OmniMuff

Building instructions

V1.0

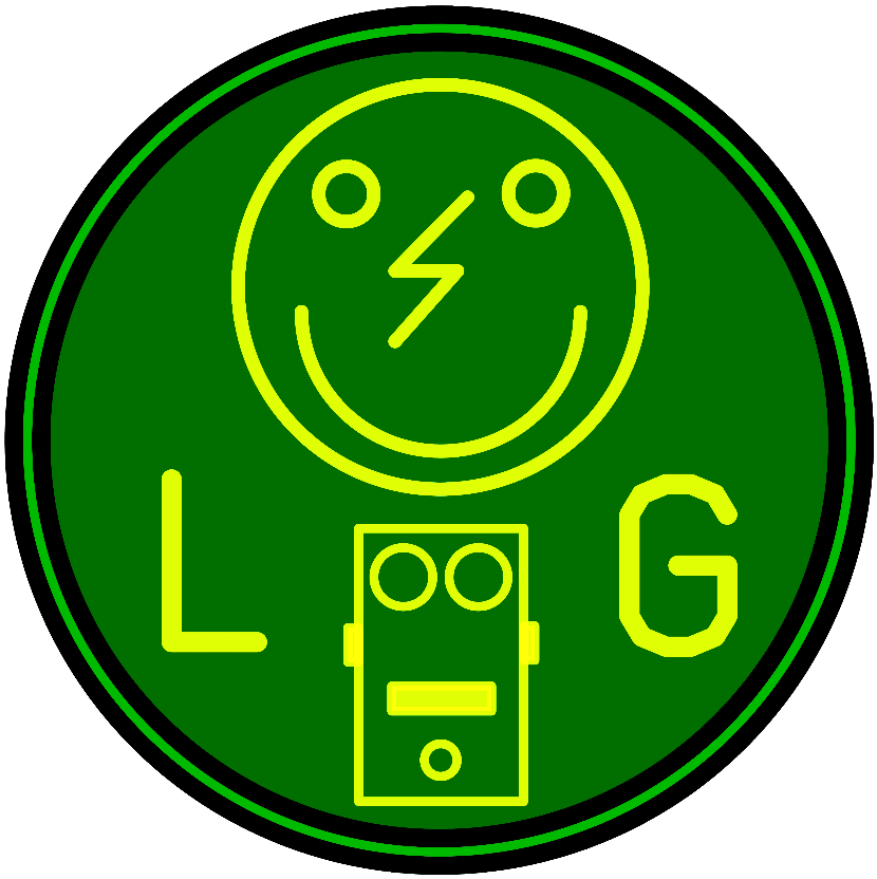


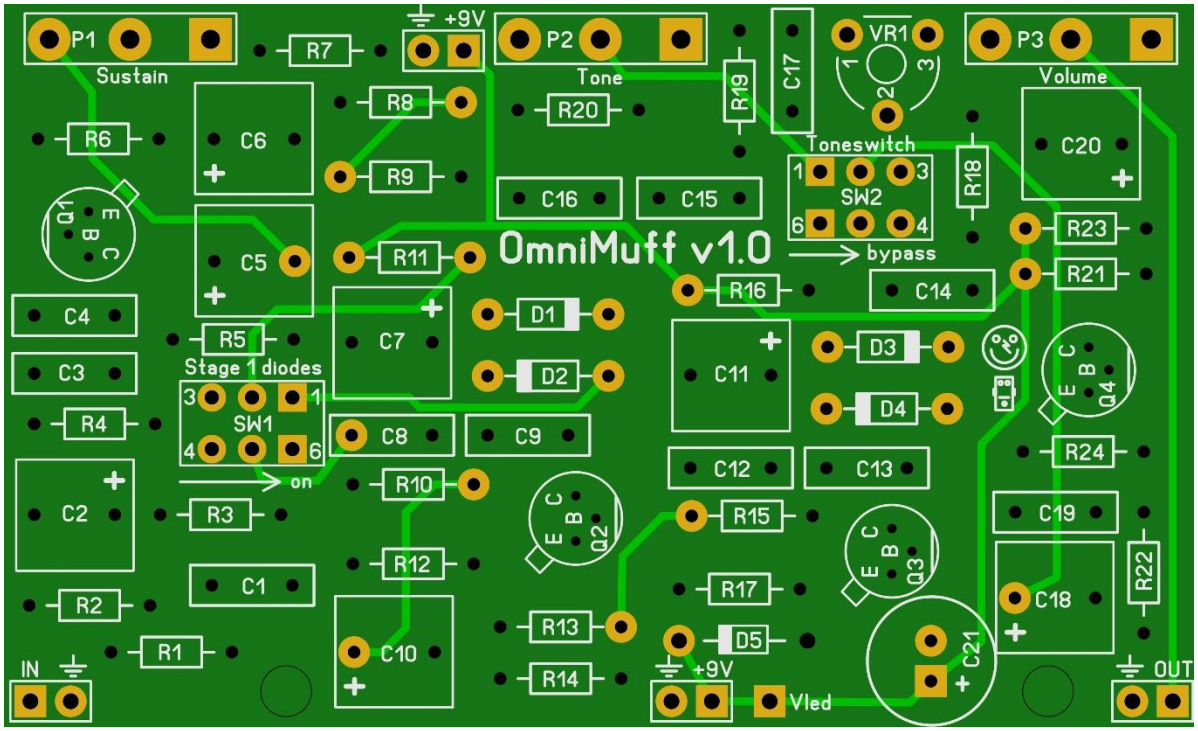
Table of contents

Read this entire manual thoroughly before you start building the effect! There are some available options and a lot of different configurations, so you should choose which one you want to incorporate before starting your build.

Last update: 23-02-2019

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

PCB layout



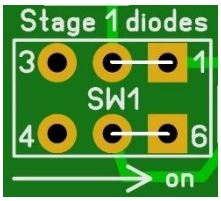
Dimensions: **75 mm x 45,5 mm**
2.95 inch x 1.79 inch

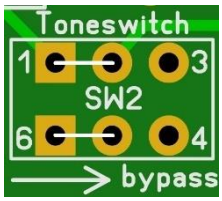
Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Build options

This PCB can be used to mimic an extreme wide range of vintage to modern Big Muffs. In total 29 versions of the original Version 1, 2, 3, 6, 7, 8 and 9 as well as some famous clones (EQD, Colorsound etc.) can be built with just this single board. OMG, the stress of choosing which one to build…. Buy more!!

In the components section you will find the components needed for the different configurations. When it says “jump”, there is no component to be place but instead you’ll need to connect both pad of the component with a jumper wire (eg a spare piece of lead wire you get when cutting a resistor to size). When it says “jump 1-2”, then you’ll need to connect pad 1 to pad 2 of the component with a jumper wire. With “jump 5-6” you’ll need to connect pad 5 to pad 6 of the component with a jumper wire, etc. A dash (“-“) means there is no component nor jumper to be placed at that spot for that type of Big Muff. Just a few examples for switch 1 and 2:





The PCB was designed to use PCB mounted potentiometers (**P1**-**P3**) on the back of the PCB, but you are free to use more traditional solder lug chassis potentiometers. Rectangular pad on the PCB marks pin3 and that corresponds with this numbering of the solder lugs:

VR1 is arranged as onboard trimpot on the PCB. This is done because for most people it is set and forget. However, you could choose to make it external by wiring a chassis pot. For this I added the pinout next to the trimpot pads so you know what pad to connect to which pin on the pot.

Because the different Big Muff versions used a lot of different transistors, I marked the pinout on the PCB (EBC/CBE). Before you place a transistor, make sure the pins correspond to the letters on the PCB. I cannot stress this enough! Also note that 72#2 version 1 Triangle is a **PNP** effect that means that it needs **-9V** (positive ground). You can do this by simply reversing the black and red wire of the DC, but that also means you can not use it with a daisy chained power supply with other +9V effects or things will go really bad and I predict smoke will come out somewhere! I suggest you leave out the DC Jack in **PNP** versions of the Big Muff and only use a reverse wired battery. This way it can be used in a chain of +9V effects.
SW1 is added so you can switch the Colorsound Tonebender stage 1 diodes and caps in and out of the circuit instead of just leaving out the components. That way it is more versatile. This option is not mentioned in the component section as it is not part of the original!

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Components

Since the OmniMuff can be used to build so many different types of Big Muffs, we split off the components section in to a separate configuration document with all possible configurations.

My advice: Capacitors with values **< 1nF** should be **MLCC or Ceramic**, values of **1nF – 1µF** should be **SMF/MKT/Wima** and all values **> 1µF** are **Electrolytic**.

Then there is always the magic question:
"***Will it really sound like a <your favorite Big Muff version>?***"
Well, that depends on the quality of the parts you are using. I will not go into a rant about parts (vintage or not) but I will give you my personal view based on building all the configurations and comparing some of them to the original.

Transistors

First, you could buy the real vintage ones. You will have a very hard time finding genuine vintage FS36999s/2N5133 and it will cost you a lot. Alternatively you can consider buying a (modern) alternative like the 2N3904, 2N5088, 2N5089, BC549C, BC550, BC239, SE4010, 2N5210 etc. Either way you should consider to socket the transistors and try more than one type and even try to mix. Experimenting is fun! PS 2N5089 and 2N5210 are very close for the Russian Big Muffs.

It is reported that the original vintage FS36999 has a H_{fe} between 160 and 200.

Resistors and capacitors

Ok, this is going to get me some hate mail. I found no (real) tonal difference using vintage carbon composition resistors or modern metal film resistors. I am NOT a certified electrical engineer so this is based on simply experimenting.

Ditto goes for the capacitors. I used mostly WIMA, SMF and electrolytic capacitors from Panasonic™ and they sound great everywhere. I know that there is a lot of discussion about this on the internet so feel free to totally disagree with me.

None the less the value and tolerance of these components influence the sound greatly. Components have a spread in their values. Every part is sold with these values inside a certain tolerance. Carbon Composite are +/- 5% and while metal film mostly are +/- 1%. Capacitors are even worse. Generally the tolerance is about +/- 10% but can also be +/- 20% or even +/- 5%. It depends. This is (only) one of the reasons that no 2 effects of the same type really sound the same. To conclude: sound is more influenced by the tolerance (and change of that over the years) of the components then the brand.

Recreating an effect should also mean that the exact value of the components should be measured and not just their advertised value. An old 20% 4.7µF will range between 3.76 µF and 5.64 µF. This can and will have a lot of influence in a RC filter!

To conclude, there is also the influence of temperature. Every component will measure different under different temperatures. So, at -20°C an effect might sound different then at +40°C.

But if you really want to read up on the different Big Muff versions and components, you should go to the Big Muff page that Kit Rae made. It is a true gem!
http://www.bigmuffpage.com/

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Build sequence

Soldering this board can be very complicated for some people since the solder pads are very 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!

Start by soldering the jumpers where needed. Next, solder the resistors and diodes.

If you want to experiment with other transistors then you could socket them instead of soldering them to the board. You'll need a some 20 SIL sockets, break off the sockets and solder them to the board. Now is the time to solder these sockets on the PCB. Place the transistors only once you are finished with all soldering and off board wiring!

Note: Orientation of the transistors vary for each different type you use. For this reason I marked the pinout on the board for your convenience. I also added a graphic line to indicate the flat side of the transistor when using a TO-92. This is based on the 2N5088 transistors. Other types possibly need to be rotated! Always consult the datasheet of the transistor and orient accordingly.

Now continue by soldering the small capacitors (MLCC) then the small SMF, trimpot and then the Electrolytic capacitors. There are some configurations with hard to find values capacitors. If you cannot find 500pF for **C3**, **C8** and **C12**, you can use 1n in series to get 500pF. So 1n in both **C3** and **C4**, **C12** and **C13**, **C8** and **C9**. For the 50n in **C7** and **C11** you can just use 47n.

I suggest you now drill the holes in your enclosure so you can use it during the off board wiring. Try to keep wires as short as possible.

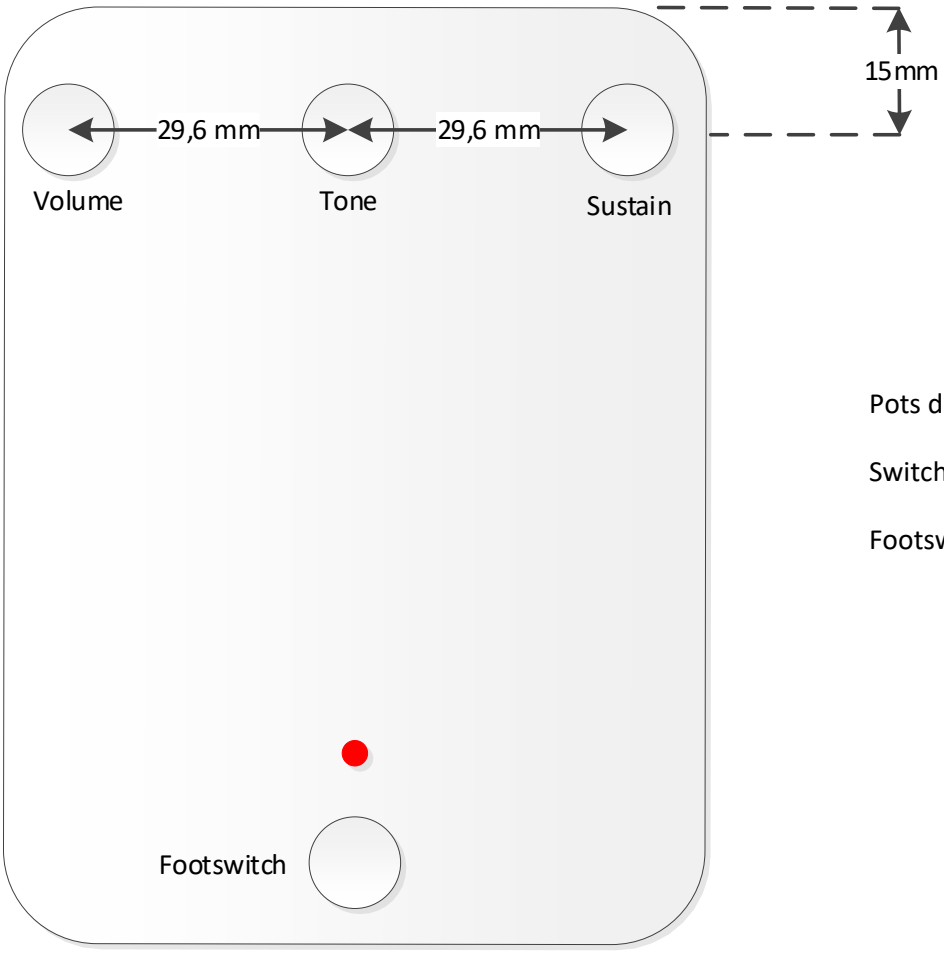
Note: Really take some time to determine where to place the pots, switches, jacks and PCB in the enclosure before you start drilling. Measure twice, drill once.

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:

- **2 input jacks**. 2 mono jacks if you are not going to use a battery but only the 9V adapter. 1 mono (for output) and 1 stereo jack (for input) if you will be using both a 9V battery and the 9V adapter.
- **1 x 3PDT footswitch** (9 pins)
- **2,1mm DC jack** (isolated) center negative.
- **9v battery clip** (optional).
- **22 gage stranded hook-up wire.**
- **A LED holder and LED.** This enables you to mount the LED in the enclosure.
- **Hammond 1590BB enclosure**. Painted in your favorite color.

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Drill template



Pots drill holes 7mm

Switch drill holes 6mm

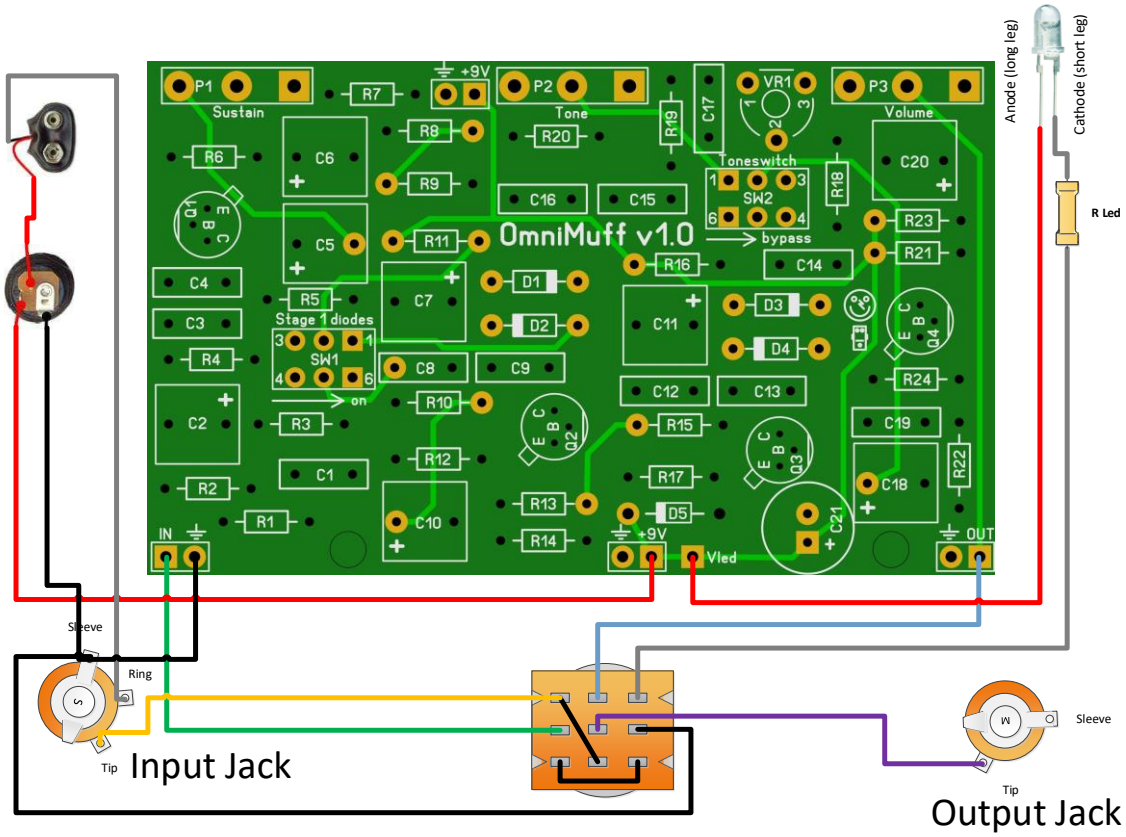
Footswitch drill hole 13 mm

After you drilled the enclosure you can insert the potentiometers and attach them firmly to the enclosure. Now insert the PCB on the pots and if all fits ok you can solder the potentiometers to the PCB.

If you want to incorporate **SW1** and **SW2** as an external switch, you can place them wherever you want. Just measure before you drill!

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Off board wiring



Note that **R led** is a **3k3/4k7** resistors. You can change this value depending on the type of LED you use but 3k3 is safe enough for almost all LEDs @ 9V.

The sleeve on the output jack is not connected on purpose. Make sure the output jack is in good electronic contact with the enclosure else you can try and connect the sleeves of the input and output together. If you test the unit outside of an enclosure you need to connect the output sleeve to ground!

Note that there are two +9V pads on the board. **You may only use 1 (!!)** at a time else you might short circuit the board. In the picture above I used the bottom one, but if you are using a top mounted DC jack, it might be more convenient for you to use the top one on the PCB.

It is now time to place your transistors in the sockets if needed. Connect everything, build it in your enclosure and enjoy your effect!

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Modifications

There are so much tweaks made to the Big Muff, it is almost impossible to write them all down. This PCB by itself is actually one big modification of the Big Muff. But… have a look at the great work Coda Effects did by gathering a lot of tweaks:

https://www.coda-effects.com/2015/11/big-muff-mods-and-tweaks.html

And if you google “Big muff mod” you will also find a ton of mods.

Troubleshooting

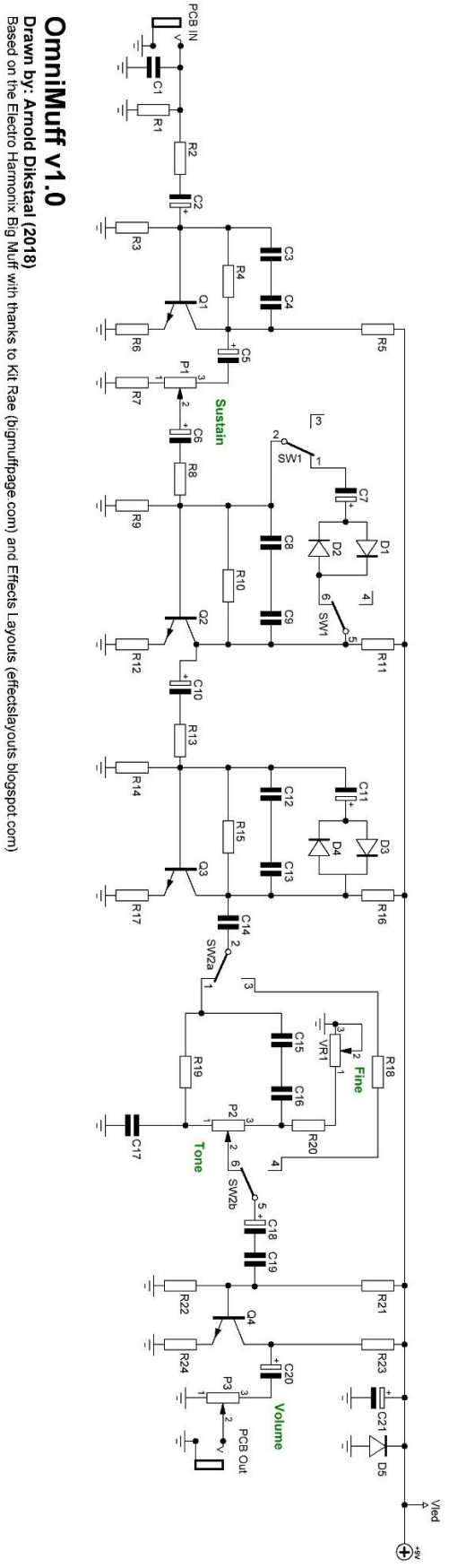
All PCB’s have been 100% factory e-tested and out of every batch I receive I build an effect to double check, so there should not be a connection problem on the PCB itself.

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.

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Schematic



Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.