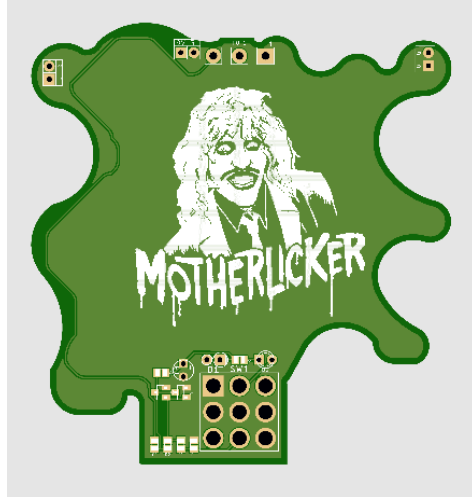


MOTHERLICKER



Thank you for purchasing your PCB project from Noise Therapy FX. Here are your build documents and basic instructions. It is recommended that you read thoroughly before beginning your project. Included here are several key bits of information:

- A Bill of Materials by part # for ease of population.
- A Bill of Materials by value for ease of ordering.
- A PCB dimensional drawing to help you with enclosure layout and control drilling.
- A schematic drawing to help with diagnosis.

These instructions are designed with the expectation that you have some basic understanding of schematic reading, soldering skill, and such. Though these projects are designed in such a way as to be accessible to builders of at any level of experience. I hope you enjoy your project and if you do have any questions do not hesitate to reach out to us at Noisetherapyfx@gmail.com

Basic Design Philosophy

All PCBs are laid out according to quality best practices for ease of population, efficiency, and electronic integrity.

Unless otherwise specified, all PCBs are designed to be installed in a 125B sized enclosure with top-mounted jacks. I have also included pads on the lower edge of the board for use of side mounted jacks if you prefer. I recommend using my footswitch daughterboard with 6pin ribbon cable for ease of connection, but you can easily wire how you choose based on your particular need.

All trademarks are the property of the respective owners. The company and product names used in this website are for identification purposes only. Copyright 2020, Noisetherapyfx.com

MOTHERLICKER

All PCBs are designed as close to original specs as possible. Some exceptions will be using modern conventions with power protection and my simplified offboard wiring layouts. If a component is not readily available, I have tried to either provide a common replacement or, as in the case of JFETS, included both through hole and SMD support right on the board. No need for adaptors or to pay outrageous prices for obsolete parts.

Component Conventions

- Resistors. 1/4W 7.5mm lead spacing.
- Capacitors <100p. 2.5mm lead spaced MLCC or Ceramic.
- Capacitors 100p-100n. 5mm lead spaced, 2.5mm wide film or MLCC
- Capacitors 120n-1u 5mm lead spaced, 4.5mm wide film
- Capacitors 2u2-47u. Electrolytic 2.5mm lead spaced 5mm.
- Capacitors 100u. Electrolytic 2.5mm lead spaced 6.3mm.
- Potentiometers. 16mm PCB mount.
- Diodes. 7.5 mm lead spaced. Clipping LEDS are 3mm.
- All Op-Amps are spaced to use sockets
- All transistors are TO-92 packages

Assembly Conventions

While you can assemble your circuit however you like, there are some general guidelines that can help make assembly a bit easier.

First and foremost, populate your board one component type at a time based on size. Recommended order:

1. Diodes and Resistors
2. Ceramic and film caps
3. Sockets and Transistors
4. Electrolytic caps
5. Offboard wiring

MOTHERLICKER

Always double check direction of polarized components and multipin devices. Diodes, LEDs, electrolytic capacitors, transistors, and ICs can only be installed in one direction to properly function. The square pads for LEDs, and polarized electrolytic caps correspond to the long lead of the device. The silkscreen is your key for diodes, transistors, and ICs.

Use sockets for ICs and any components you think you might want to experiment with. I've taken great care to leave plenty of space for flexibility.

Never install ICs until you are ready to test your device as they tend to be heat sensitive and can be damaged from the heat of a soldering iron.

It is a good idea to test fit your pots and switches to your enclosure before soldering to your PCB. This will help ensure the correct height and fit of each device and make assembly simpler. Tape your components to your pcb and then install in your drilled enclosure. Install the nuts to lock the components in place on the enclosure, then once satisfied with the fit, solder the components to the PCB and remove the tape.

Motherlicker Details

Controls Layout:

- Mangina: Controls overall level of circuit.

Motherlicker is an ode to a classic Effector 13 circuit, the Silver Crank. It does interesting things on its own and even more quirky fuzzy things when stacked or driven, Have fun!

Due to the fact that the footswitch is direct mounted on the PCB it will be easier to fit into your enclosure if you mount the controls to the enclosure first and then solder the pins to the PCB. Otherwise it may be more difficult to fit into the enclosure due to the different heights of the potentiometer and footswitch.

MOTHERLICKER

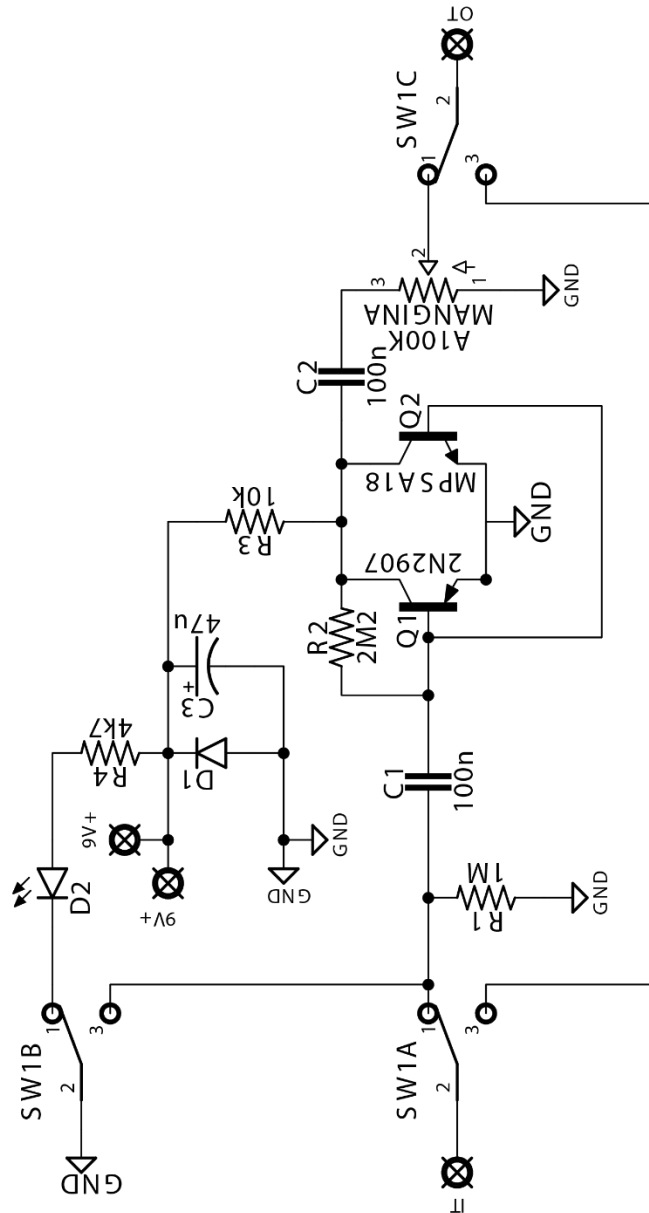
Bill of Materials

MotherLicker BOM	
Part	Value
C1	100n
C2	100n
C3	47u
D1	1n4001
R1	1M
R2	2M2
R3	10k
R4	4k7
Q1	2n2907
Q2	MPSA18
MANGINA	A100k

Parts by QTY		
Type	Qty	Value
Resistors	1	4k7
	1	10k
	1	1M
	1	2M2
Capacitors	2	100n
	1	47u
Transistors	1	2N2907
	1	MPSA18
16mm	1	A100k
Diodes	1	1n4001

MOTHERLICKER

Schematic



MOTHERLICKER

