

# GK RAILS DIY PCB GUIDE V1.1

**Circuit Designed by:**

Frank A. and John K.

**PCB by:**

Deadbug Circuits

**For More Information:**

<https://www.talkbass.com/threads/gk-rails-in-a-pedal.1122467/>

## About:

The GK Rails was designed by TalkBass forum members Frank A. and John K. **I claim absolutely none of the circuit design as my own**, all I did was take their schematic and design a PCB for it since it did not appear to have a PCB available, only a protoboard layout.

The GK Rails pedal is meant to emulate the "growl" of the Gallien-Krueger 800RB's power amp section. This sound is most notably found on Guns N' Roses' Appetite for Destruction album. A very mid-heavy pushed bass tone with some grit. Engage the boost circuit and you can get to some near fuzz territories.

While this pedal was designed for bass, it also sounds great on guitar if you are into the doomy stuff. It lets some clean signal through underneath the driven signal which nails the cleanish doom riffs, think Kyussish kinda stuff.

## Controls:

### Pre:

The Pre control determines how much signal goes into the power amp simulator section of the pedal. This control acts most similar to a gain control.

### Post:

The Post control determines the final output volume of the pedal. While this is just a volume control, it is very interactive with the Pre control and also acts almost like a gain control sometimes.

### Frequency Cutoff:

The Frequency Cutoff decides the cutoff of a low pass filter placed just before the post control. When in the up-position frequencies above 2.3 kHz are cut off, in the down-position frequencies above 4.8 kHz are cut off, and in the middle position no additional frequencies are cut off.

### Boost:

The Boost Control adjusts the gain of the boost circuit. When the boost footswitch is off, this control will have no effect on the sound because it is bypassed.

**For any questions you can contact me here:**

[deadbugcircuits@gmail.com](mailto:deadbugcircuits@gmail.com)

Or just DM me on Instagram - @deadbug.circuits

## BOM:

### Resistors

Qty	Value	Parts
5	10R	R20, R21, R28, R31, R32
2	100R	R12, R25
2	220R	R4, R5
6	1K	R1, R9, R13, R23, R24, R33
1	2.7K	R27
1	3.3K	R19
1	4.7K	R10
1	5.6K	R7
1	10K	R26
3	12K	R6, R11, R15
3	47K	R14, R29, R30
2	56K	R16, R22
2	100K	R17, R34
1	220K	R8
1	1M	R18
2	2.2M	R2, R3

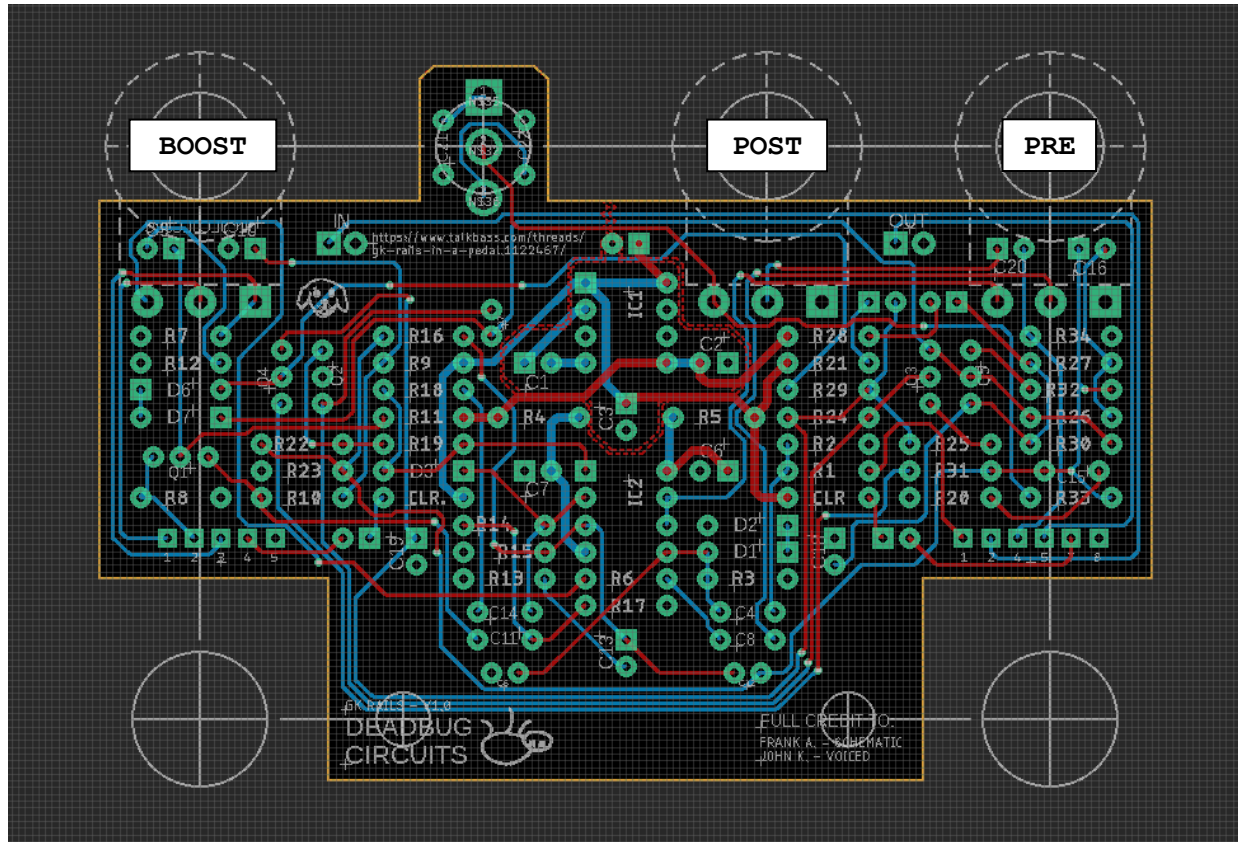
### Capacitors

Qty	Value	Parts
1	27p	C12
1	47p	C5
1	100p	C17
1	10n	C15
1	33n	C21 *Keep the toggle switch in mind when soldering this cap
1	68n	C22 *Keep the toggle switch in mind when soldering this cap
4	100n	C4, C8, C11, C14
1	4.7u	C20
2	47u	C2, C3
6	10u	C1, C6, C7, C9, C10, C13
2	100u	C18, C19
1	220u	C16

### Other Crap

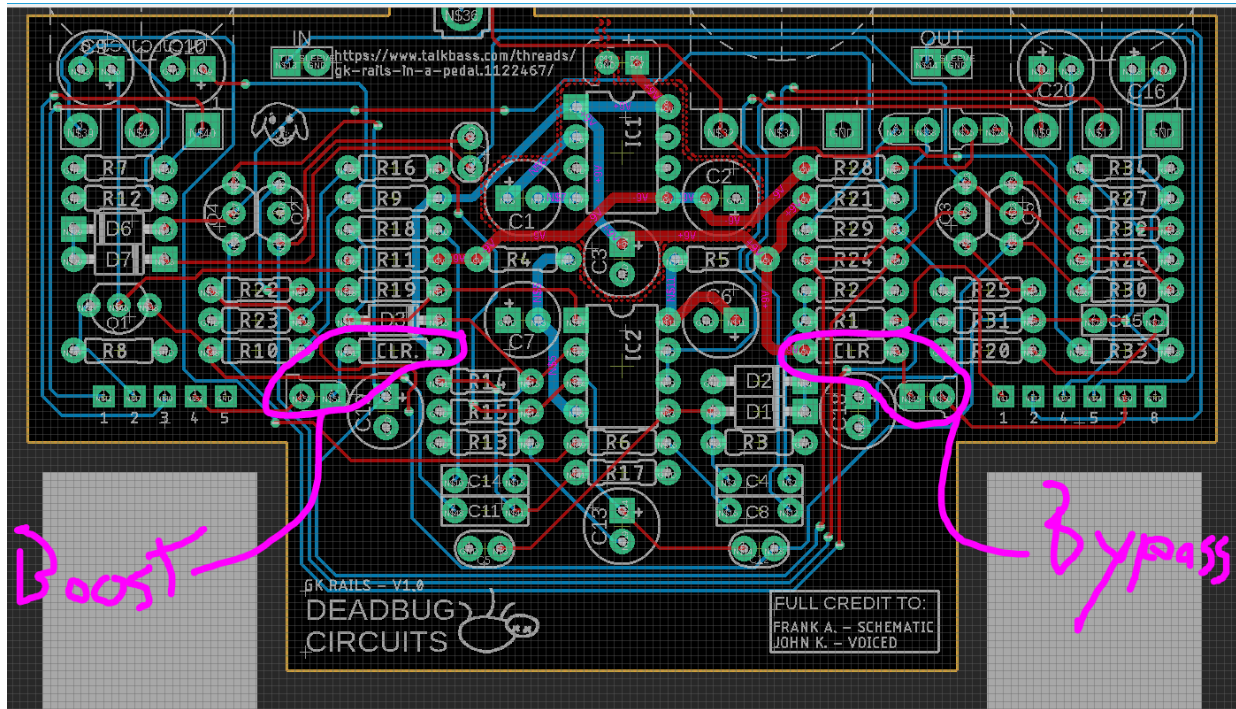
Qty	Value	Parts
2	1N4733	D1, D2
4	RED LED	D4, D5
3	1N4002	D3, D6, D7
1	LT1054	IC1
1	LF353	IC2
1	2N5457	Q1
2	2N4401	Q2, Q4
2	2N4403	Q3, Q5
2	A10K	POST, PRE
1	B50K	BOOST
1	SPDT	SW1
1	1590BB	Enclosure

# Potentiometers



Here is the layout of potentiometers, Boost is a B50k, Post and Pre are both A10k.

## LEDs and CLRs



The CLR resistor and LED on the left side of the board are for the Boost indicator light.

The CLR resistor and LED on the right side of the board are for the bypass indicator light.

\* If drilling the LEDs where they are on the drilling template, the legs of the LEDs will have to be bent accordingly to match up with the holes in the enclosure\*

\*Depending on what color LED you use, and how bright you want it, adjust the value of the CLR resistor to taste\*

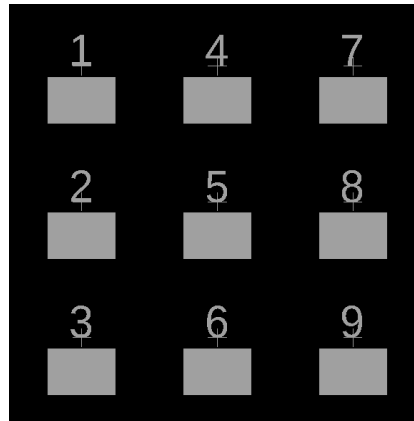
\*The positive leg (long leg) of the LED goes to the square pad, and the negative leg goes to the circle pad\*

## Off-Board Wiring

### Bypass Footswitch:

Use a 3PDT Footswitch. If using the supplied footswitch PCB, just match the numbers from the footswitch PCB and the main PCB to each other, that's all.

If you are wiring it manually, each number on the PCB corresponds to a lug on the footswitch, just connect a wire from the PCB to the respective lug on the footswitch:

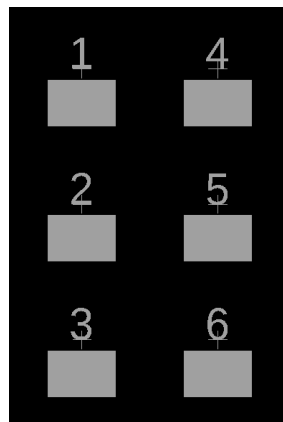


*Lug numbering for a 3PDT footswitch*

A jumper will also have to be placed across lugs 3 and 6 if wiring manually.

### Boost Footswitch:

Use a DPDT Footswitch. Each number on the PCB corresponds to a lug on the footswitch, just connect a wire from the PCB to the respective lug on the footswitch:



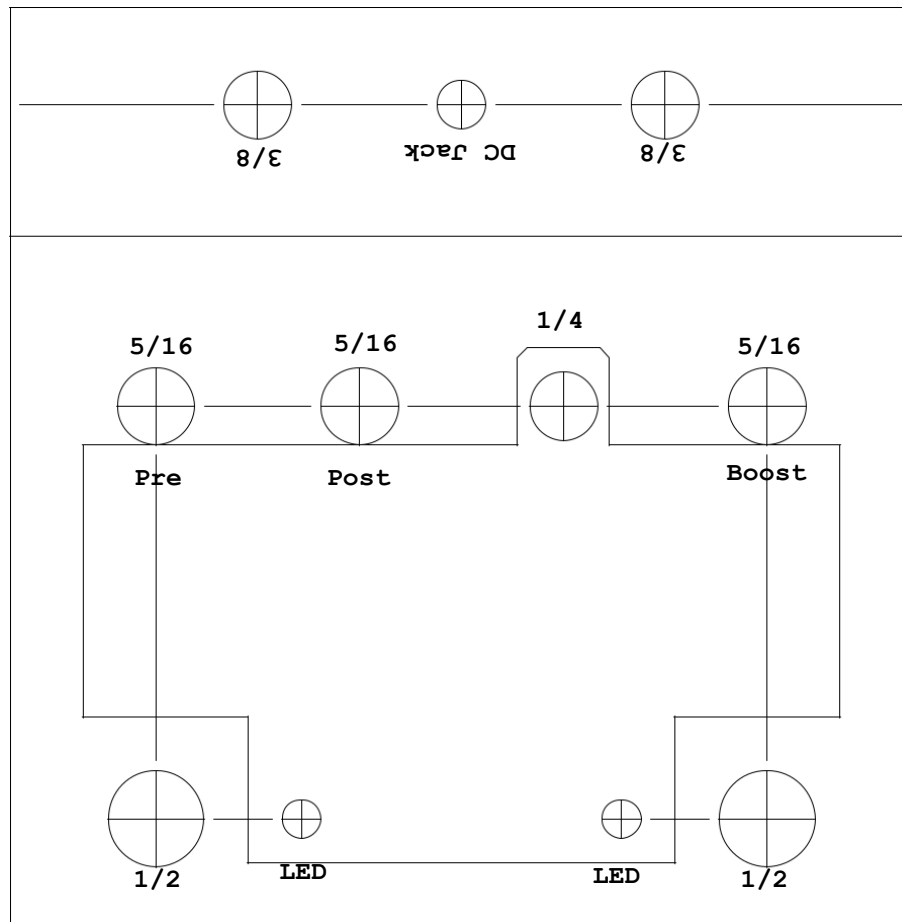
*Lug numbering for a DPDT footswitch*

# Drill Template

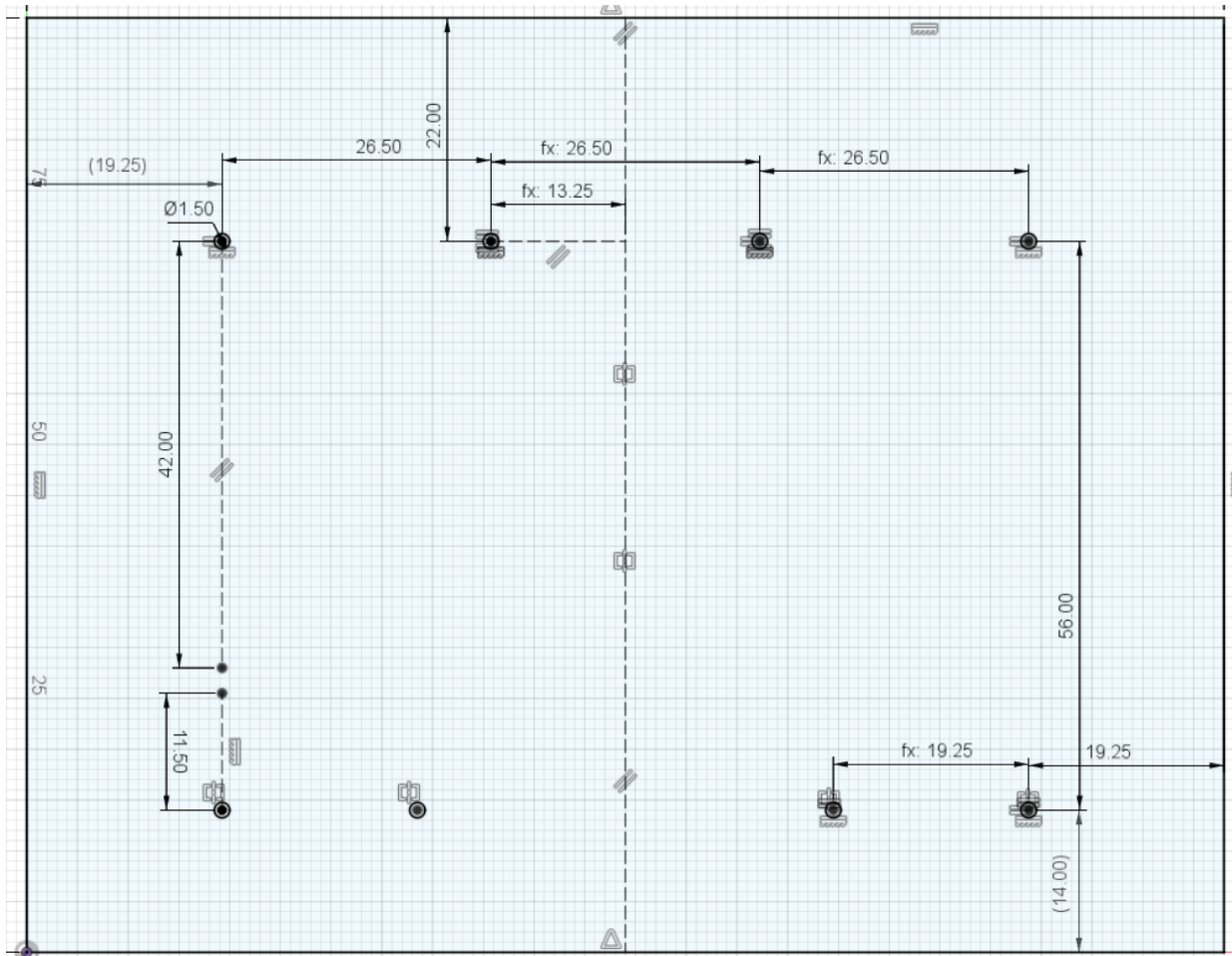
Do not scale! Print at full scale.

Since there are so many different 1/4 and DC Jacks out there, double check clearances with the potentiometers and switches before drilling holes.

If drilling the LEDs where they are on the template, the legs of the LEDs will have to be bent accordingly to match up with the holes.



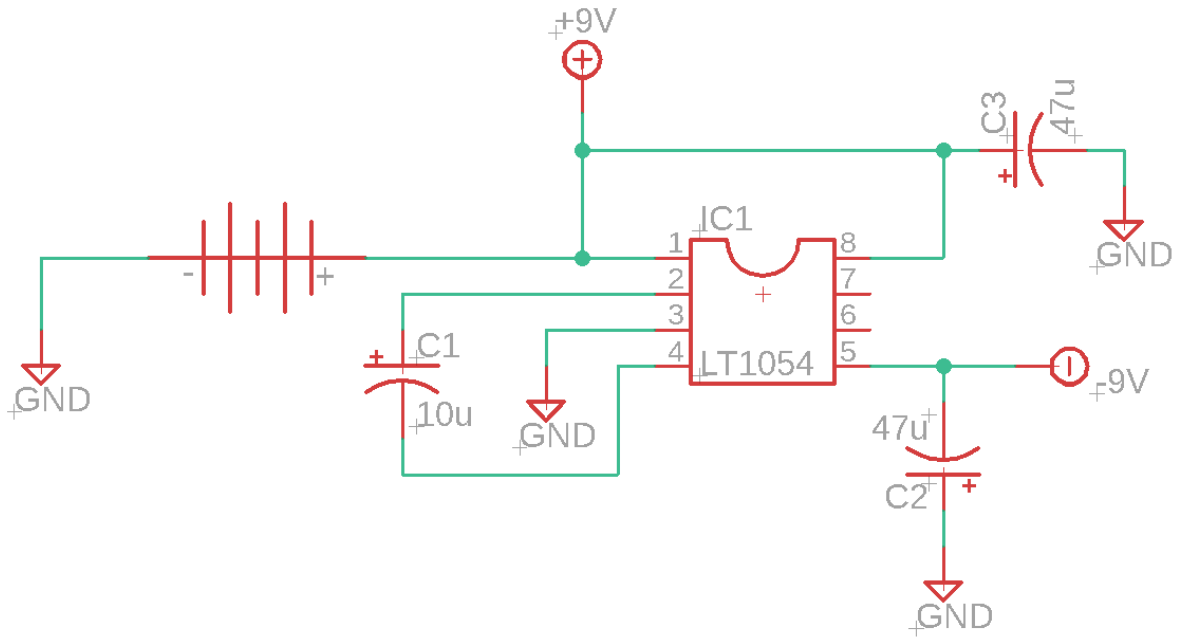
# Drill Template (in mm)



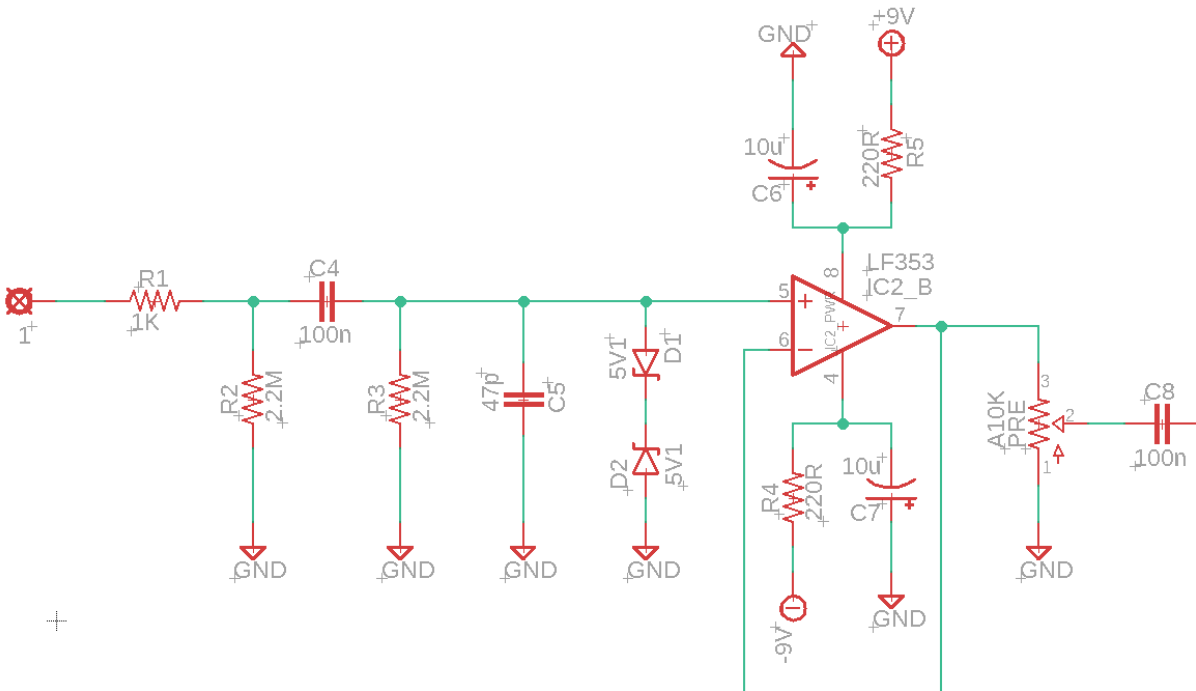


# Schematics

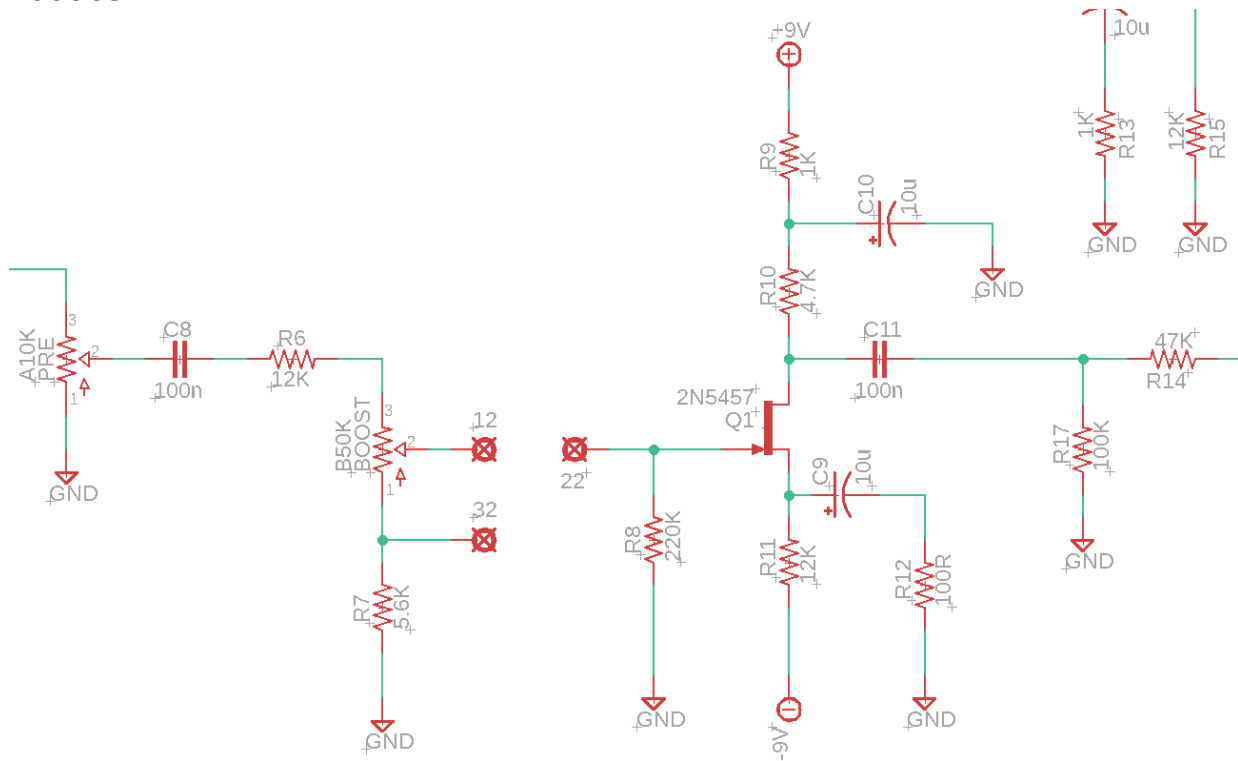
## Power



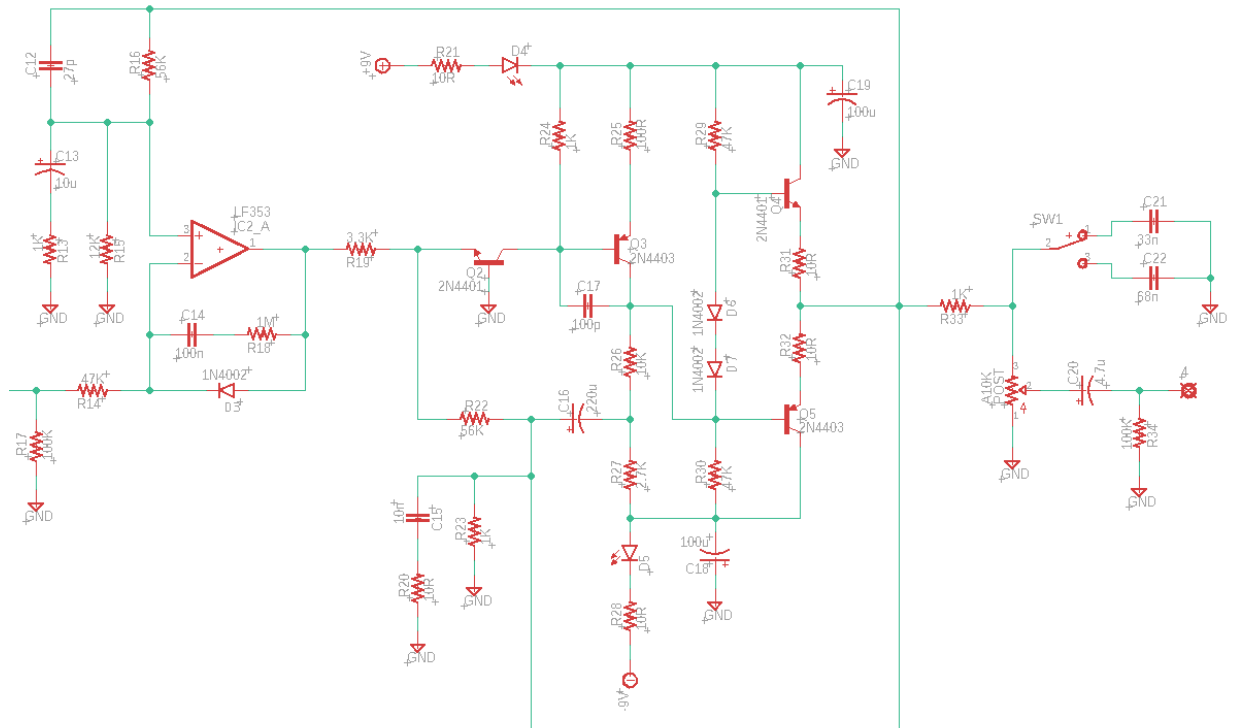
## Input Buffer



## Booster



## Power Amp Simulator



# Full Schematic

Sorry it's a bit blurry  
It's the best I could do since it's so long

