PhotoFet

True Bypass Relay

Building Instructions

v1.0.1

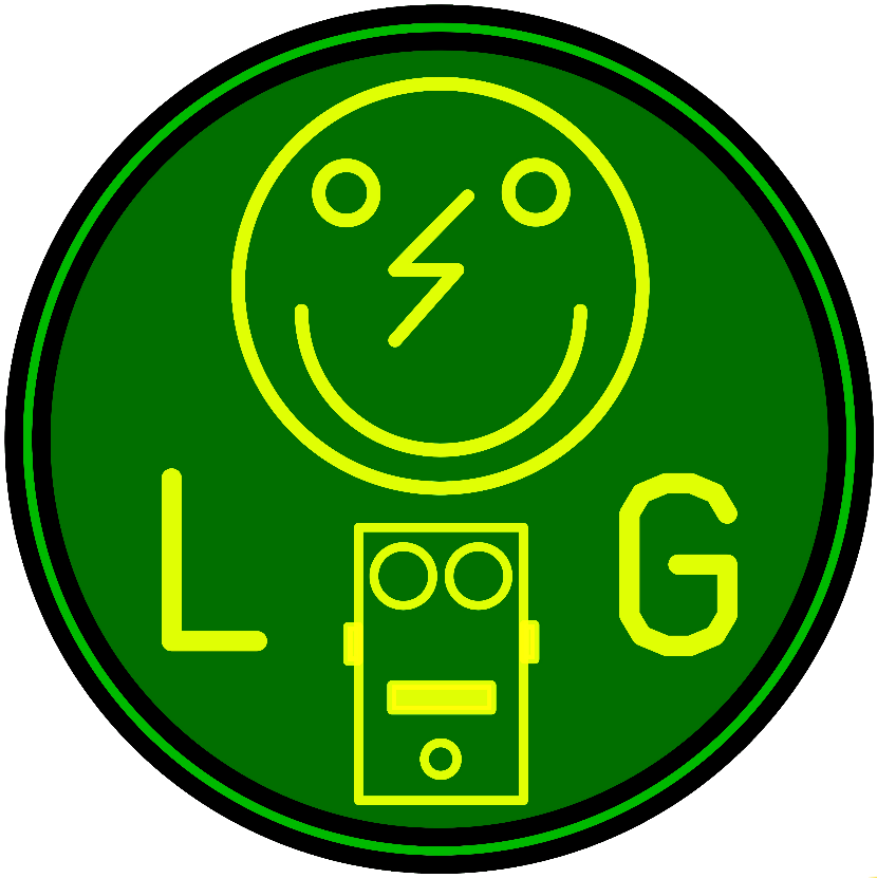


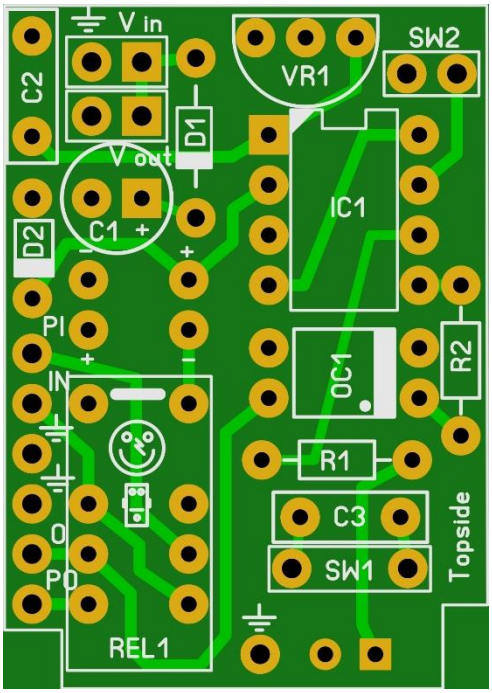
Table of contents

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.

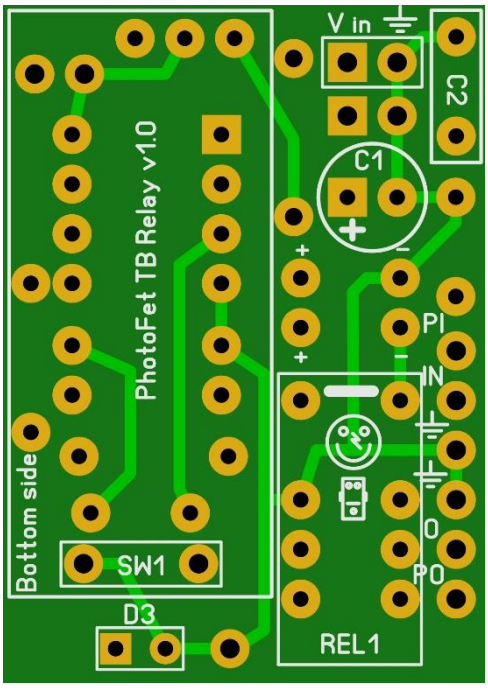
Last update: 26-02-2021

PCB layout

Top-side



Bottom-side



Dimensions: 24,6 mm x 34,7 mm
0.97 inch x 1.37 inch

Components

All parts must be rated 25V+ if used in conjunction with a charge pump

Name	Value	Comment
C1	100u	Electrolytic
C2	100n	MKT, Wima or SMF
C3	100n	Ceramic
D1	1n5817	
D2	1N4148	
D3	LED	Color and type of personal choice
IC1	PIC12F675	
OC1	TLP222A	
R1	4k7	Led resistor
R2	1k5	
REL1	NA5W-K	DPDT relay 5V alternative Omron G6S-2-5VDC
SW1	SPDT	Effect on - off (Momentary)
SW2	SPST	Momentary/Latching mode selection
VR1	LM78L05	

Make sure you also get some not to thick double sided tape

The technique behind the switch

Let's start by emphasizing that the switch is guaranteed 100% true bypass and does not affect your signal. The digital part is only used to switch the relay and light up the LED. The signal of your guitar only goes through the DPDT part of the relay, which is electrically completely separate from the rest of the switch.

It works as follows. When you press the momentary switch (**SW1**) you are sending a signal to the microcontroller which is picked up by the internal program. Depending on the state, it will in turn either put +5V on the relay or remove the +5V from the relay. This way the switch will turn on and off switching the DPDT relay. The state of the switch is saved to memory so when you disconnect it and then reconnect it again, it will return in its last known state.

To prevent the switch from loud "popping", the photocoupler comes in to play. It is turned on just before the relay switches and grounds the output of the effect, making it totally silent. The relay switches, the LED is turned on and then the photocoupler is released, thus removing ground from the output and letting your signal flow through freely again. And all this in less than 60ms. The photocoupler is not in the signal chain when you play and so does not affect your signal chain!

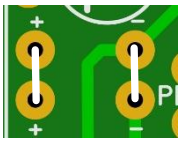
SW2 enables you to make the switch, which is by default latching (like all common effect switches), act as a momentary switch. Depending on the state the switch was in before shorting SW2, it will make the switch act as a "make" or "break" version. This way you can shortly activate an effect or deactivate an effect while you stand on the switch. If you do not want this option, then the only thing you have to do is not connect SW2. Simple! This feature is especially fun to make an effect act like a kill switch. Lastly, the switch supports Longpress momentary mode. Read the "Using the switch" chapter for all features.

Build sequence

Soldering this board can be complicated for some people since the solder pads are very close together and components must be placed on both sides of the board. 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 differ a lot in size depending on their rating.

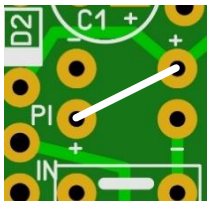
Note: Do not blow on your solder in an attempt to cool it down. That will possibly result in a bad join that might corrode! Watch this video from EEVblog (no that's not me) to learn more.

This PCB is made to fit the relay on both the top-side as well as the bottom side. If you want to make the build as compact as possible then you should place the relay at the bottom side, this is the default layout. This way the build PCB will even fit a 1590A enclosure mounted with double sided tape on top of the SPST momentary switch (**SW1**). In this case you will need to solder jumpers on the **bottom side** like this:

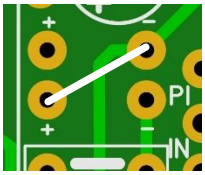


You can also mount the relay on the top-side. For example when you want to place the PCB further away from the SPST momentary switch. In this case, the polarity is switched and you will need to solder the jumpers in a different way. As the jumpers need to cross, you will need to solder jumpers on both the **top- and bottom-side** like this to prevent a short circuit.

Top-side:



Bottom-side:



If you are using isolated wire, you could just solder it top-side and let the wires cross as indicated on the PCB.

Now, start by soldering the resistors and **D1** and **D2** on the *top-side*. You can experiment with the value of **R1**. If the LED is to dim, lower the value up to 1k5 to make it brighter, but do remember that the brighter you set the LED, the shorter it's lifespan and the more power it consumes.

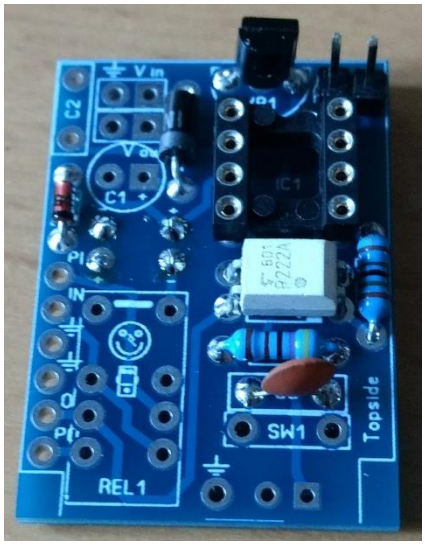
Next, solder **IC1** (pin 1 is marked by the square pad), **OC1** (pin 1 is marked by a dot) and **VR1**, all on the top-side. Make sure you cut the leads as short as possible on the bottom side without breaking any solder points. This is important as we will be sticking the switch to the bottom of the PCB later and we want to save as much space as possible. Solder **C1** and **C2** and **REL1** to the *bottom side*.

Now, something about the photocoupler. **R2** determines the reaction time in relation to the brightness of the LED inside the photocoupler. I tested a lot of batches of TLP222A and they all seem to work fine with a 1k5 resistor at **R2**. However, if you are using a different photocoupler for example a TLP222G, the value can be different. If the photocoupler is configured incorrect, you will hear the loud "pop" again when switching the effect on and off.

Having experimented with several different types and batches, you could try a value for **R2** between 1k5 and 2k2. When it does not pop anymore, you got the that value!

On a side note. The capacitance of a TLP222A (C_{off}) is about 130pF where as a TLP222G is only 30pF. This capacitance is in the off position of the photocoupler **OC1**. I could not really hear a difference, but if you do, I would advise to use a TLP222G.

PS. I would stick to a TLP222A as it is cheap and works great!

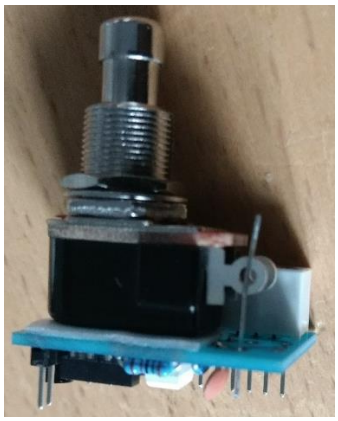


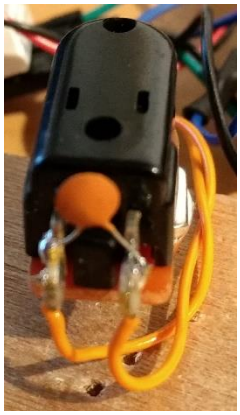
Connecting the switch

Now you've come to a point to make some more decisions. This build is designed to save as much space as possible. To do this you could stick the momentary switch to the bottom of the PCB. If you do not want to do this then you can skip this part and connect the switch in another way as you desire.

1. First off, I advise you to cut a piece of double sided tape to fit the bottom of the momentary switch and stick one side to the switch. The board has been designed to fit the standard type of momentary switch with a short shaft:

2. Now stick the switch to the bottom side of the PCB with the contacts facing the text **SW1**. Be sure that the tape does not cover the pads of **SW1**.
3. Then use a 100nF capacitor (this is actually **C3**) with long enough leads to solder the contacts to the **SW1** pads. If the leads of the capacitor are not long enough (or you do not want to solder the switch directly to the PCB) then solder some spare leads directly to the switch using the **SW1** pads and solder the 100n using the pads of **C3**.





Optional SW2

If you install **SW2**, you can make the switch act as a latching switch (default) or as a momentary switch (when SW2 is shorted). You can use a simple SPST or SPDT switch and connect it as mentioned in the off board wiring section further on in this document. If you do not want to use **SW2** then you can just simply leave it out and the switch will only act like a normal effects on-off switch (latching).

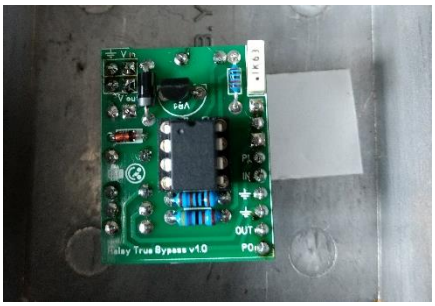
Lining up the LED

We are almost ready. Now it is time to decide how to connect the LED (**D3**) and where to drill a hole in your enclosure. I used and old enclosure I had lying around (BYOC, how fitting) so it does not really represent how tight you can build this! I will describe the procedure of fitting the led directly on the PCB, but you can always decide to do it differently eg. by using a Led flange.

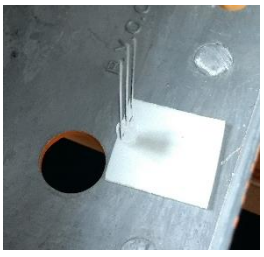
1. Start by drilling the hole for your switch (approx. 15 mm from the bottom, but measure and decide before you start drilling!!).

2. Stick some double sided tape to the right of the hole and remove the sticker revealing the sticky side.

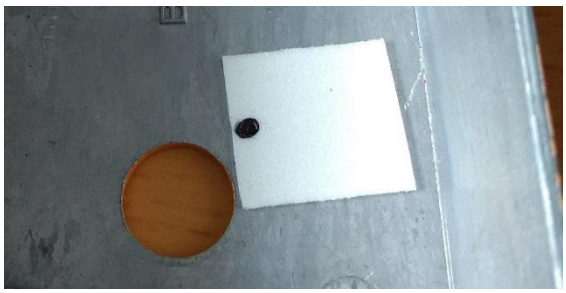
3. Insert the LED in the board (bottom side!) , but do not solder it! Orientation of the LED is not yet important as we are only going to use it as a placeholder for measurement.
4. Insert the PCB in the enclosure while holding the LED so it does not fall out. Place it the way you fit right and press the led so it sticks to the double sided tape.



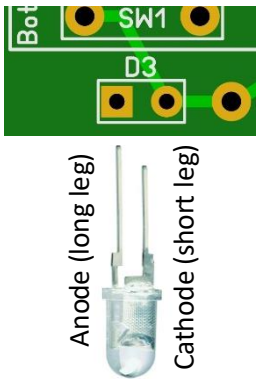
5. Now gently pull the board with switch out, making sure the LED sticks to the tape.



6. Press down on the led making sure not to damage it, but still hard enough to leave a mark in the tape. Remove the LED and mark that dent with a sharpie.



7. Now drill the hole. Be extra careful as you will be drilling from the inside out instead of the other way around.
8. Finish by reinserting the PCB and LED **D3** (still <u>not</u> yet soldered to the PCB). Make sure this time that you have oriented the LED correctly.



9. Make sure you screw the switch tightly to the enclosure and if all fits well, you can solder the LED to the PCB. I advise you to uninstall the PCB/switch for the off board wiring.

PS. If you want to save some extra space, you could turn everything 90 degrees. All steps still remain the same, but only 90 degreed different. Note that I was too lazy to remove the tape before making the picture.

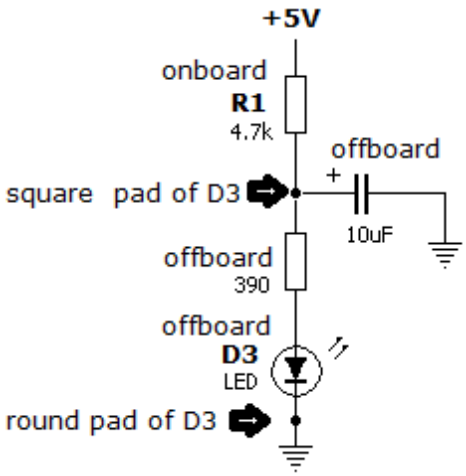
The finished switch

When you are ready, the switch should look something like this. Note that I used male headers. This is not a good idea when you use it in your build as it will take up a lot of space. I chose to use headers as I use the switch to test all my new boards and made it modular this way. Saves me a lot of time when testing.

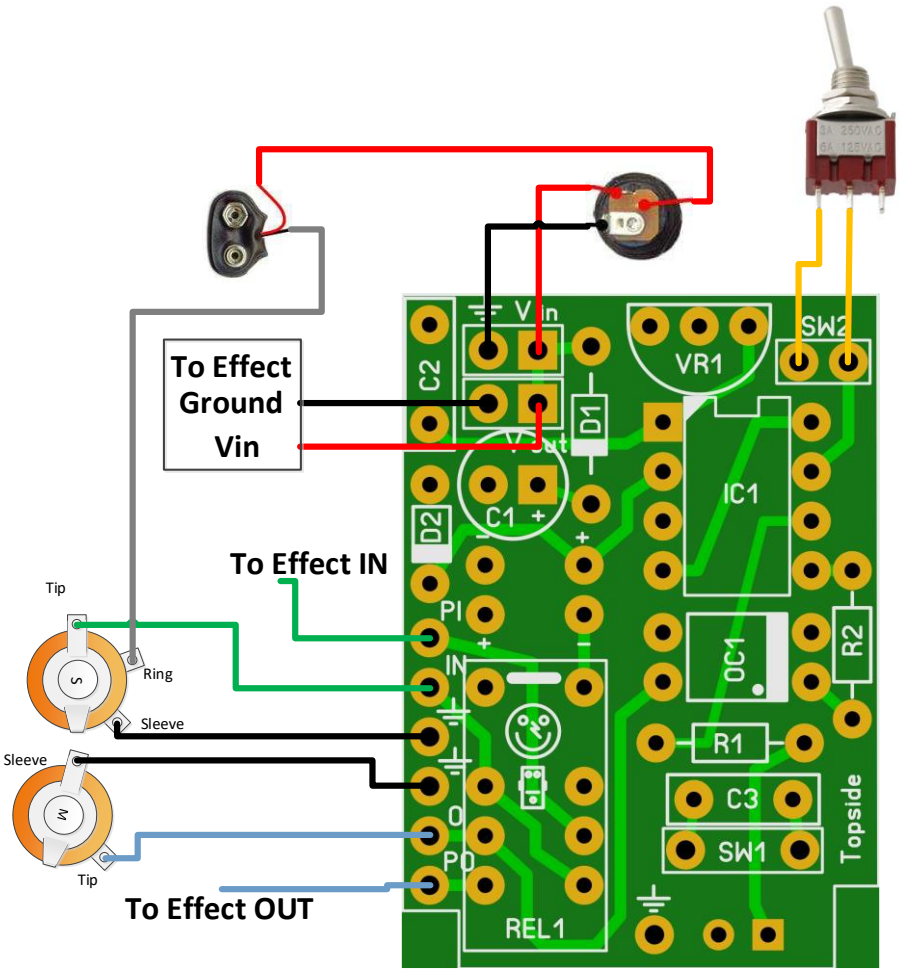
Residual noise (LED popping)

Although the photofet prevents the effect from loud popping, there can still be some light pop due to the LED turning on and off. One idea is to extend the time that the effect is turned off, but that could lead to a feeling that the switch is acting sluggish. Another way to solve this is to use the AMZ trick to prevent led popping.

In stead of directly adding D3 to the PCB, you connect a wire to the square pad of D3 on the PCB and solder the other side of the wire to the + pin of a 10uF cap and a 390R resistor. The – of the 10uF cap goes to ground and the other side of the resistor goes to the long leg of the LED. The short leg of the LED can be connected to another wire and that wire can be connected to the round pad of D3 on the PCB. Schematic:



Off board wiring



As you might have noticed in the diagram: Green means input and Blue means output.
Note: This board works on either +**9V or +18V input**. Do not use any adapter rated over 18V!

Modes: Using the switch

The switch can work either as a (default) latching ON-OFF switch (like all ordinary effects switches) or as a momentary switch (either “make” or “break” depending on your order).

When you keep the switch pressed for 2 seconds or longer, it will temporarily switch to momentary mode.

When you short **SW2** the switch will go in to default momentary mode using the last know position as default. If the switch was in on mode when you shorted SW2, the switch is default on and pressing the switch will turn it off until you release the switch (break mode). If the effect was off before the shorted **SW2**, the switch will turn on when you press it and turn off when you release it (make mode).

	Effect is On	**Effect is Off**	**After release**
SW1 Pressed < 2 sec	Latching Off	Latching On	
SW1 Pressed > 2 sec	Momentary Off	Momentary On	back to latching
SW2 is shorted	Momentary Off	Momentary On	stays momentary

So when **SW2** is left out or off, you are using a standard latching switch, but with the option of using it temporarily as momentary switch when pressing **SW1** longer than 2 seconds. If **SW2** is on you are using a standard momentary switch and pressing **SW1** will have no effect.

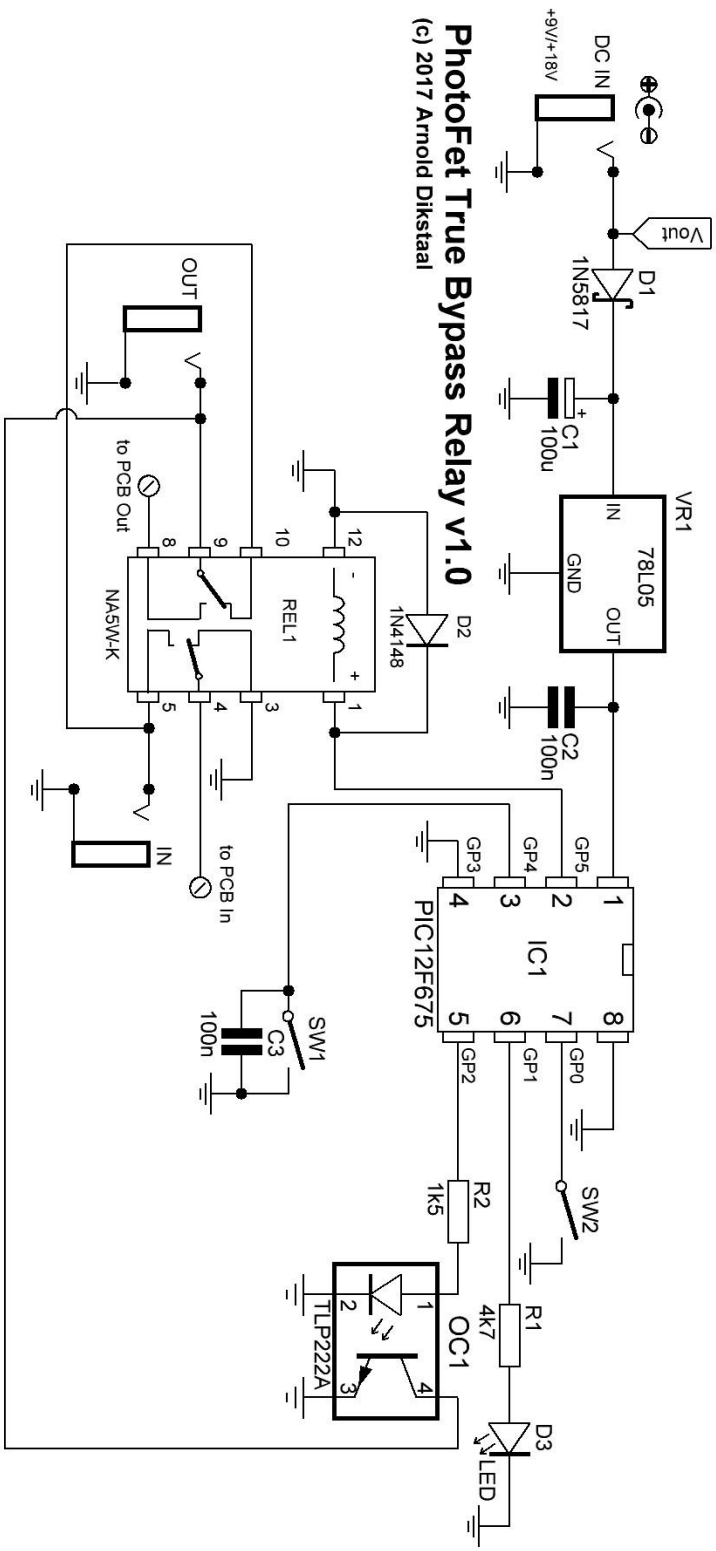
Troubleshooting

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 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’ve 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 joint 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


PhotoFet True Bypass Relay v1.0
(c) 2017 Arnold Dikstaal
DC IN
+9V/+18V
Vout
D1
1N5817
C1
100u
VR1
78L05
IN
GND
OUT
C2
100n
OUT
to PCB Out
NA5W-K
REL1
D2
1N4148
IN
to PCB In
GP3
GP4
GP5
PIC12F675
IC1
GP2
GP1
GP0
C3
100n
SW1
R2
1k5
SW2
TLP222A
OC1
R1
4k7
D3
LED