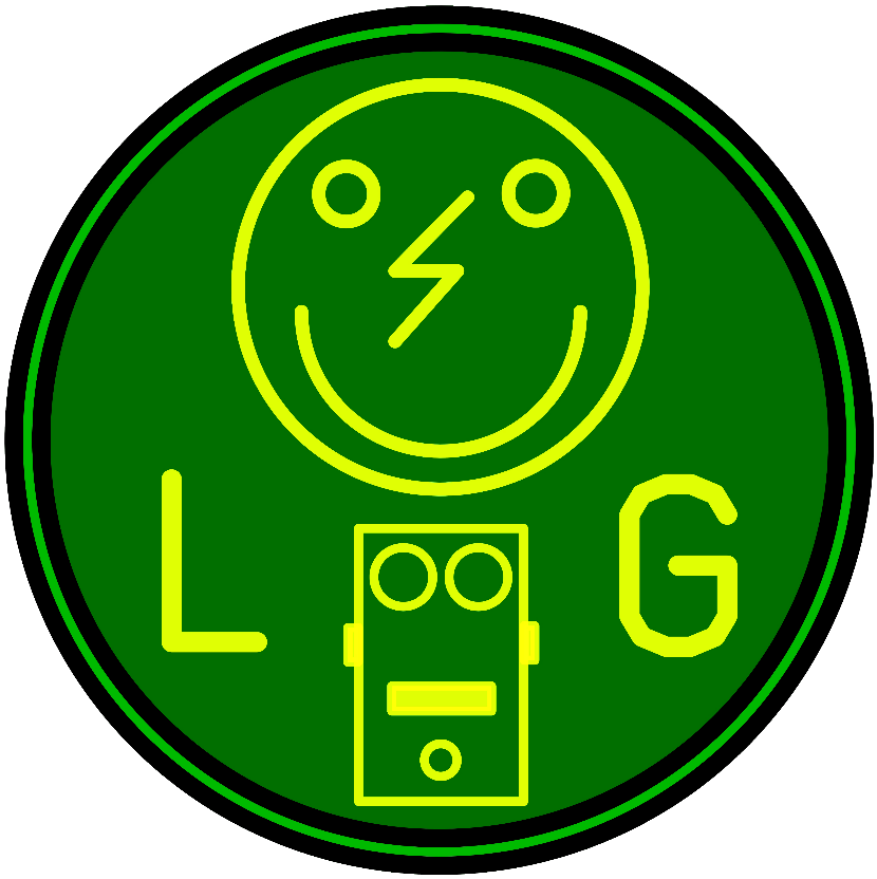
Klonulator

Building instructions

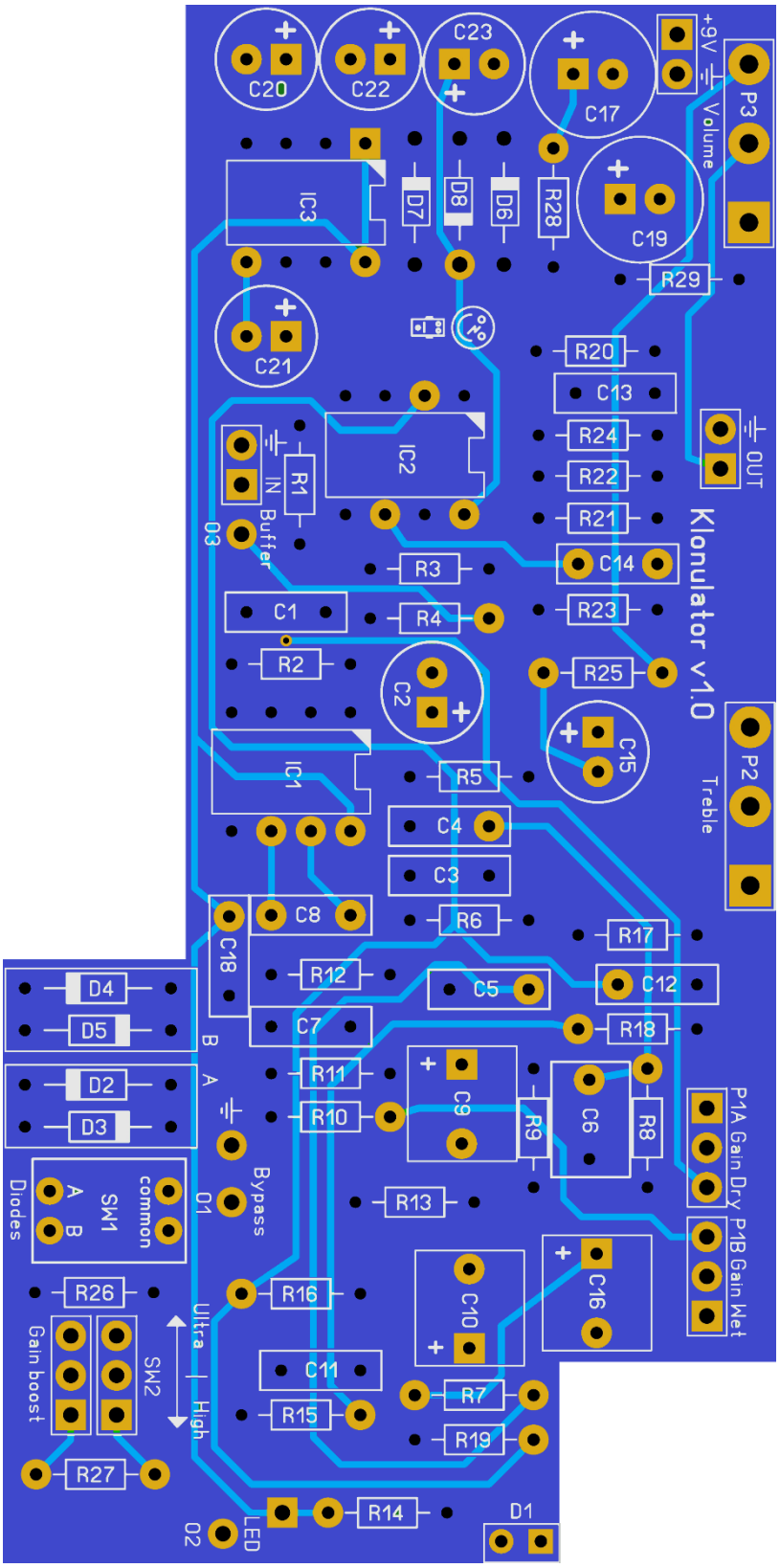
v1.0



Last update: 22-10-2019

Table of contents

PCB layout



Dimensions: **100 mm x 49,6 mm**
3.94 inch x 1.95 inch

Components

Name	Value	Comment	Name	Value	Comment
C1	100n	SMF/MKT/Wima	**P1A***	B100k	Gain Dry
C2	4u7	Electrolytic 25V+	**P1B***	B100k	Gain Wet
C3	100n	SMF/MKT/Wima	**P2**	B10k	Tone
C4	68n	SMF/MKT/Wima	**P3**	B10k	Output
C5	68n	SMF/MKT/Wima	**R1 ****	10k	1% metalfilm
C6	390n	SMF/MKT/Wima	**R2**	1M	1% metalfilm
C7	82n	SMF/MKT/Wima	**R3**	100k	1% metalfilm
C8	390p	SMF/MKT/Wima	**R4**	560R	1% metalfilm
C9	1u	SMF/MKT/Wima/Electrolytic	**R5**	5k1	1% metalfilm
C10	1u	SMF/MKT/Wima/Electrolytic	**R6**	10k	1% metalfilm
C11	2n2	SMF/MKT/Wima	**R7**	1k5	1% metalfilm
C12	27n	SMF/MKT/Wima	**R8**	1k5	1% metalfilm
C13	820p	SMF/MKT/Wima	**R9**	1k	1% metalfilm
C14	3n9	SMF/MKT/Wima	**R10**	2k	1% metalfilm
C15	4u7	Electrolytic 25V+	**R11**	15k	1% metalfilm
C16	1u	SMF/MKT/Wima/Electrolytic	**R12**	422k	1% metalfilm
C17	47u	Electrolytic 25V+	**R13**	1k	1% metalfilm
C18	100n	SMF/MKT/Wima	**R14**	3k9	1% metalfilm
C19	47u	Electrolytic 25V+	**R15**	22k	1% metalfilm
C20	1u	Electrolytic 25V+	**R16**	47k	1% metalfilm
C21	1u	Electrolytic 25V+	**R17**	27k	1% metalfilm
C22	1u	Electrolytic 25V+	**R18**	12k	1% metalfilm
C23	1u	Electrolytic 25V+	**R19**	15k	1% metalfilm
D1	LED	Status (Red diffuse)	**R20**	392k	1% metalfilm
D2	1N34A/D9E	Germanium Diode	**R21**	1k8	1% metalfilm
D3	1N34A/D9E	Germanium Diode	**R22**	100k	1% metalfilm
D4	1N4148	Si Diode	**R23**	4k7	1% metalfilm
D5	1N4148	Si Diode	**R24**	100k	1% metalfilm
D6	1N4742	Zener 12V 0,5W	**R25**	560R	1% metalfilm
D7	1N5817		**R26**	10k	1% metalfilm
D8	1N5817		**R27**	20k	1% metalfilm
IC1	TL072		**R28**	27k	1% metalfilm
IC2	TL072		**R29**	27k	1% metalfilm
IC3	ICL7660S		**SW1**	DIP2	Diodes
			SW2	DPDT	Gain boost

* P1A and P1B form a stereo potentiometer and not two separate potentiometers.

** Pre 1995 original versions have a jumper instead of a resistor as **R1**. It is just a current limiting resistor. There is no sonic difference if you put it in or leave it out.

All parts need to be 25V+ rated **A=Log, B=Lin, C=Rev. Log**

Bill of Materials

Name	Value	Type	Amount	Name	Value	Type	Amount
C	390p	SMF/MKT/Wima	1	**R**	560R	1% metalfilm	2
	820p	SMF/MKT/Wima	1		1k	1% metalfilm	2
	2n2	SMF/MKT/Wima	1		1k5	1% metalfilm	2
	3n9	SMF/MKT/Wima	1		1k8	1% metalfilm	1
	27n	SMF/MKT/Wima	1		2k	1% metalfilm	1
	68n	SMF/MKT/Wima	2		3k9	1% metalfilm	1
	82n	SMF/MKT/Wima	1		4k7	1% metalfilm	1
	100n	SMF/MKT/Wima	3		5k1	1% metalfilm	1
	390n	SMF/MKT/Wima	1		10k	1% metalfilm	3
	1u	SMF/MKT/Wima/Electrolytic	3		12k	1% metalfilm	1
	1u	Electrolytic 25V+	4		15k	1% metalfilm	2
	4u7	Electrolytic 25V+	2		20k	1% metalfilm	1
	47u	Electrolytic 25V+	2		22k	1% metalfilm	1
D	1N34A	Germanium Diode	2		27k	1% metalfilm	3
	1N4742	Zener 12V 0,5W	1		47k	1% metalfilm	1
	1N5817	Scotky Diode	2		100k	1% metalfilm	3
	LED	Red difuse	1		392k	1% metalfilm	1
	1N4148	Si Diode	2		422k	1% metalfilm	1
IC	ICL7660S	Charge Pump	1		1M	1% metalfilm	1
	TL072	Op Amp	2	**SW**	DIP2	Diodes	1
P	B10k	16mm Alpha potentiometer	2		DPDT	Gain boost	1
	B100k	Stereo Alpha 16mm Pot	1				

Besides the components mentioned in the table, you will need:

- **2 mono input jacks**.
- **2,1mm DC jack** (isolated).
- **22 gage stranded hook-up wire.**
- **1 x LED holders (Optional)**. This enables you to mount the LED in the enclosure.
- **Footswitch 3PDT** (9 pins)
- **LED (3mm or 5mm depending on your taste).** These are the status LED
- **Hammond 1590BB** case (or similar) in your favorite color. This case will fit very tight and leaves little room for error. If you need more room you could consider using a **Hammond 1590XX**.

Read this entire manual thoroughly before you start building the effect! There are some available options and you should choose which one you want to incorporate before starting your build.

Introduction

The Klonulator is based on Bill Finnegans famous Klon Centaur™. While sticking close to the original, I added a few optional onboard modifications and dedicated an entire modifications section in this manual so you can add your own flavor to the effect (only if you want to of course). Read the entire manual before you start building. All mods require different components from the ones stated on the previous page, so there are many options that you should for prepare properly!

The PCB has been designed to be able to let you use a wide range of stereo gain potentiometers while still leaving room for a DPDT switch (eg. for the gain boost). Although the Output and Treble pots are designed to be used with PCB mounted pots, you can always choose not to use them and use different pots with standard solder lugs. This can be the case if you want to build it in a bigger enclosure (default is Hammond 1590BB). For this there are also four 3mm holes in the PCB for standoffs.

To close things off, note that I do not claim any originality or genius. I do however claim I did my homework. I've read and tested all I could find on the Internet to be able to bring you the maximum available options. Enjoy your build! If you want to read up on the Klon technology, then take a look at these sites:

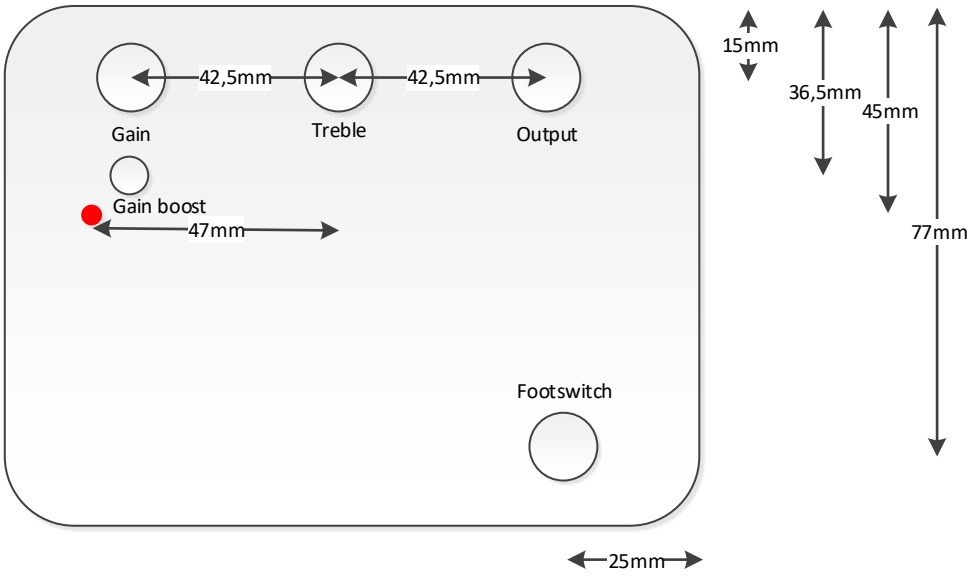
- Freestompboxes.org – Schematic (thank you for tracing Martin Chittum!)
- Electrosmash – Klon circuit analysis
- Freestompboxes.org – Bare Bone Klon
- La Revolution Deux – Klon mods
- DIYStompboxes – Klon Output mixer

Build sequence

Enclosure (drilling)

I suggest you now drill the holes in your enclosure so you can use it during the off board wiring.

Here is a template for drilling the pots, LED and switches in a Hammond 1590BB:



Pots need a **7 mm** hole, footswitches **13 mm,** other switches (Gain boost and/or Diodes) need **6 mm** holes and the LED is either **5 mm** or **3 mm** depending on your choice of LED.

Diameters might differ with different brands of switches, so before drilling the holes, measure if it fits!!!!! The 36,5 mm for the gain boost hole is just a suggestion based on the switch I used.

The hole for the LED is based on the position when it is PCB mounted. You are of course free to put is somewhere else using a LED holder.

You are free to choose where to drill the holes for the input/output, gain/diode switches and DC jacks and the footswitches as it is very taste dependent.

Note: Really take some time to determine where and how to place the pots, switches, jacks and PCB in the enclosure before you start drilling. Measure twice, drill once. I suggest you drill the holes for the pots first, insert the pots and attach the not yet populated PCB. Measure where to drill the gain boost switch (if applicable) and LED

At the back of this document you will find a drill template to make it a bit easier.

Populating the PCB

Before starting with this section, make sure you have read the modifications section first!

Soldering this board can be very complicated for some people since the solder pads are very close together. Use a magnifying glass to make the job easier.

The trick to soldering a PCB is to work from small to big components. My building sequence suggestions in this section are based on the parts I used myself. Sometimes some components are smaller (or bigger) so always use your own common sense and change the order accordingly. Usually capacitors can differ a lot in size depending on their rating and value.

Note: Do not blow on your solder in an attempt to cool it down. That may result in a bad join that might corrode! Also take extra care not to short components or to overheat them.

Start by soldering the resistors and jumpers (if applicable). If needed you can create a jumper using a spare piece of lead from a resistor or diode. Next come the diodes **D2 - D8**. Watch out! Germanium diodes are very fragile. Measure if they are not defective before soldering them. Now solder the IC sockets.

Now continue by soldering the MLCC, SMF/MKT/Wima capacitors and finish with soldering thes Electrolytics. You are almost ready rock!

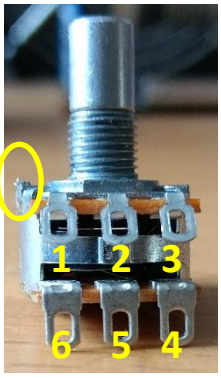
Off board wiring

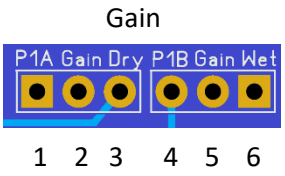
You can either mount the potentiometers directly to the PCB with special potentiometers or use the more traditional solder lug potentiometers.

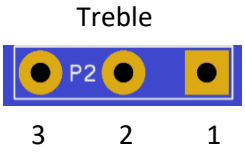
Potentiometers non PCB mounted

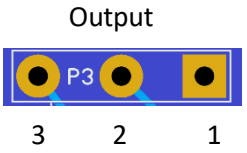
In the pictures below you see the correct pin numbering of the pots (Alpha 16mm style).
Solder the wires accordingly and it is always a good idea to twist the wires together to create some extra shielding against external noise. The rectangle pad marks the pad for **pin 1**.

You can break off the pin I marked with the yellow circle with a small pair of pliers.









The wiring for the switches is the same, marking position 1 with a rectangle pad. It is a good idea to angle the top row of pins of the stereo pot about 45 degrees. This will give you a better fit for attaching the wires (top right picture).

Potentiometers PCB mounted

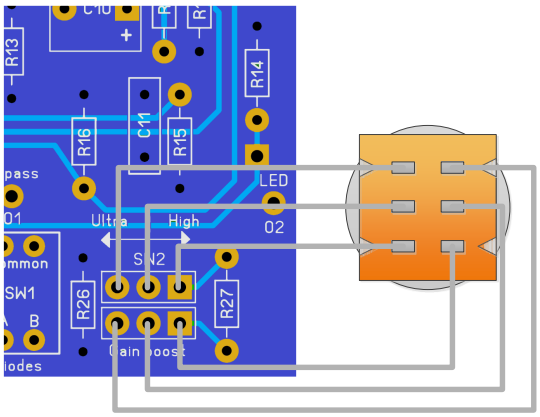
P1 (A and B) are still not PCB mounted and can be wired as mentioned in the previous paragraph.
Start by mounting **Output** and **Treble** pots in the enclosure and make sure you first removed the pin with a small pair of pliers.

I would advise to use some standoffs on the PCB to make it extra sturdy in the enclosure. Especially the standoff in the lower right corner of the PCB. Do not yet tape them to the enclosure because after attaching the pots, you will still need to do a lot of wiring! Now attach the bottom side of PCB to the pots until they fit, but do not solder them yet! Next, mount the Gain pot in the enclosure and if applicable the gain boost switch and status LED (**D1**).

If all fits well, solder the pots to the PCB, else adjust where needed.

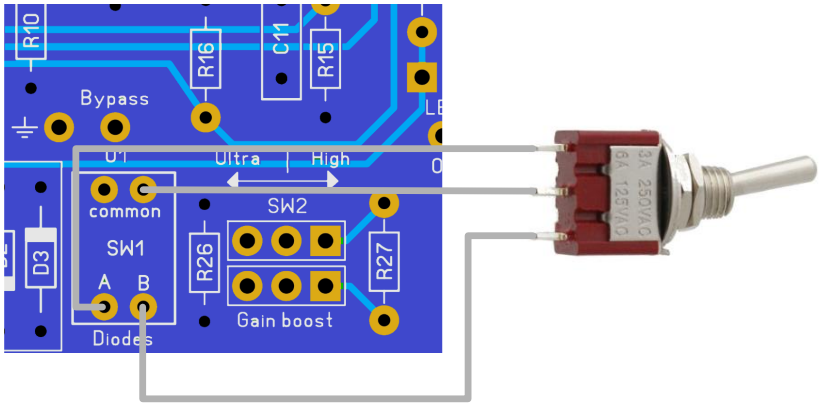
Switches

Gain boost

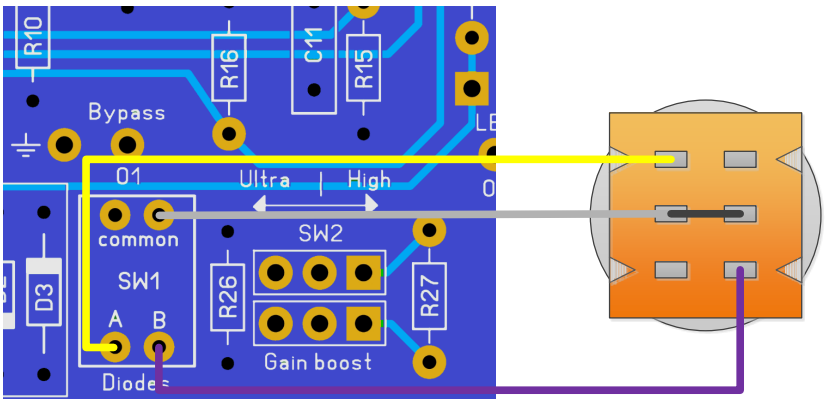


Clipping diodes

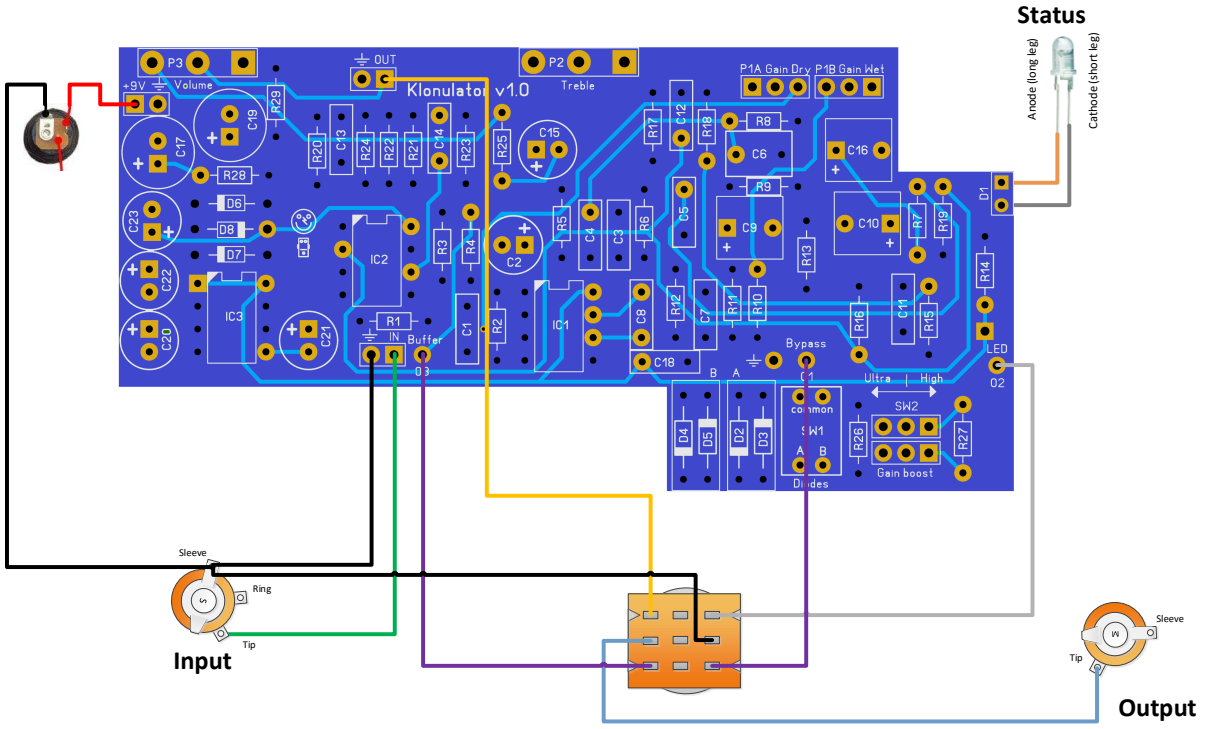
If you want to be able to switch the diodes externally and not via the internal DIP switch, then leave out the DIP and wire it like this using a SP3T (ON/OFF/ON) switch:



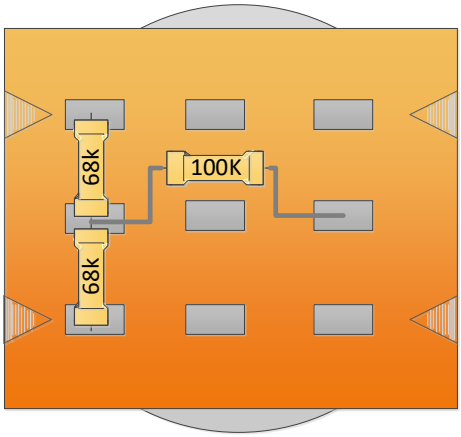
Or as a DP3T On/Z/On switch so you can also switch all diodes on (but not off!) Test the pole that are active in the middle position before you solder it.



Original wiring (Buffered bypass)



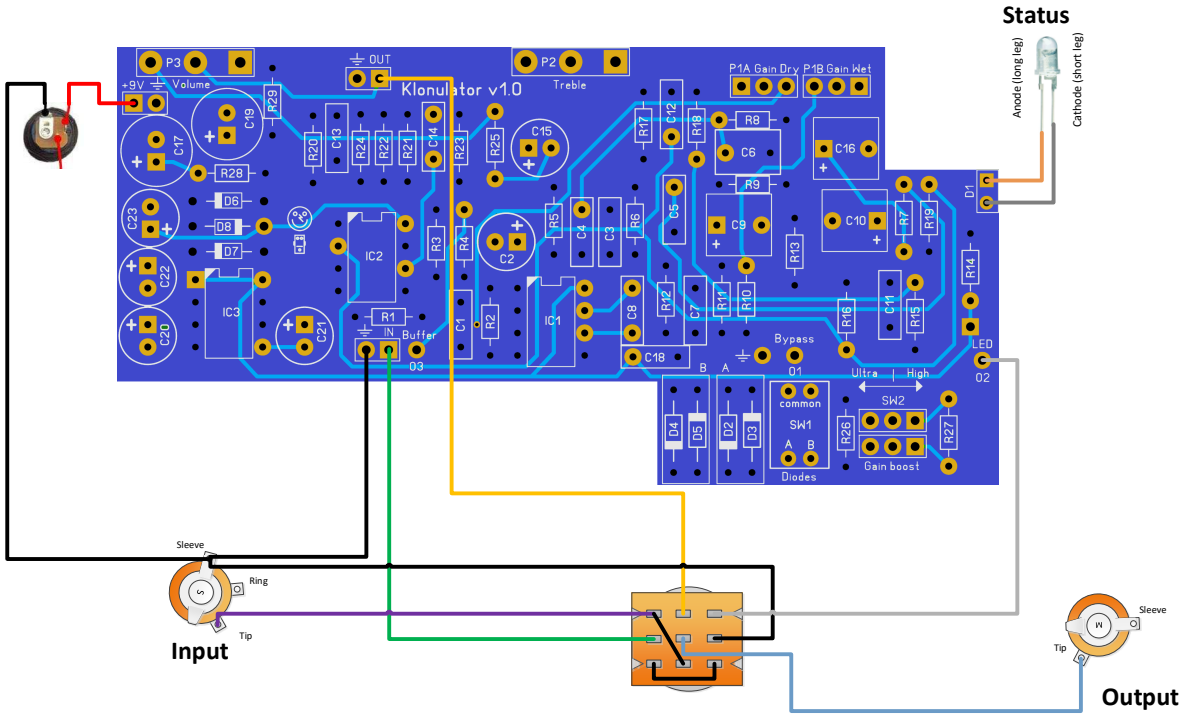
There is no wire from the output sleeve to ground so make sure there is good electrical contact between the jack and the enclosure, else you will need to solder a wire from the output sleeve to the input sleeve. The footswitch needs to have the missing output mixer resistors fitted directly on the switch like this:



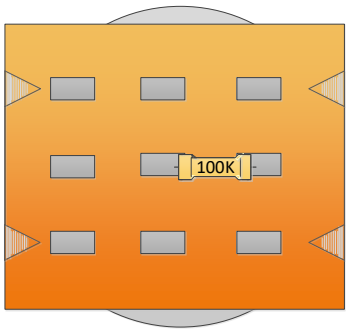
Note that you can also use a DPDT footswitch for this instead of a 3PDT!

True bypass wiring

There are several ways to fit true bypass to this PCB. First off I suggest you leave out the output mixer resistors (both 68k). If you really want to, you can keep the 100k between the output jack and ground although not really necessary.



There is no wire from the output sleeve to ground so make sure there is good electrical contact between the jack and the enclosure, else you will need to solder a wire from the output sleeve to the input sleeve. The footswitch needs to have the 100k resistors fitted directly on the switch like this:



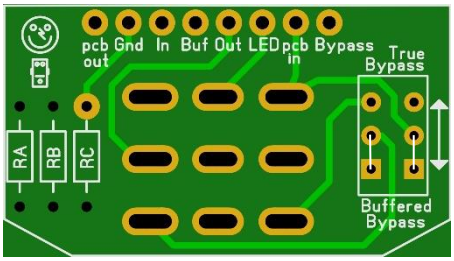
Klon Bypass PCB

When you bought a klon bypass PCB, you can solder and wire it up as follows.

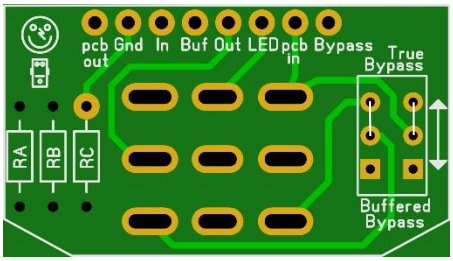
RA and **RB** are 68k. **RC** is 100k. Some clones do not use **RA**. The miniature DPDT switch lets you switch between regular buffered bypass and true bypass. There are some handy switches you can buy on Aliexpress or Ebay.

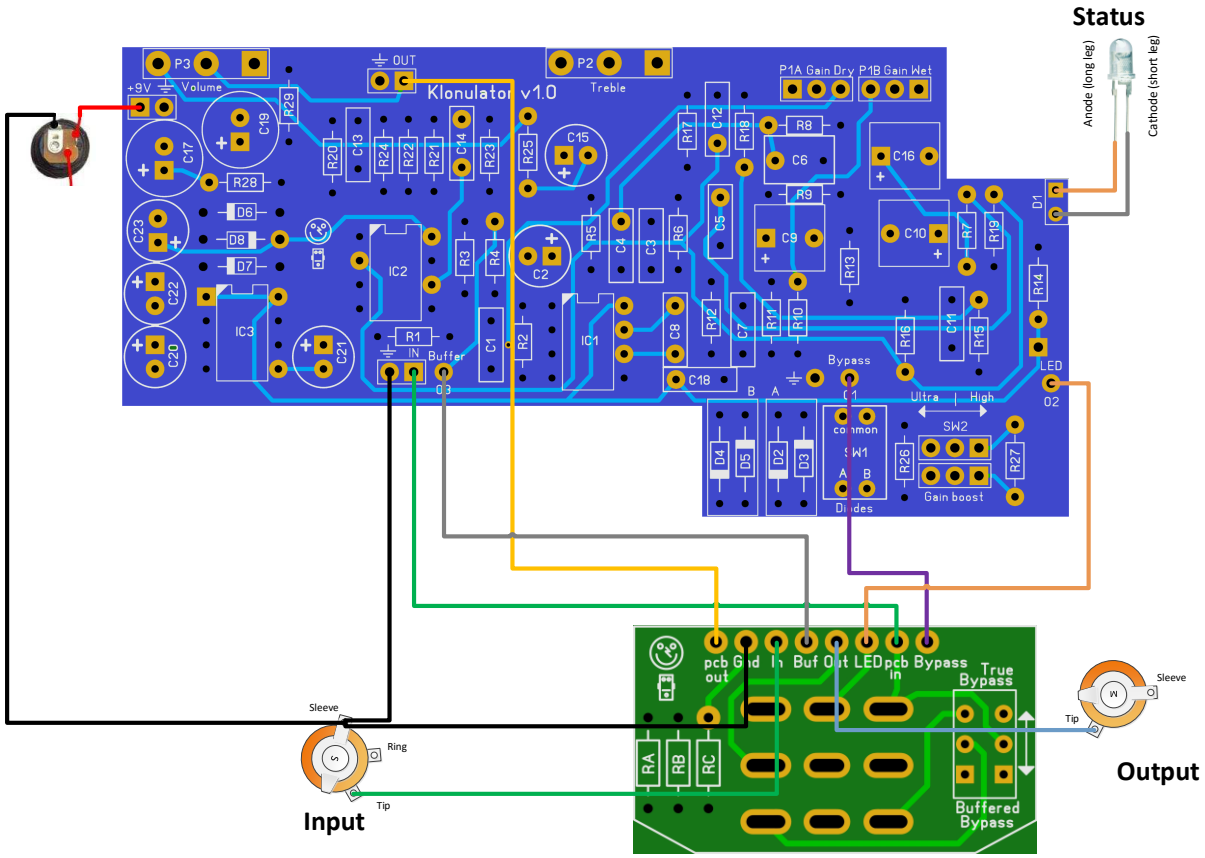
If you do not want to use the mini switch then you can connect the pads using a piece of spare lead wire like this:

Buffered bypass



True Bypass



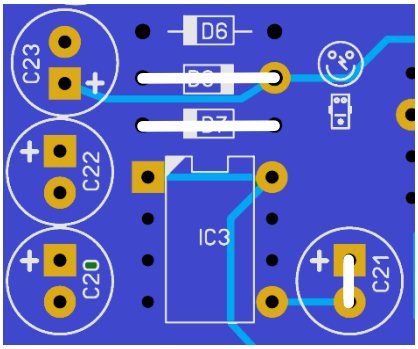


Modifications

Bare bones power section

Some do not see the added value of the charge pump as it is only there to keep IC2 from clipping. There are however clipping diodes before IC2 is reached so that way you can doubt the use of the charge pump. If you want to leave it out, do not install **IC3**, **D7**, **D8**, **C20**, **C21**, **C22** and **C23**.

You will need to solder a jumper on the pads of **D7** and **D8** as well as on the pads of **C21**.



Treble mod

There are some people who think the treble is bit thin on the original. You can fatten it up by changing **C14** from 3n9 to 6n8 or 8n2. You could also try a DP3T (On/Off/On) with (from left to right) a 4n7, 3n9 and 2n7.

Bass guitar mod

If you want to use the effect with a bass guitar, change **C1** from 100n to 390n.

Output Pot

Some people do not like the response of the output pot. If you are one of those, then change the B10k to a A10K.

Output mixer

Arguably a missed discussion in a lot of clones. You will not find these on my PCB as I moved them to the footswitch. In the original schematic these are the 68k resistors between the output and the buffer on one side and the effect on the other (**R26** and **R27**). Up until the 3rd revision of the original they were not on the PCB. There is no clear story on why they are added, but as a result there will always be a little clean signal leaking through the wet signal when the pedal is engaged. It is however the question if the clean part is at all audible through the overdriven sound. If you are a cork sniffer, then leave them on the footswitch, else you can leave them out and safe a few cents and trouble.

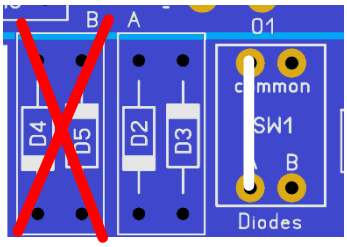
Op amp

You could use a OPA2134 instead of the TL072. Actually you can use almost any dual opamp as long as the pinout is similar to the TL072 and can handle up to +18V. Some suggestions NE5532, TL082, JRC4558P or JRC4580D. Feel free to experiment! There have been positive reports using a LF353 or JRC4558D for **IC1** and a NE5532 for **IC2**.

Clipping diode section

The Diode section has a lot of options. Let's walk through the standard layout. There have been many discussions on which diodes were used and finally the D9E and/or 1N34A were designated as "originals". This version of the PCB offers you the change to experiment. I suggest you put the originals in **D2** and **D3**, and your choice of diodes in **D4** and **D5**. My suggestion is to put some red LEDs in there or maybe other silicon diodes (1N4148, 1N914 or 1N4001). With the DIP switch you can then choose between both sets, or even none. Haven't we seen this before in another famous pedal? Well, yes… it looks like the diode setup in the King/Prince of Tone™. If you want to make the diodes externally switchable, you can use a SP3T switch and attach pads **A** and **B** to the outer lugs and the pad marked **common** to the center lug.

If you do not want the clipping diode selection then put the original diodes in **D2** and **D3**, leave out **D4**, **D5** and the DIP switch **SW1** and short Pad **A** and the **common** pad.



Gain mods

Splitting the Gain (P1)

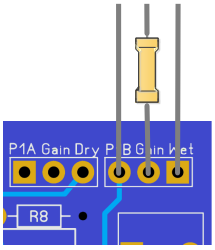
P1 is as true to the original a stereo potentiometer. When you choose to split it into 2 separate B100k potentiometers, you can adjust the mix of wet and dry signal. This can give some extra tone options to your pallet. You might have to consider a bigger enclosure due to the tight space in the 1590BB (try a 1590XX or bigger).

Adding Distortion

If you want to go into the distortion zone by boosting the gain but keep the frequency response of the original then use 2 red LED's for **D4** and **D5** (and set the **SW1** dipswitch to **A** Off, **B** On). Modify **R10** to 680R, **R11** to 4k99 and **C7** to 180nF.

Adding Growl with Gain boost

This is such a good mod that it is pre-built on the PCB. By default **R16** is 47k. By changing this to somewhere between 8k2 and 20k, the overdrive gets a nice growl while keeping the characteristics of the original. I suggest you populate the board as mentioned in the component section and off board wiring section. This way you can choose between the original 47k (middle position) and a high and even ultra gain boost. The lower the resistance, the higher the gain gets, but if your effect starts squealing or motor boating at high gain pot settings, then add a resistor in series to lug 2 on **P1B** with a value between 20k and 47k, like this:



If you do not want to use this mod then leave just out **SW2, R26** and **R27**. You will also not need the optional resistor on lug 2.

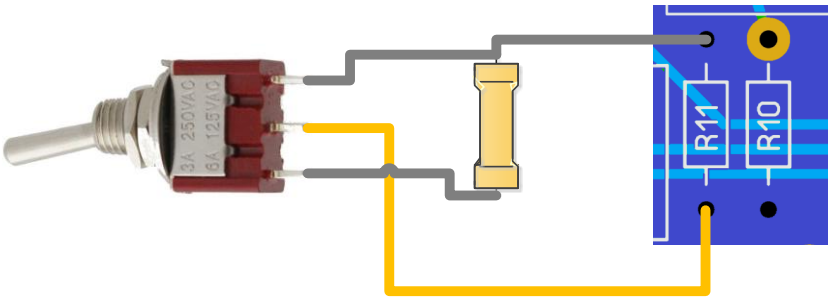
Silver Pony

Also a well-known mod based on the reverse-engineered silver Centaur, serial S2207 (by Build Your Own Clone). The brought up some different valued components.

Changing **R10** to 47R will give you some more gain. Changing **R17** to 10k and R18 to 4k7 will give more volume and treble for the dry signal. Change **R21** to 4k7 and **R23** to 1k8 to shift the range of the tone knob. Lastly, changing **C13** to 560p will increase the overall brightness.

1994 specs

In 1995 **R11** was introduced among some other value changes, but they do not really affect the tone. **R11** will boosts the bottom mids a bit. Before that, there was a jumper as **R11**. You could make it switchable using a SPST.



Troubleshooting

All PCB's have been 100% factory e-tested and out of every batch I receive I build an effect to double check, so there should not be a connection problem on the PCB itself.

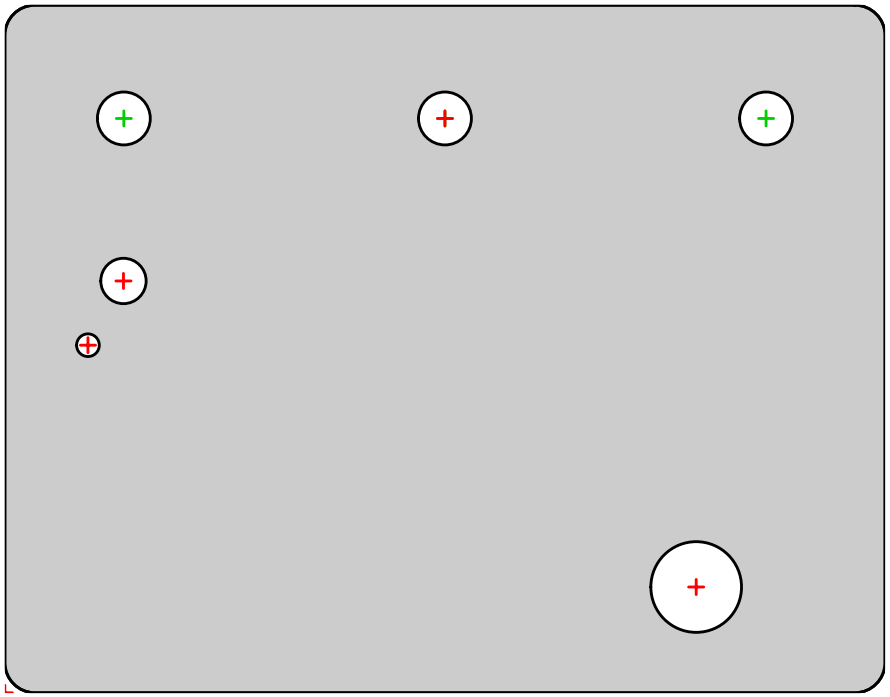
The board is not working (at all), what now?

- Check if your 9V is plugged in correctly (and/or soldered correctly on the board). Pay special attention to the polarity.
- Check that you oriented the capacitors, IC's ,transistors and diodes the right way. SMF, MKT and ceramic capacitors as well as resistors do not need to be oriented. A likely sign of incorrect capacitors and/or orientation is when an effect is sputtering, rumbling or "motorboating".
- Check if you used the correct values of the components. For resistors you can look here: http://www.diyaudioandvideo.com/Electronics/Color/
- Double and triple check your soldering! A loose or cold solder can be really bad for your board.
- Replace the IC and/or transistors, one might be defective. Before doing that first unplug the 9V and wait for 5 seconds.

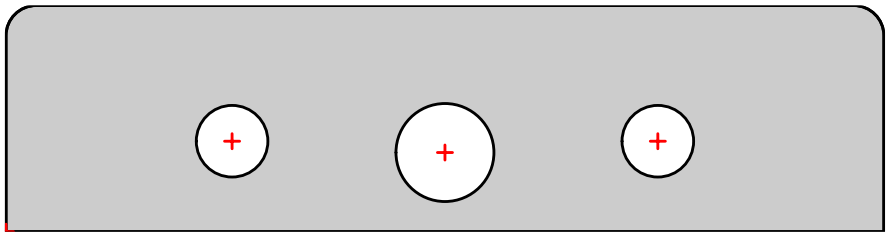
Check that you have good/high grade components. A lot of Chinese sourced parts are fakes (especially high end opamps, audio capacitors, vintage diodes and transistors) so be careful that you source your parts from reliable suppliers.

Drill Templates

Topside Hammond 1590BB



Back side Jacks Hammond 1590BB



Schematic