Purple Vibe

Building instructions

V3.0

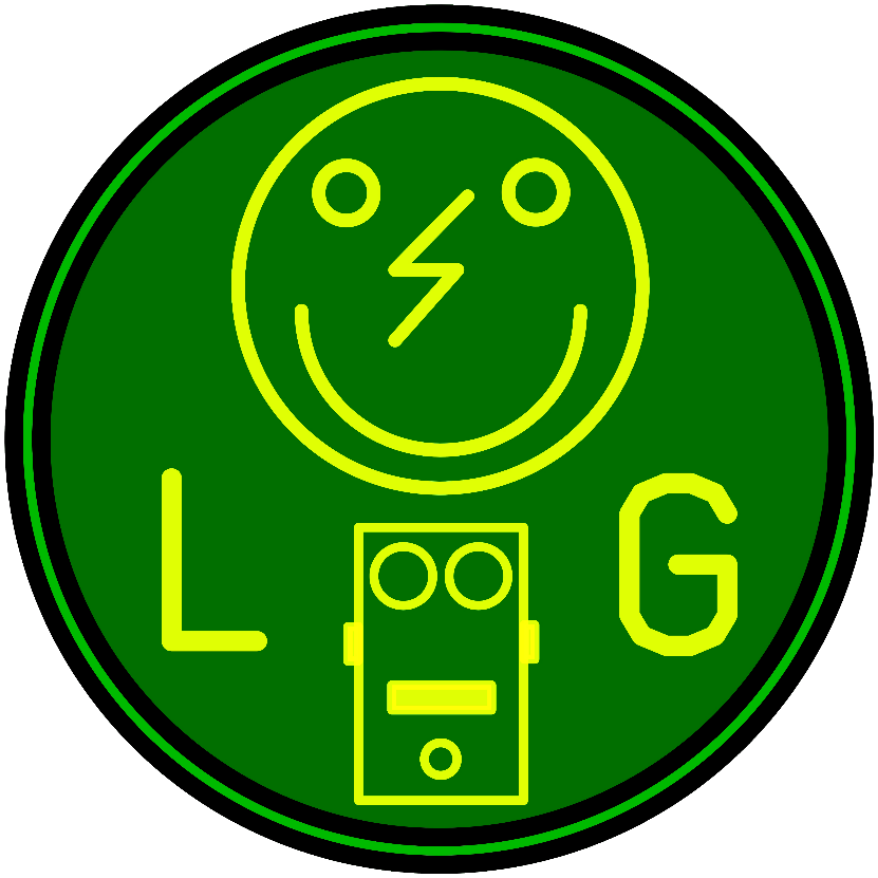
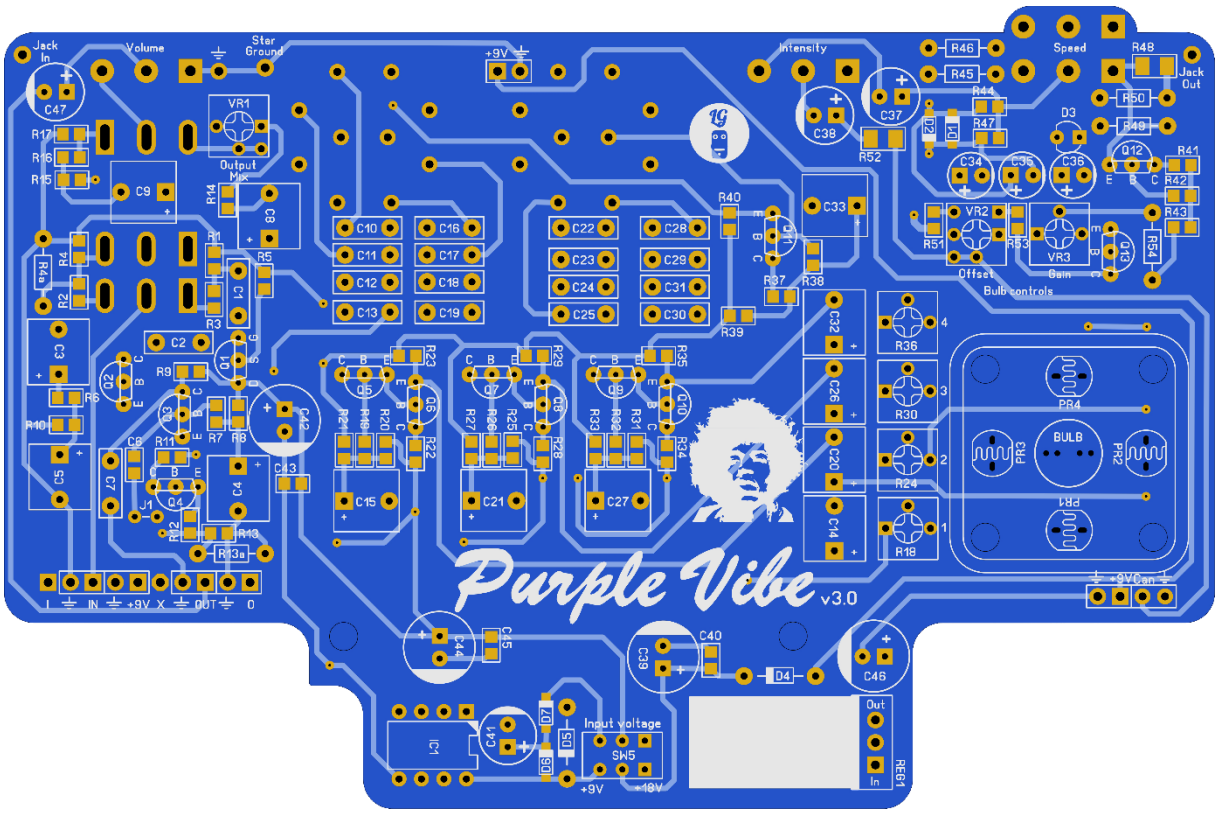


Table of contents

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

Last update: 09-05-2024

PCB layout



Components main PCB

Part	Value	Comment	Part	Value	Comment
C1	100n	SMF/MKT/FKP2	D3	LED	Led 3mm Green
C2	100n	SMF/MKT/FKP2	D4	1N5817	
C3	1u	SMF/MKT/FKP2	D5	1N4742A	Zener 15V 1W (Tayda part)
C4	1u	SMF/MKT/FKP2	IC1	LT1054	
C5	1u	SMF/MKT/FKP2	J1		Optional Jumper for C6
C7	330p	SMF/MKT/FKP2	P1	B100k	Volume (Tayda part)
C8	1u	SMF/MKT/FKP2	P2	C100k dual	Speed (Tayda part)
C9	1u	SMF/MKT/FKP2	P3	B50k	Intensity (Tayda part)
C10	10n	SMF/MKT/FKP2	PR1	GL-5626	GL-55xx/56xx NSL7532/7530 PDV-P9203
C11	4n7	SMF/MKT/FKP2	PR2	GL-5626	GL-55xx/56xx NSL7532/7530 PDV-P9203
C12	15n	SMF/MKT/FKP2	PR3	GL-5626	GL-55xx/56xx NSL7532/7530 PDV-P9203
C13	15n	SMF/MKT/FKP2	PR4	GL-5626	GL-55xx/56xx NSL7532/7530 PDV-P9203
C14	1u	SMF/MKT/FKP2	Q1	MPF102	alternative 2N5457
C15	1u	SMF/MKT/FKP2	Q2	2N5089	
C16	10n	SMF/MKT/FKP2	Q3	2N5088	
C17	3n3	SMF/MKT/FKP2	Q4	2N5088	
C18	100n	SMF/MKT/FKP2	Q5	2N5088	
C19	220n	SMF/MKT/FKP2	Q6	2N5088	
C20	1u	SMF/MKT/FKP2	Q7	2N5088	
C21	1u	SMF/MKT/FKP2	Q8	2N5088	
C22	10n	SMF/MKT/FKP2	Q9	2N5088	
C23	2n2	SMF/MKT/FKP2	Q10	2N5088	
C24	47n	SMF/MKT/FKP2	Q11	2N5088	alternative MPSA13
C25	470p	SMF/MKT/FKP2	Q12	MPSA13	
C26	1u	SMF/MKT/FKP2	Q13	MPSA13	
C27	1u	SMF/MKT/FKP2	R18	B5k	LDR1 offset (Tayda part)
C28	10n	SMF/MKT/FKP2	R24	B5k	LDR2 offset (Tayda part)
C29	1n	SMF/MKT/FKP2	R30	B5k	LDR3 offset (Tayda part)
C30	4n7	SMF/MKT/FKP2	R36	B5k	LDR4 offset (Tayda part)
C31	4n7	SMF/MKT/FKP2	R45	2k7	1% metalfilm 0,6W
C32	1u	SMF/MKT/FKP2	R46	2k7	1% metalfilm 0,6W
C33	1u	SMF/MKT/FKP2	R49	10k	1% metalfilm 0,6W
C34	1u	Electrolytic 25V+	R50	10k	1% metalfilm 0,6W
C35	1u	Electrolytic 25V+	R54	47R	1% metalfilm 0,6W
C36	1u	Electrolytic 25V+	R13a	3k9	1% metalfilm 0,6W
C37	10u	Electrolytic 25V+	R4a	51k	1% metalfilm 0,6W
C38	10u	Electrolytic 25V+	REG1	LM7815	Regulator TO-220
C39	220u	Electrolytic 25V+	SW1	DPDT	Buffer (Tayda part)
C41	10u	Electrolytic 25V+	SW2	SPDT	Mode (Tayda part)
C42	100u	Electrolytic 25V+	SW3	2P4T	Voice 1 (Tayda part)
C44	100u	Electrolytic 25V+	SW4	2P4T	Voice 2 (Tayda part)
C46	220u	Electrolytic 25V+	SW5	DPDT	Input voltage (MSS22D18)
C47	10u	Electrolytic 25V+	VR1	B50k	Output Mix (Tayda part)
			VR2	B250k	Offset (B200k alternative)
Bulb	18v 26mA	Schiefer 8099	VR3	B500R	Gain (Tayda part)

A=Log, B=Lin, C=Rev. Log

Options

Let me start by saluting all previous versions made by the greats like GeoFx, Madbean, the Forum vibe etc. I incorporated most of their mods and added a voice selection which lets you choose between 4 vibe capacitor combinations of the other versions:

1. Original Univibe combination
2. Voodoo vibe combination
3. Resly tone combination
4. Phaser combination

You can even combine them by selecting different caps in voice 1 than in voice 2.

Before you start, you must decide which options you want to incorporate in the build.

1. **Capacitors**
   The original Univibe™ the 1uF caps were electrolytes. Most clones replaced them with SMF (stacked metal film) caps. To give you the ability to keep it as original as possible, I marked the polarity of these caps on the PCB so you can use electrolytes instead of SMF. You can recognize these SMF caps on the PCB by their rectangle marking. If a capacitor is marked as a circle, it is meant to be an electrolyte.
   If you decided not to use SMF 1uF caps then make sure you source your 1uF electrolytes with care! For the electrolytics, buy Nichicon or Elna quality caps to keep the sound quality as best you can.

   Placing jumper J1 will enable **C6** with which will brighten the overall tone. Ju must leave out C7 when enabling C6!

2. **Input buffer**
   The original has no input buffer, but there are a lot of people who have a problem with the volume drop when engaging the effect. To prevent this, there is a switchable input buffer ready on the PCB. If switched off, it bypasses the buffer. However, I altered **R4** to 2M2 to prevent tone sucking as in the original (which has a 47k in R4). If you want it to be as the original, you must populate **R4a** with a **51k** resistor, else leave r4a empty.

3. **Unity mod**
   The original is not quite unity gain. If you hear that difference, you can populate **R13a** with a **3k9**, else leave R13a empty.

4. **Alternative transistors**
   If you can't find a MPF102, use a 2N5457 for Q1.
   Some people advise to use a MPS13 in Q11 instead of the 2N5088 for better phasing.

5. **Lamps and LDRs**
   There are a lot of options for choosing the combination that is right for you. Regarding lamps, you could use a 12V filament lamp or a 12V / 18V bi pin JKL/Schiefer lamp (8097SBP/8099SBP).

   LDR's can be 9203's, GL-5626, GL-5539, GL-5549 or Silonex NLS-7532. Of course if you have personal lamp/LDR preferences, do not hesitate to use them!
   I personally found the combination of a 18V Lamp (8099) and flat GL-5626 LDR's to be a good combination. But there are companies out there on the web that sell great ready to use lamp/LDR kits which you might prefer.

6. **Power supply**
   The PCB is designed to use a regular 9V adapter and raise the voltage to 18V with a built in charge pump when the switch **SW5** is set to 9V. I do not advise the use of a battery in this build so I left it out in the descriptions! But feel free to have a different opinion and incorporate it anyway.
   Some prefer to use a dedicated 18V wall wart and in that case you can switch the internal switch **SW5** to the 18V setting. This will bypass the internal chargepump.

   WARNING! Do ***not*** set SW5 to 9V when using a 18V wallwart. This will overload the circuit and may possibly damage it.

7. **Light shield**
   You can make a light shield out of almost anything, buy my fitting lightshield or 3D print your own. The mounting holes are 19 mm apart. Note that there are traces close to the mounting holes so when using screws please also use plastic rings to prevent the screws from making contact with the traces.

Off board wiring

The Footswitch PCB's are pin compatible with the main PCB.

For the footswitches you will need the following parts:

2x 3PDT footswitch
2x 10uF (C1)
2x 390R (R1)
2x B10k (VR1)
3mm Red LED
3mm Orange LED
2x 3mm LED flange

Setup Lightbulb

Start with these settings:

- **VR1** (mix) at 12 o'clock
- **VR2** (offset) all the way to the right
- **VR3** (gain) at 12 o'clock

Intensity and Volume set to max. Speed 50% mode set to Vibrato.

Turn the Gain (VR3) until the bulblight is moderate and not too bright.

Turn Offset (VR2) until you think it is the best sounding vibe ever.

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