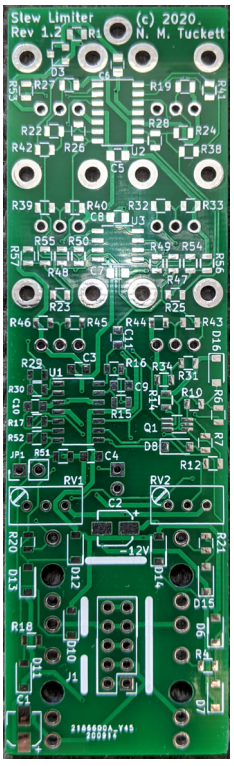
This design has two SMD transistor pairs, Q1 and Q2. On the board, these have the following footprints:

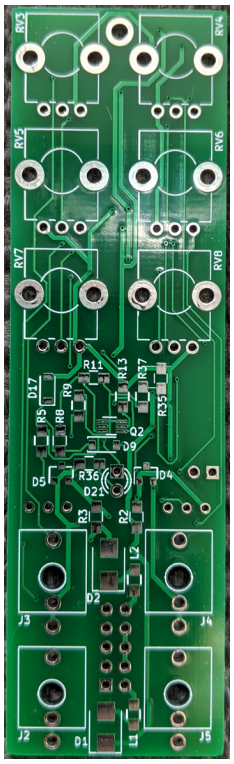


Pin 1 on each pair is marked by a small pin-prick – this should go on the pad with the adjacent extended line. Also be sure to place the right type of transistor pair: Q1 is PNP (DMMT3906, marked K4B on chip) and Q2 is NPN (DMMT3904, marked K4A on chip). **The markings are very hard to read without good lighting and magnification** – so please check them carefully!

Orienting the PCB

The PCB is taller than is wide, and should be oriented so. The Eurorack 2x5 pin power connector and jack sockets (J2 to J5) should be at the bottom of the board.





Heat gun/Hand Soldering Step-by-Step Assembly

The overall direction of assembly is as follows:

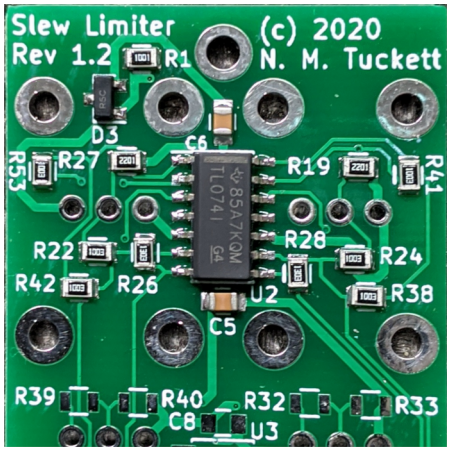
1. Surface mount components on PCB rear, top to bottom.
2. Surface mount components on PCB front.
3. Eurorack power header, trimmers and 2 pin header.
4. Jacks, LED and potentiometers.

Each stage of assembly is detailed below, with images showing the component placement and a list of the relevant parts in a suggested placement order.

If you want to clean any flux off the PCB, the recommended points to do this is to clean both sides after step 2, then clean the power header, trimmers and pin header joints after step 3, and finally clean the jack, LED and pot joints after step 4.

PCB Rear Surface Mount Components

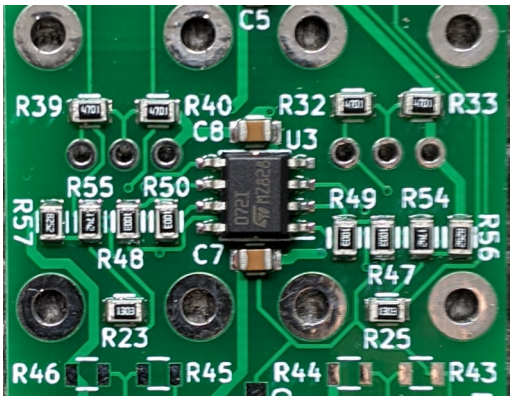
Start at the top of the PCB with the first section of components around (and including) U2:



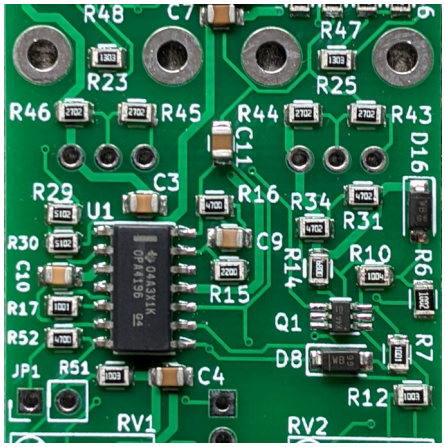
- IC: U2
- Capacitors: C5, C6
- Diodes: D3
- Resistors:
  - R1, R27, R19, R53, R41
  - R26, R28, R22, R24
  - R42, R38

Move down to the next section of components around U3:

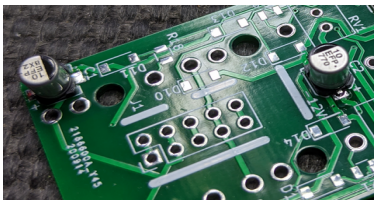
- IC: U3
- Capacitors: C7, C8
- Resistors:
  - R40, R32, R39, R33
  - R50, R49, R48, R47
  - R55, R54, R57, R56
  - R23, R25



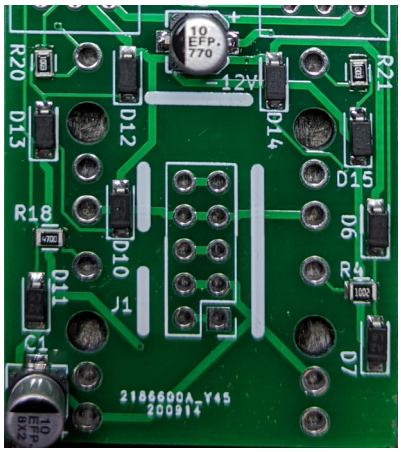
And next, add all the parts around U1:



- IC: U1
- Capacitors: C3, C4, C9, C10, C11
- Resistors:
  - R46, R45, R44, R43
  - R29, R30, R17, R52, R51, R15, R16
  - R31, R34, R14, R10, R6, R7, R12
- Diodes: D16, D8
- IC: Q1



Next, solder on the two electrolytic capacitors C1 and C2. It can be a bit easier to do these by soldering iron than by hot air, but you can do either.

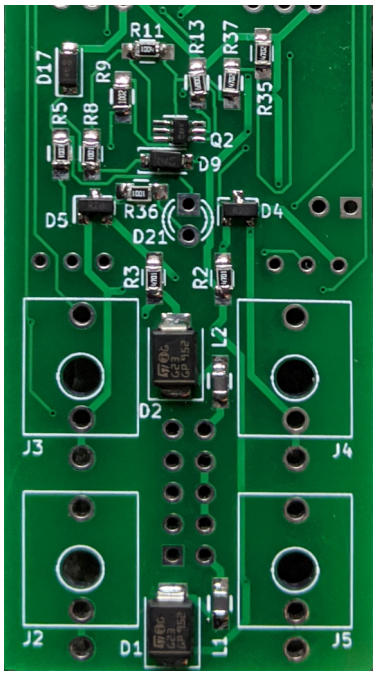


Finally, fit the remaining surface mount components for the board rear:

- Resistors: R20, R21, R18, R4
- Diodes: D12, D14, D13, D15, D10, D6, D11, D7

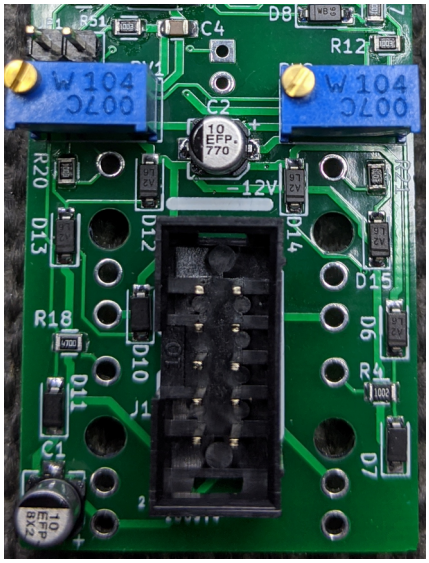
PCB Front Surface Mount Components

The next stage is to fit the surface mount components onto the front of the board. It is recommended to use a soldering iron for these as it minimises the amount of heat that will get to the rear of the board and potentially unsolder components placed there. To be safe, put some masking or Kapton tape over the components on the lower half of the PCB rear to ensure nothing drops off.



- IC: Q2
- Resistors: R11, R13, R37, R35, R9, R5, R8, R36
- Diodes: D17, D9, D5, D4
- Resistors: R2, R3
- Diodes: D1, D2
- Ferrite beads: L1, L2

Power Header, Trimmers and Jumper



Mount the shrouded power header J1, trimmers RV1 and RV2 and the two pin jumper JP1 on the rear of the board, and secure them using some sort of temporary adhesive (e.g. kapton tape, masking tape or blu-tak).

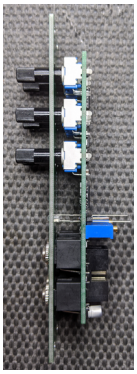
Solder the pins for these on the front side of the board. It is recommended to solder one pin first, then re-flow that pin while pushing the component up onto the board to ensure it sits correctly. If it does not, re-heat the joint and adjust the connector position again. Once the position is right, solder the remaining pins.

For the trimmers, cut off the excess leads on each one after soldering.

Jacks, LED and Potentiometers

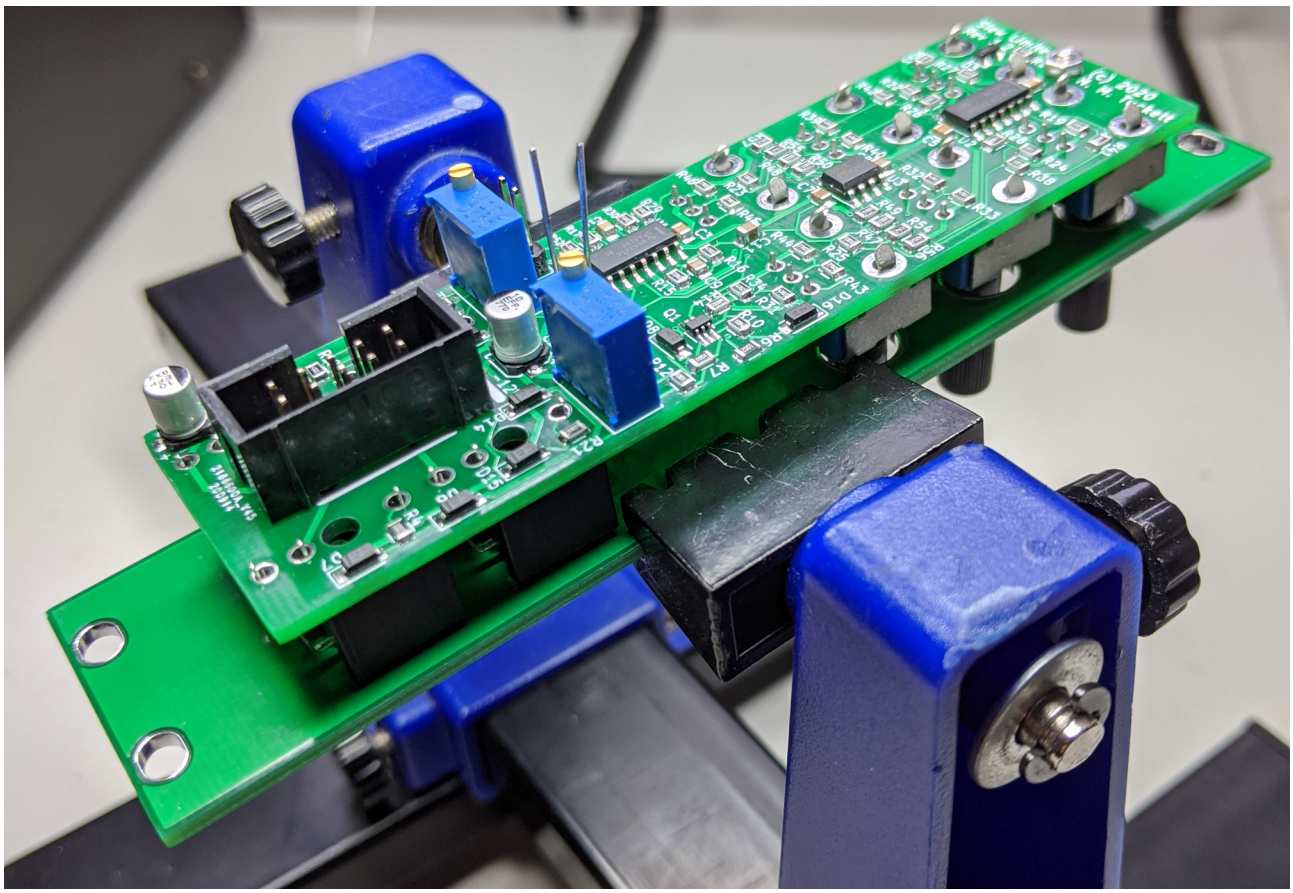
Place the jacks (J2, J3, J4, J5), potentiometers (RV3, RV4, RV5, RV6, RV7, RV8) and LED (D21) into their holes on the front of the PCB. The long leg of the LED should go in the lower hole of the PCB – if you get this wrong, the LED will still work but the colours will be switched.

Fit the M2 screw to the screw hole on the panel, and secure it using one of the M2 nuts. Screw on a second nut about 5mm onto the thread. Then mount the panel onto the PCB, fitting over the jacks and potentiometers – and ensure the M2 screw goes through the corresponding hole at the top of the PCB.



Adjust the lower nut on the screw so it is flush against the PCB and that the panel is level from top to bottom, and secure it with the third nut on the other side of the PCB – but don't tighten it fully yet. Fit the nuts onto the jacks so they don't fall off, and again don't tighten them up.

Adjust how the panel lines up with the PCB to ensure all the pots are pretty much centred in their holes and turn freely – having not tightened the nuts, there should be a little room to maneuver. Make sure all the pots and jacks are flush against the PCB, then tighten up all the nuts and check alignment and pot turning again to be sure.



Clamp the panel into your PCB holder with the PCB on top, and solder the pins on each jack, pot and LED. A good idea is to solder the same pin on each one, so you don’t accidentally overheat any of the components. If you want to, you can also solder the larger lugs on the pots for extra mechanical grip, or just bend them outwards or inwards to achieve the same. Finally trim the legs of the LED.

Congratulations, you have now assembled your Slewlim!

Hot-plate Soldering

The suggested way to approach hot-plate soldering is as follows:

1. Apply paste to all pads on rear of PCB.
2. Place all components on PCB rear, and use hot-plate to solder.
3. Hand-solder the non power SMD components on the front of the PCB
4. Hand-solder the power SMD components on the PCB front (D1, D2, L1, L2).
5. Proceed with the remaining hand soldering as above: power connector, trimmers, jacks & LEDs, pots.

Calibration

There are only two calibrations for the Slewlim, using the two trimmers. These set the minimum slew rate for rising and falling parts of the input independently.

A simple way to calibrate is as follows:

1. Connect the input of Slewlim to an adjustable constant voltage, e.g. an unpatched attenuverter or the output of a MIDI-to-CV module with a MIDI keyboard attached.
2. Connect the output of Slewlim to an oscillator’s pitch input.
3. Connect the oscillator output so you can hear it.
4. Turn the rise pot on Slewlim to minimum (anti-clockwise), and the fall pot to maximum (clockwise). Ensure the shape pots are both vertical.
5. Set your voltage source to a low voltage (e.g. 0V) and hear the pitch.
6. Now set it to a high voltage (e.g. 5V) and listen to the pitch rise – you could even time it.
7. If you’re not happy with the rate, adjust RV2 and go back to step 5.
8. Turn the fall pot on Slewlim to minimum (anti-clockwise), and the rise pot to maximum (clockwise). Ensure the shape pots are both vertical.
9. Set your voltage source to a high voltage (e.g. 5V) and hear the pitch.
10. Now set it to a low voltage (e.g. 0V) and listen to the pitch fall – again, you could even time it.
11. If you’re not happy with the rate, adjust RV2 and go back to step 9.

There’s no one right calibration - this is very much a taste thing; Slewlim can cover quite a range of slew rates, so calibrate to whatever suits you! You’ll probably want a low end rate that gives you a good few seconds of rise/fall time across the full range.

If you want more accurate calibration, try the following:

- Ensure you use the same low and high voltages for the input all the way through.
- Measure the rise and fall time across that consistent voltage range.
- Use a multimeter or oscilloscope to measure the output voltage rather than using pitch by ear.

Tweaks

If you want good accuracy for CV pitch, fit a jumper onto the 2-pin header JP1. This adds a little gain to compensate for the output protection resistor R18, ensuring minimal error when connecting the output to a 100K impedance 1V/octave input. If you want to tweak that accuracy further, ensure R18 and R52 match closely – you could even use 0.1% resistors for those two.

If you want to customise the range of slew rate further, you could try the following:

- Try different values for C9. This will affect the whole range of slew rate.
- Try different values for R15. This will affect the high limit of slew rate; the lower the value the higher the maximum slew rate.
- Try different values for the voltage dividers R54/R56 and R55/R57 to affect the amount of shape adjustment. Be careful though; this can become uncontrollable.
- Use an OPA4192 or OPA4197 for U1, which have a higher inherent slew rate and can give you a higher maximum slew rate as a result. However they will increase the power draw of the module.