Solar Trem

Building instructions

v1.0

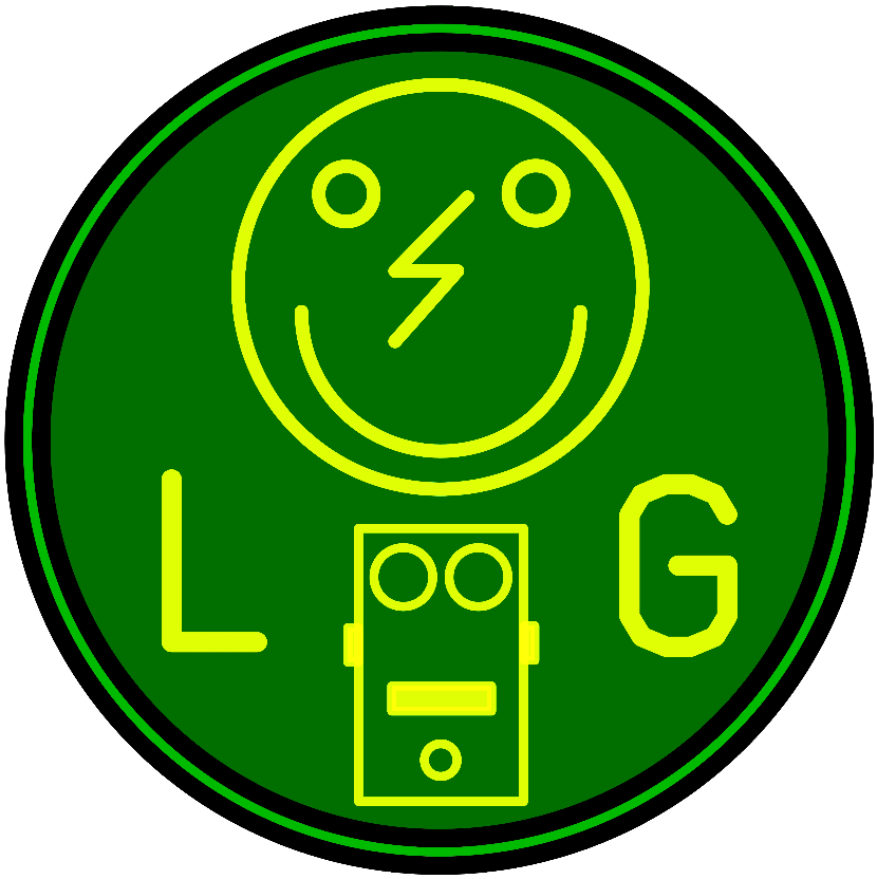
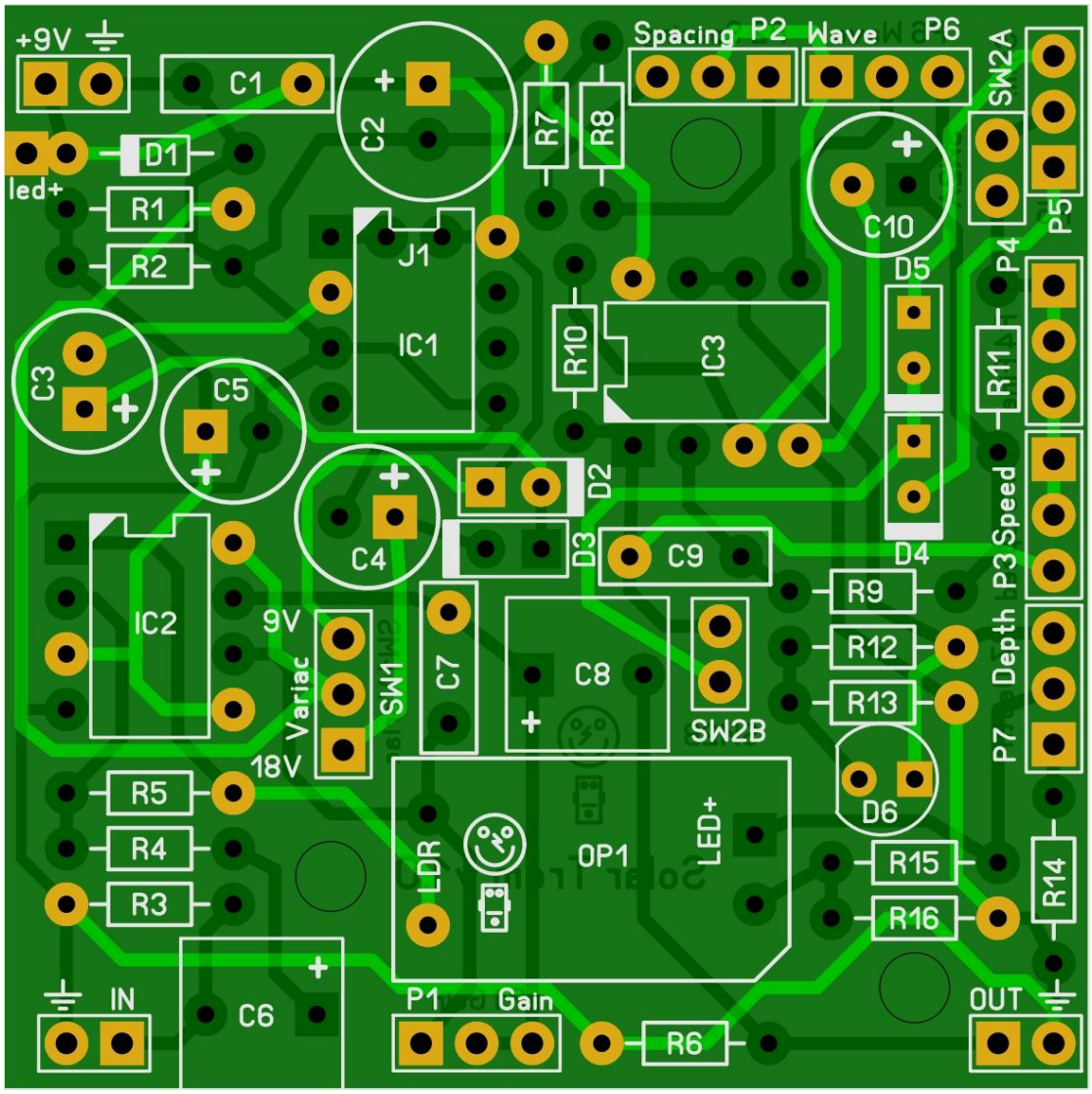


Table of contents

Read this entire manual thoroughly before you start building the effect!

Last update: 06-06-2018

PCB layout



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

Components

Name	Value	Comment	Name	Value	Comment
C1	100n	SMF/MKT/Wima	**P1**	B10k	Gain
C2	100u	Electrolytic 25V+	**P2**	B500k	Spacing
C3	10u	Electrolytic 25V+	**P3**	B100k	Speed
C4	10u	Electrolytic 25V+	**P4**	B5k	Fine
C5	47u	Electrolytic 25V+	**P5**	B500k	Symmetry
C6	1u	SMF/MKT/Wima/Electrolytic	**P6**	B500k	Wave
C7	330p	MLCC	**P7**	B1k	Depth
C8	1u	SMF/MKT/Wima/Electrolytic	**R1**	100k	1% metalfilm
C9	10n	SMF/MKT/Wima	**R2**	100k	1% metalfilm
C10	10u	Electrolytic 25V+	**R3**	1M	1% metalfilm
			R4	220k	1% metalfilm
D1	1N4001		**R5**	220k	1% metalfilm
D2	1N5817		**R6**	1M	1% metalfilm
D3	1N5817		**R7**	220k	1% metalfilm
D4	1N4148		**R8**	470k	1% metalfilm
D5	1N4148		**R9**	220k	1% metalfilm
D6	LED	Speed indicator (diffuse)	**R10**	220k	1% metalfilm
IC1	LT1054		**R11**	1k5	1% metalfilm
IC2	OP2134		**R12**	1k	1% metalfilm
IC3	JRC4558		**R13**	1k	1% metalfilm
OP1	VTL5C1	*See modifications section*	**R14**	330R	1% metalfilm
SW1	SPDT	Variac	**R15**	1k	1% metalfilm
SW2	DPDT	Symmetry	**R16**	1k	1% metalfilm

Build sequence

Before you start, you must decide which mods you are going to build (see modifications section)!

Soldering this board can be very complicated for some people since the solder pads are 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 can result in a bad join that might corrode! Also take extra care not to short components.

Start by soldering the resistors and then the diode **D1**. Next, solder the diodes D4 and D5 in an upright position, solder the IC sockets for IC's and then the MLCC capacitor. Next, solder D2 and D3 in an upright position. Finish by soldering the MKT/SMF/Wima capacitors and then the electrolytic capacitors. See the off board wiring section for details about the potentiometers and how to wire them. Also, D6 is the rate indication LED.

You are almost ready to rock, well… not really. The difficult part starts now.

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

- **1 mono input jack and 1 stereo input jack**.
- **3PDT footswitch** (9 pins)
- **2,1mm DC jack** (isolated) not recommended in positive ground setup.
- **22 gage stranded hook-up wire.**
- **OPTIONAL LED holder and LED (3mm or 5mm depending on your taste) for the effect status.**
- **OPTIONAL Battery clip.**
- **Hammond 1590BB** case (or similar) in your favorite color.

Modifications

The board layout is such that you get the most options as possible. Not all people like that so you can modify the board to scale down the number of options. NB I always mark pad 1 on a switch or potentiometer by using a rectangle pad.

No spacing

If you do not want to incorporate the spacing knob then do not connect **P2** but instead solder pad 1, 2 and 3 together. Also change **R8** to 220k.

No speed fine tuning

If you do not want to incorporate the fine knob then do not connect **P4** but instead solder pad 2 and 3 together. Also change **R11** to 2k7.

No symmetry control

Do not solder **P5**, **D4**, **D5** and **SW2**.

Fixed symmetry control

If you do not want to make the symmetry option switchable then leave out SW2 and short pads **SW2A** and **SW2B**.

Electrolytic caps

C6 and **C8** can be either Electrolytic (polarized) or Film (non polarized). You can choose which ever version you like just note the polarity as marked on the PCB.

No chargepump

Originally the tremolo does not have 18v charge pump and is only powered by +9V. The greater headroom (18V) is used to prevent any unwanted distortion by the tremolo. The effect of the extra headroom is only small so if you want to leave out the charge pump and safe some money, do not install **IC1**, **C3**, **C4, D2, D3** and short pad 1 and 2 on **SW1**.

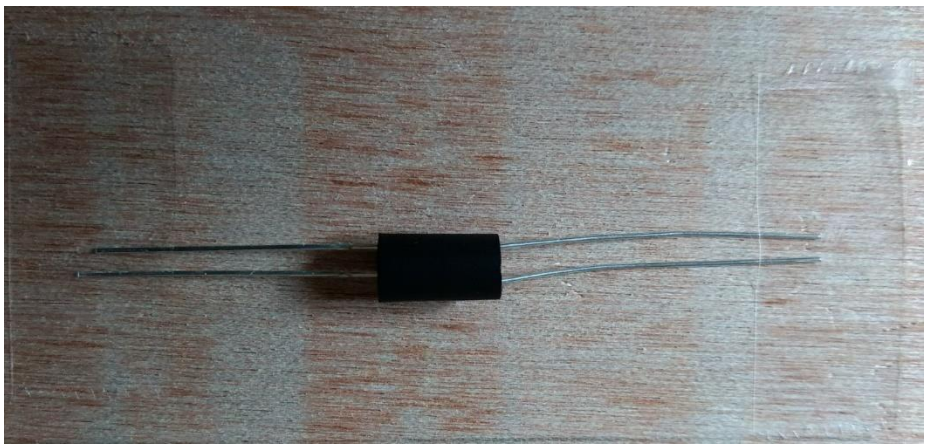
DiY Vactrol for OP1

OP1 is a VTL5C1, which are very expensive. There are cheap ones you can buy on Aliexpress, but you can even keep the costs lower if you build it yourself. You will need a 5mm LED, LDR and some ¼" shrink tube, or even smaller a 3mm LED, LDR and 3/16" shrink tube. I would advise you to use a clear red high bright LED and a cheap 5539 LDR.

1. Position the LED and LDR so they line up good. LED and LDR should touch!

2. Slide over the shrink tube and see to it that the LED and LDR are still aligned.

3. Use some tape to keep the LED, LDR in place and then heat the shrink tube. Watch out that the tape does not catch fire ☺.



4. When ready, make sure that the LED and LDR cannot move in the shrink tube and if so bend the legs.

Op amps

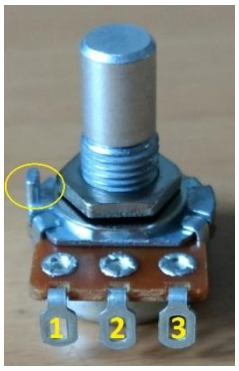
You can experiment with different opamps. Soundwise, **IC2** is the most important one. You can use the OPA2134 here to keep it as clean as possible, but cheaper op amps can also be used with good results like the TL072, TL082 or NJM/JRC4558 just make sure it can handle +18V. **IC3** only functions as the LFO so it does not really affect the sound (besides being the LFO off course). You could try a TL022 as **IC3** as it is less power consuming then the TL072/NJM4558.

Off board wiring

Potentiometers

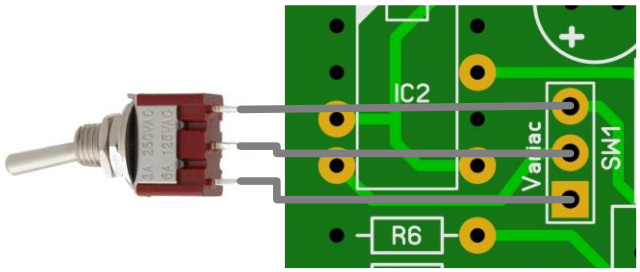
In the picture below you see the correct pin numbering of the potentiometers (P1-P7).
Solder the wires accordingly and it is always a good idea to twist the wires together to create a sort of extra shielding against external noise. The **rectangle pad** on the PCB always marks **pin 1.**

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

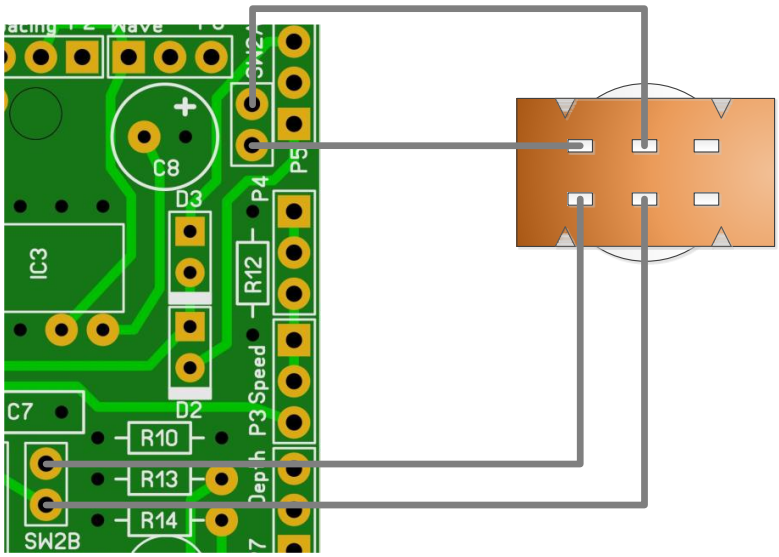


Switches

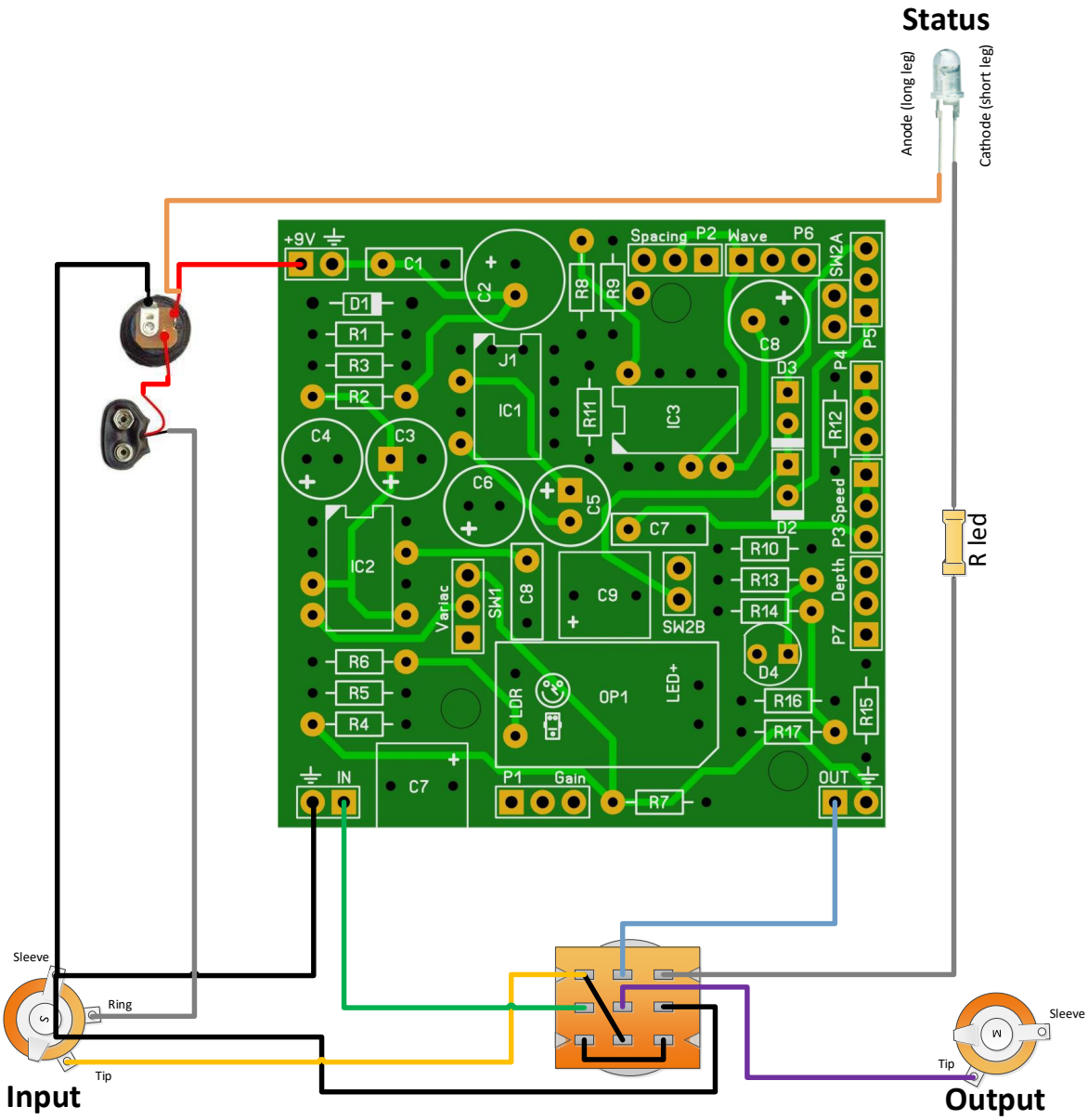
With **SW1** you can switch between voltages for the IC2. Regular +9V or +18v.



SW2 can remove/add the symmetry from/to the circuit and is wired as follows:



Wiring



Note that **R led** is a **4k7** resistors. You can change these value depending on the type of LED you use but 4k7 is safe enough for almost all LEDs @+9V.

The sleeve of the output jack is not connected on purpose. It does however require a good contact between the output jack and enclosure to work. If this is not possible then connect the sleeve of the output jack to the sleeve of the input jack.

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.

Schematic