Zorglonde
Mounting instructions.

Summary

Important notice.

This DIY kit is not that easy and require a bit of knowledge. If you're a beginner, you're likely to go into some hardcore problems and should try easier kits before. There's a debugging chapter that you can check in case of emergency, but:

- I cannot be held responsible of any malfunction or a component burning. This board has been tested and I use it in when I build pedals myself. It's 100% functioning when everything is done correctly.
- The debugging chapter cannot take in account all the problems you may reach. (Murphy's law you know....)
- I won't refund any malfunctioning kit that has been mounted.
- Here's what you should do in case of problems:

  1- Keep calm.

  2- Check another time that each component is at it's right place and all the solders are ok.

  3- Check the debugging chapter at the end of this document.

  4- Ask for help in your surrounding family or friends. Someone who can see, plug, check and test your board is more valuable than someone on a forum or mail 10000km far.

  5- Check the freestombox forum, and ask for help if needed. When asking for help be sure to give the maximum of information: TODO

  I may reply to you on freestombox, I check it sometimes.

  6- Mailing me is the very last thing you will do. And if you do, be sure to write the maximum of information I need to answer you. Yes you may add pictures if you think it's relevant. Mails with only "My kit is not working" will be either ignored, either replied with a kind of passive aggressive tone, if not clearly aggressive... After all this is "Do it YOURSELF" and not "Zorg, can you do it for me please?", and I'm always under a heavy load of work, so please spare me at the maximum!!!
- Any feedback on this document is welcome. If there's something missing, something you don't understand, something you're not sure, if you reworked the document with better explanations, pics and pink elephants, grammatic or ortografik faults, please feel free to mail me.

What's in the kit?

This is all you must find in your Zorglonde kit:

Part number	Value	Qty
C2	Capacitance 100nF	1
C3	Capacitance 100uF	1
C4 C10	Capacitance 1uF	2
C5 C9	Capacitance ceramic 100nFc	2
C6 C8	Capacitance 10uF	2
C7	Capacitance 220nF	1
C12	Capacitance Panasonic 10uF 50v	1
C11 C16	Capacitance 680nF	2
C13	Capacitance 6.8nF	1
D1	Diode 1N4001	1
D2	Led white	1
D3,D4	Diode 1N4148	2
DEPTH1,DEPTH2	Potentiometer stereo B100k (lin)	2
EXP_GND1, EXP_TIP1, GND1..GND3, IN1, OUT1, 9V1, LED	Connecteur SIL	10
P1	ICL7660s	1
Q1, Q4	Transitor NPN BC550	2
Q2	Transistor FET J107	1
Q3	Transistor FET J201	1
Q6	Transitor NPN 2N3904	1
R1, R4, R7..R9, R14, R20, R27, R29, R31	Resistance 10k	10
R2,R3, R15,R16	Resistance 1M	4
R5, R19	Resistance 47k	1
R6	Resistance 4k7	1
R10	Resistance 82k	1
R11, R17, R24, R25, R30	Resistance 1k	5
R12, (R35)	Resistance 39k	2
R13	Resistance 100/150	1
R18	Resistance 3k9	1
R21	Resistance 560	1
R22	Resistance 20k	1
R23, R33	Resistance 27k	2
R26	Resistance 4k3	1
R28	Resistance 470k	1
R32	Resistance 2k	1
R34	Resistance 200	1
R36,R37	Dip switch	2
RATE1	Potentiometer C100k/C5K (rev log)	1
SW2,SW3	Switch 2PDT	2
Trimpot1	Trimpot B100k (lin)	1
U1	AOPTL074	1
VOLUME1	Potentiometer A10k (log)	1
Support CI 8 pins +14 pins		3
Led socket		1
Footswitch 3PDT		1
Enclosure +wood frame		1
PCB		1
JACKS 6,35mm		3
JACK 6,35mm stéréo		1
Jack DC		1
White knobs		3
Wire, eat shrink tube, window insulator		

According to the RATE range option you choose, the following components should be found in your kit:

Normal: RATE1= C5KOhms (Rev Log). R13 = 150 Ohms.

Extended: RATE1=C100kOhm (Rev Log). R13 = 150 Ohms. R35 = 39kOhms.

Components numbers in the left column, C1, R1 etc. are tied to the PCB's marks.

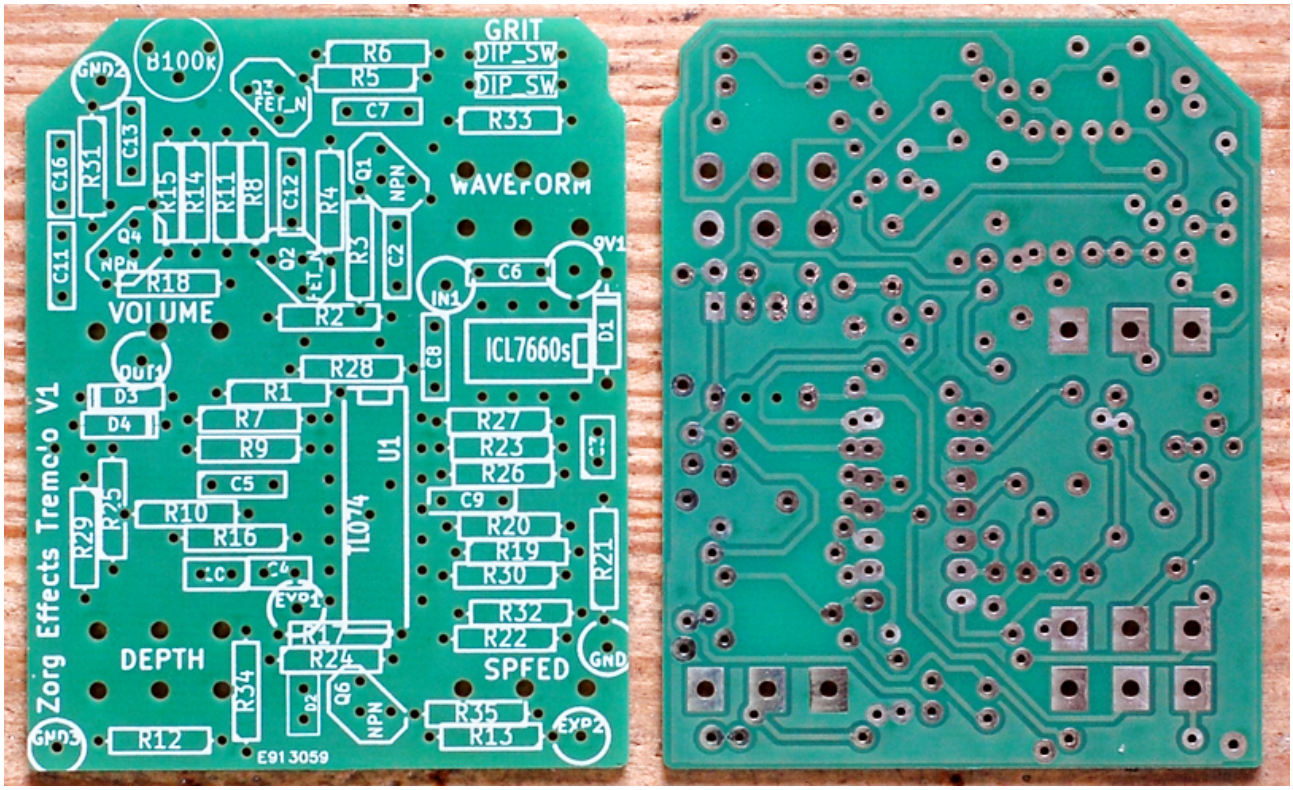
What you'll need.

The following tools are needed to build your Zorglonde pedal:

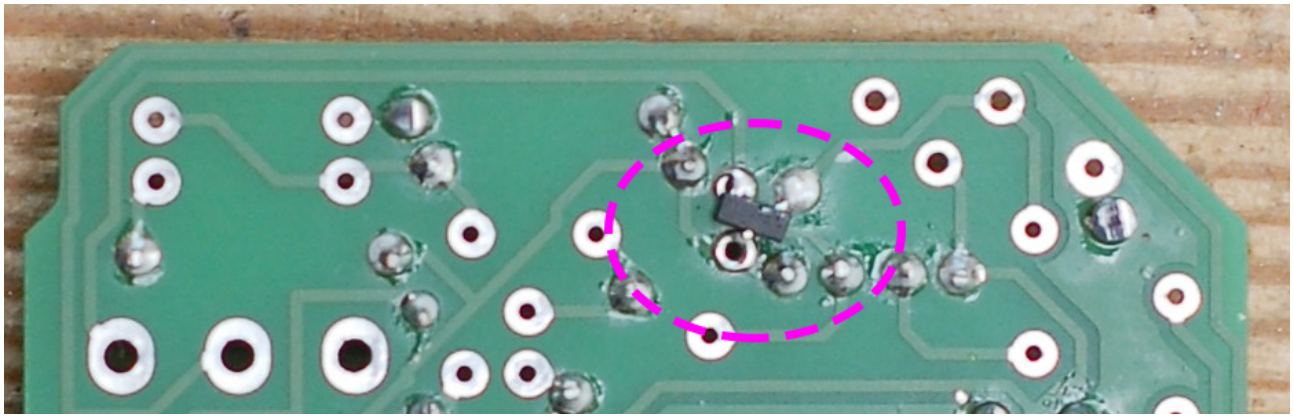
- A soldering iron.
- A un-soldering pump.
- A voltmeter/ohmmeter.
- Pliers to cut wire and remove the wire sheath.
- Pliers to screw nuts.
- A cruciform screw driver.
- And eventually wrenches.
- A 9v dc power unit, center negative.
- an oscilloscope, and a frequency generator.

Soldering on the pcb.

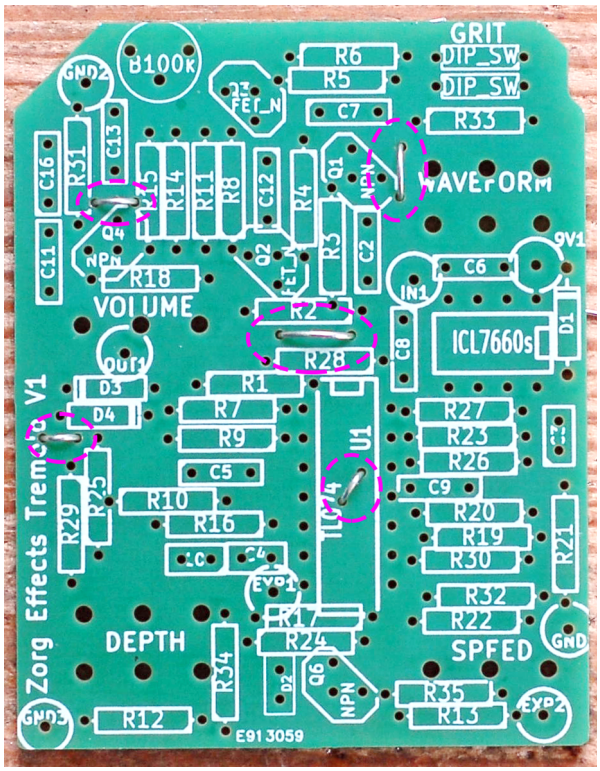
This is the PCB (Top/Bottom):



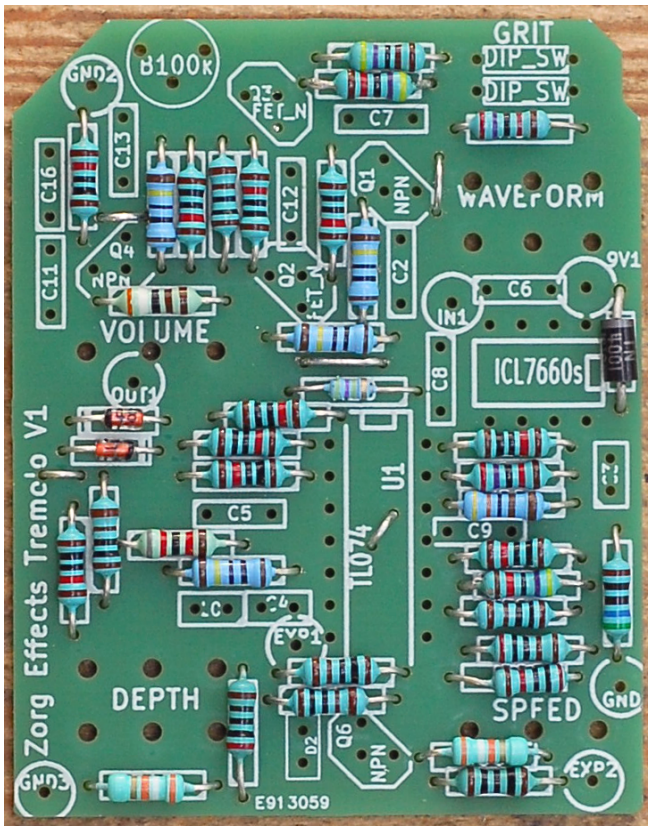
On copper side, begin by soldering the J201 CMS transistor. Beware of its direction, they must be like on the picture below. The grid being the single leg on the rectangle's side. Because of the transistor's size, one must use a precise soldering iron. Be careful that the solders do not overlap each others. Test with your multi-meter if needed.



Flip the board and solder the 5 jumpers by using resistance's legs.



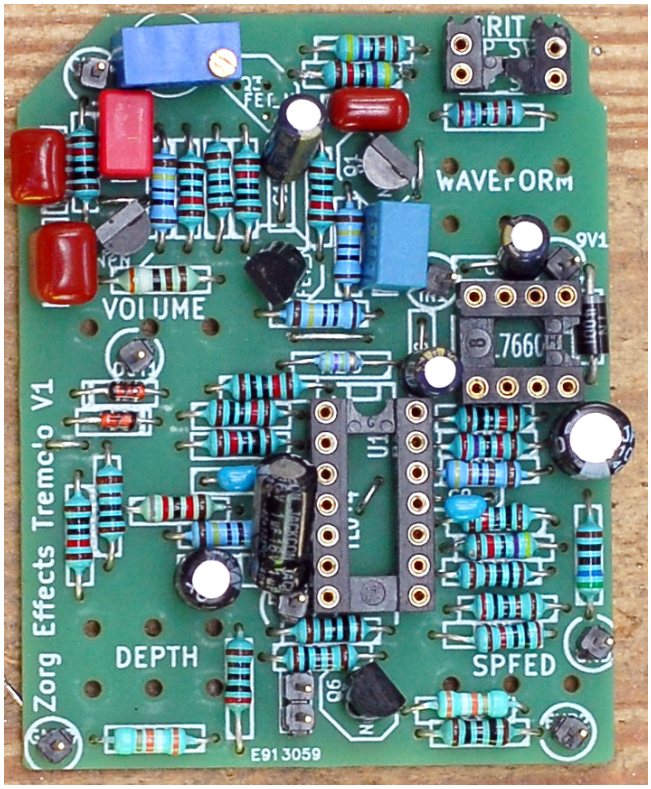
Then we're going to solder components from the smaller to the taller. First, diodes and resistances. You shall take care of the diode position. It MUST be on the same direction as on this picture :



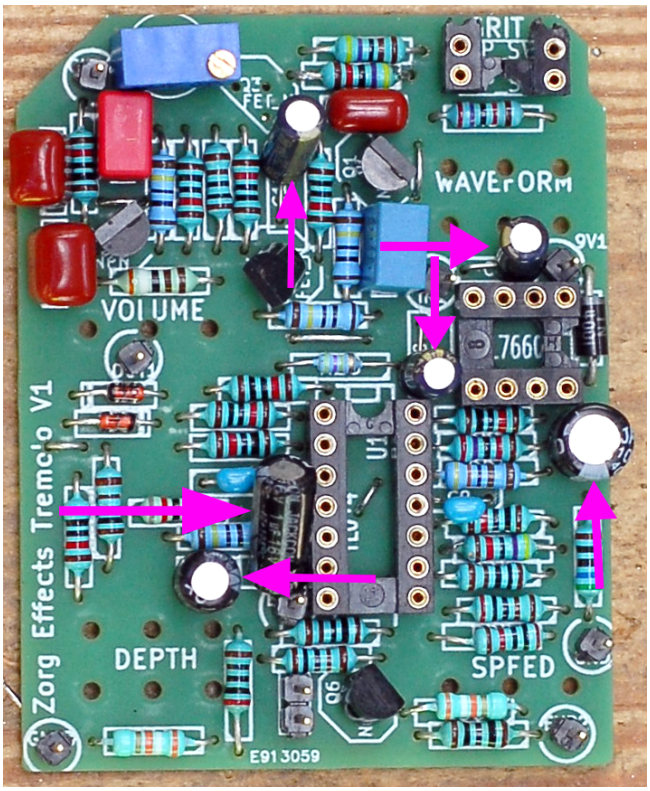
Note: The above picture is for extended range of RATE. For normal RATE range R35 is not soldered and R13 is 150 Ohms.

And then solder by size :

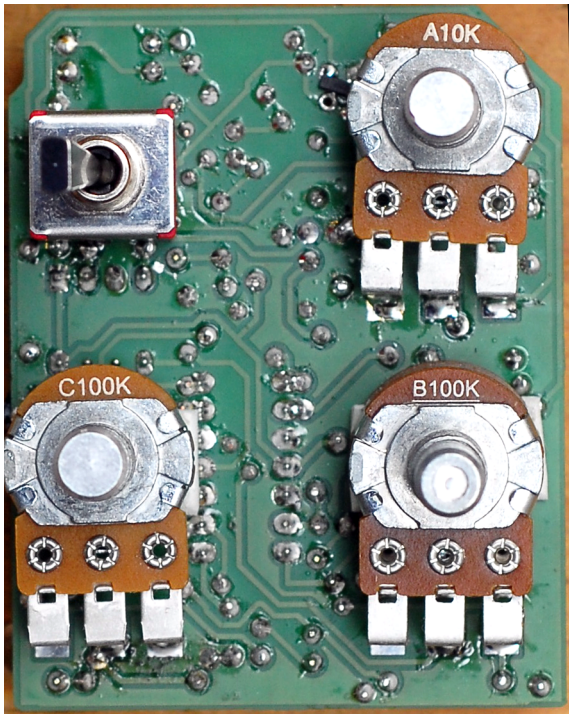
- The decoupling caps C5 and C9
- CI supports. Use half of a 8 pins support to elevate the upcoming mini dip switch.
- Transistors Q1, Q2, Q4 and Q6, beware of their directions! Important note: transistor J107 MUST be paired with resistances R12, R26, R23 and R27. The transistor given in the kit has been specially selected to meet this requirement. But if you've brunt this transistor or sourced the components from someone else than me, do mount CI supports pin instead of Q2 to help choose the right transistor.
- Caps C3, C6, C8, C11 and C16. (Beware of directions for C3, C6, C8. See picture on next page)
- Cut the SIL connector to make terminals used for GND1,2,3, IN1 etc…
- Les caps C4, C10, C12, C2. (Beware of directions for C4, C10, C12, See picture on next page)
- The trimpot



Beware, electrolytic caps C4, C10, C12, C3, C6, C8, have a direction. Check them out on the picture below. Their white stripe should match the arrows :



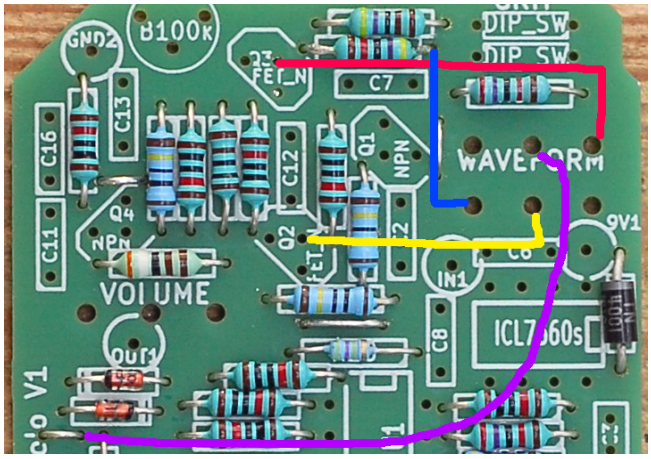
Now flip the board. We're going to solder the switch and pots on the other side.



Note: On normal RATE range, the pot is C5k.

Solder the DPDT first. After soldering it you must test that your switche's solders are ok. **This stage is really important: 99% of your problems will come from a badly soldered switch!!!**

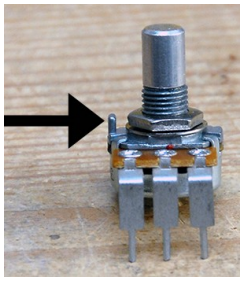
To do so, use your ohmmeter and check that the following path are not opened:



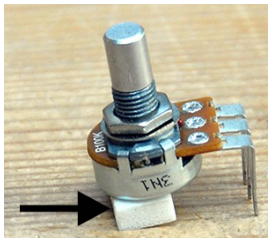
Now we're going to solder the pots on the same side of the switch.

First you'll need to prepare the pots:

Cut the little rectangular shaft next to the axis, you won't need it.



Stick 16mm length of window insulator under each pot. It's in order to prevent solders on the board to connect with the body of the pot and shortcut some circuits.



Now you are ready to solder them on the board. But don't go too quick! First put one of them and solder ONLY the middle leg. Then try to fit the card in the enclosure. It can happen that the pot is not in right the middle of the hole in the enclosure. If it's the case, you have only one solder to heat to move it a bit and rectify the position. Then add them one by one, soldering only the middle leg, and adjusting after each one to have them in front of their holes. At the end it should enter the enclosure without to much force (sometime a bit though).

Next, you'll need to prevent the stereo jack leg on the bottom right to touch the enclosure's bottom right corner when a jack is plugged in. In order to do such a thing, plug a jack in the stereo jack and crush the leg with pliers:

Now add the card in the enclosure. Screw the switches (there's no screw for "bad"). Add the wooden plate and screw the bypass led's socket to hold it.

Wiring the pedal.

So there you go with the card in the box.

Now you can screw the audio jacks, led socket, leds, footswitches and DC jack.

Important: Use the eat shrink tube to strengthen and protect all your wire connections (on the board and on the connectors).

Here's a list of the wirings to do:

9v1 goes +9v of DC jack (long leg if you wish to wire it center negative)

Gnd2 (top left of the board) goes to ground of the DC jack (short leg if you wish to wire it center negative)

GND1 and GND3 go to audio jacks grounds.

LED2 upper leg goes on the positive leg of the bypass LED (longest leg).

If we number the legs of the footswitch :

1 2 3

4 5 6

7 8 9

Then :

1 Is connected to IN1 on the board

2 Is connected to OUT1 on the board

3 Is connected to the negative leg of the bypass led (shortest leg, flat side)

4 Is connected to input jack

5 Is connected to output jack

6 Is connected to LED2 lower leg

7 and 8 are soldered together.

If the stereo jack for the expression pedal is numbered:

1 2 3

4 5 6

EXP1 and EXP2 are connected to legs 4 and 5.

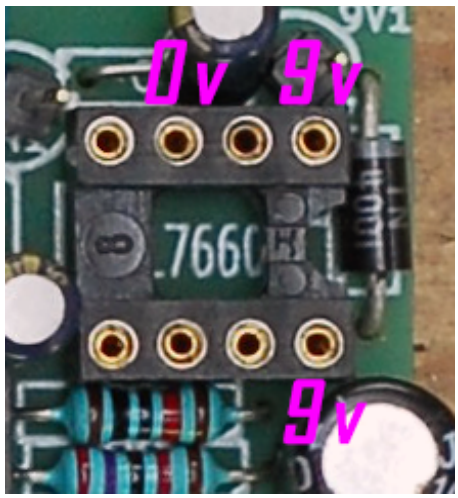
1 and 2 are connected together.

Now, there's only left to...

Test the board.

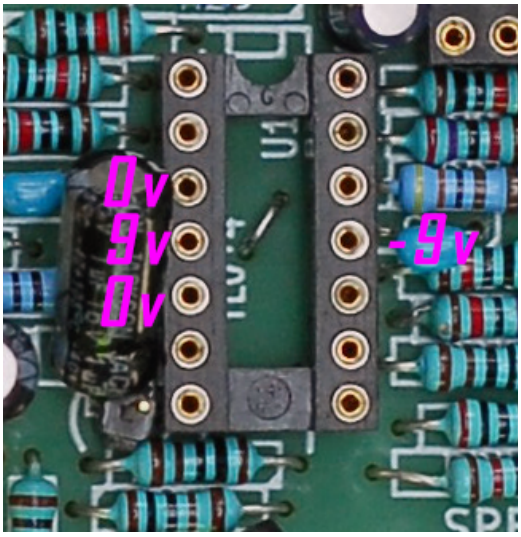
Now don't put the ICs in the box. First we're going to test the power supply.

Step 1: connect your 9v DC power unit to the DC jack. Check out the tensions on the ICL7660s's socket:



Step 2: If step 1 is ok, add the ICL7660s. Beware of it's direction, otherwise it'll blow up.

Then test the following tensions on U1 (Tensions between -8 and -9v are ok on leg 11):



Step 3: insert the TL074 in its socket. After this step when the pedal is plugged and not in bypass, the led should blink at the tremolo's rate and according to the rate knob's position. If it does not work move on to the debugging chapter.

Step 4: Bias the J201.

If you made everything fine, the Zorglonde should be working now. Plug in your favorite instrument and rock it! (And then screw all the pots screws as well as the knobs).

If this is not working you're good to read the…

Debugging chapter.

First, voltages!

If at step 1 of tests chapter you don't have 9v voltages on the DC plug check:

- That your DC power unit is working.
- That the connections between the plug and the board are ok.
- The directions of your diodes and caps.
- Check you soldering. Big soldering can overlay others tracks, that’s bad.

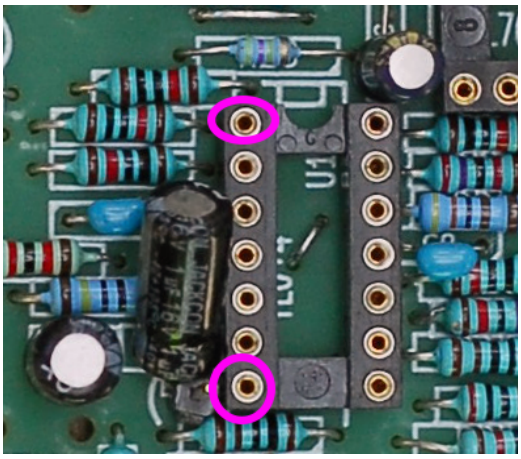
If after step 3 of the previous chapter (test the board), the LED does not blink, there’s the following possibilities:

- The LED has been soldered backward.

- Your TL074 is dead.

- Some resistances are not where they should have been or their soldering are bad.

The LED is directly on the oscillator’s output. If it doesn’t work, it may be that the oscillator doesn’t work. To check if the oscillator works, set the RATE pot to the minimum and with a voltmeter or oscilloscope check the tension on the following TL074’s leg:



If the oscillator doesn’t work, it may be because of the following problems:

- Either one of your TL074, 1N4148 diode, C10/C4 cap is dead.

- Some resistances are not where they should have been or their soldering are bad.

Poor audio: If you didn’t do the bias with an oscilloscope, do it. If you did, check that your J201 are not fakes (It can happen). Be careful not to confuse bad audio with the normal distortion that the pedal can cause. Please read the user’s manual to know more.

A last, the J107 transistor must be selected carefully to pair with the fixed values of R12, R26, R27 and R23. The transistor given in the kit has been specially selected. But if you burn it or you didn’t source the components from me, do solder an IC ssocket instead of Q2 to help choose the right transistor.

Hacks!!!

Sure you can also try some other JFETs than J201. But six others fun mods can be done:

- Change the LFO rate. It's possible to do so by replacing C4 and C10 caps by a pair of different values, keeping C4=C10. By dividing the cap values by 2, for example 470nF, you'll multiply the range by 2. From 2Hz to 32Hz for the extended range and C4=C10=470nF. Be careful because above 12Hz, in sine mode the LFO is injected in the audio signal.

  It's also possible to change the range by changing RATE1 pot, R35 and R13.

  R13 gives the maximum frequency. If R13=100 Ohms, Fmax=16Hz. If R13=150 Ohms, Fmax=12Hz.

  R32 and or RATE1 pot, sets the whole range of frequency starting from Fmax. You should know that if R35//RATE1= 5k, Fmin=2Hz. If R35//RATE1= 27k, Fmin=1Hz.

- Less gain: don't put C12.

- Change the square LFO's duty cycle. Resistances R32 and R21 allows to trim the duty cycle. The values are made to have a 50/50 ratio, but by changing R32 or replacing it by a 10k (or more) pot, you may change it from 50/50 to 20/80 or 80/20,

