

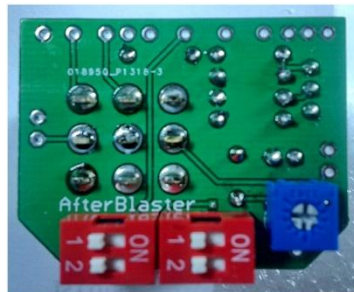
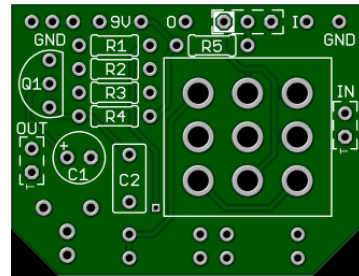
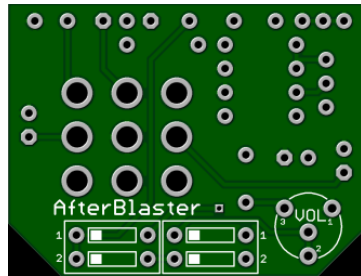
GuitarPCB.com

AfterBlaster - Build Instructions

The AfterBlaster is a cool THREE-IN-ONE circuit that provides several useful functions. It is a handy bypass footswitch wiring board with a status LED. It has a switchable boost circuit that provides a clean boost enhancement for any circuit. It can also provide a final tone balancer to attenuate the output level of any circuit. Or anything in between!

Add an extra gain stage to any circuit and more.

The DIP switches and trimmer must be installed on the opposite side of the PC board from the other components and footswitch. See the views below. The left view should be facing up when viewing the gut shot of the pedal. When you wish to turn the trimmer up or down as well as turn the boost off, simply remove the back panel.



PARTS LIST

Part	Value	Part	Value
C1	22u	R3	1k
C2	220n	R4	2k4*
D1	BiColor CA LED	R5	1k8***
Q1	MPF102**	SW1	3PDT_FS_TB
R1	33k	DIP1-DIP2	2-POLE_DIP
R2	1M	TR1-Vol	A100k

Build Notes:

Please use the photos above as a guide to properly install all components and switches!

* **Socket R4** – See build notes below for details. 2.4k is an average value to get MPF102 or J113 close to the wanted 5v Drain reading.

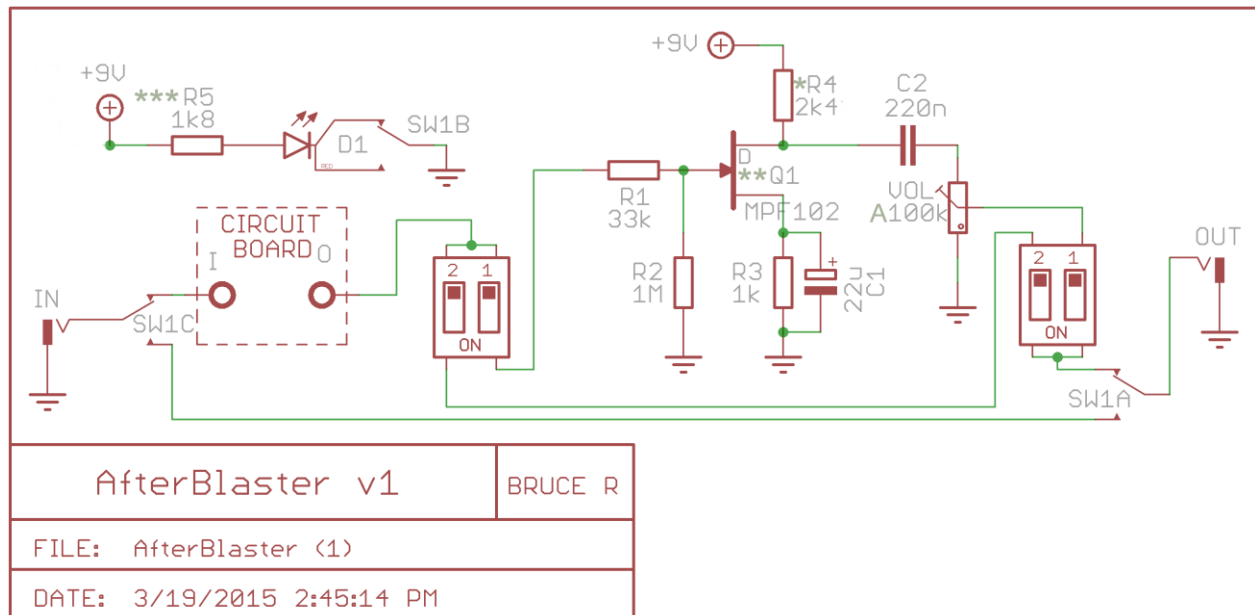
** **Q1** may also be a J113 (MPF102 replacement) or changed to a 2N5457 for more grit, however 2N5457 will require an adjustment to the value of **R4**. Beware of “Fake” MPF102 or 2N2457 transistors from China or eBay! I do carry authentic **J113’s** in my [PCB Shop](#).

*** **R5** is the CLR (current limiting resistor. Adjust this value to suit for brightness.

Dip Switch positions

- To activate the AfterBlaster both Number 1 Switches should be in the **ON** position, and both Number 2 switches should be in the **OFF** (or facing Number 2) position.
- To bypass the AfterBlaster both Number 1 switches should be in the **OFF** (or facing the Number 1) position and the Number 2 switches should be in the **ON** position leaving you with the stock circuit.
- The Dip switches will not be functional in any other configuration. Either set both on 1 or both on 2.

SCHEMATIC



Additional Build Notes:

While 2.4k will give you desirable results, if you wish to fine tune the Bias resistor use a socket on R4 then measure the Drain voltage of Q1 and adjust the resistor to achieve the exact results you like. Remember every MPF102 is going to be different so 2.4k is a working approximation. The preferred voltage range for the Drain of Q1 to ground is between 4.5V and 5.0V. Once you find the perfect resistance you can solder it to your socket. (Again this is an optional procedure)

It is very convenient to use (4) connected socket sections and either push out the two middle pins or clip off the middle pins with wire cutters to easily solder a solid section to the board when only needing (2) sockets.

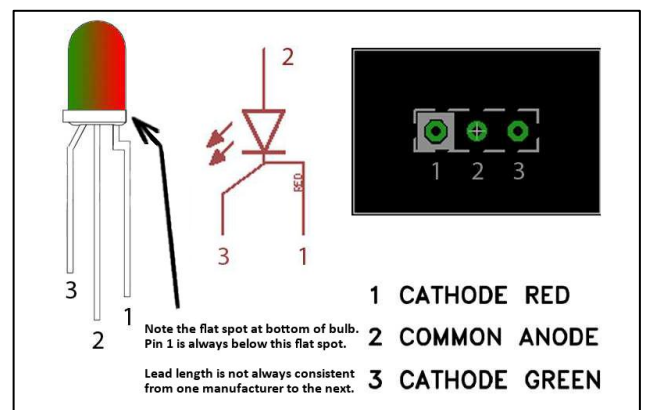
If you are having trouble determining which Dip Switch setting is After Blast and which is Bypass, just turn your Trimmer and you will hear it clearly when you have it in After Blast Position. Typical Unity setting is at about 1:00. Below that you are attenuating the circuit (which can be useful, while maintaining natural Bass loss) and anything above 1:00 will add enhancement to finally a clean Boost.

STATUS LED

D3 is a common anode bi-color LED. The diagram at right shows the pin-out, schematic symbol and pad connection for a common anode LED. The pin-out for the bi-color LED is typically (but not always) as follows:

1st Color Cathode	Is on the "flat" side of the LED (see graphic); 90 degree bend in the lead
Common Anode	Middle lead
2nd Color Cathode	45 degree bend in the lead

The lead 1 pad on the circuit board is marked with a white box.



When connected correctly, the LED will light red when power is applied and the circuit is in bypass mode. The LED will light green when in effects mode. If you wish to use a standard LED, connect the anode to the middle pad and the cathode to the right pad to show the circuit in effects mode. If you use a 3PDT wiring board that includes an LED, you can omit this LED and R5. ***R5 is the LED's Current Limiting Resistor (CLR). If you use a different LED, you may want to change this value to adjust LED brightness.

IC's and transistors are easily damaged by heat from soldering and should never be directly soldered to the PCB.

For transistors, diodes, and LED's, use SIP (Single inline package) sockets. You simply cut the number of sockets required with an Exacto / Stanley knife or by gripping and rocking with pliers. This allows for easy changes and troubleshooting.



Mod:

**Aside from using the on-board Dip Switch and Trimmer (some people prefer the set and forget method) you may use a DPDT switch and potentiometer for mounting on the outside of the enclosure. Keep in mind that the potentiometer should be at roughly 2:00 for unity gain and higher for boost and clarity. It can also be used to cut the volume as well below 2:00 if needed.

Another Mod is to install a 2N5457 for a bit more grit when turning it up than a cleaner MPF102 will provide. As usual Bias from 4.5v to 6v. to suit your rig and ears. You will need to socket R4 and find the correct resistor.

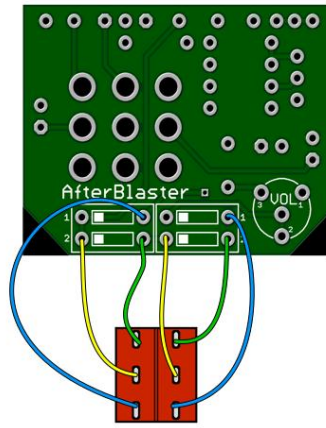
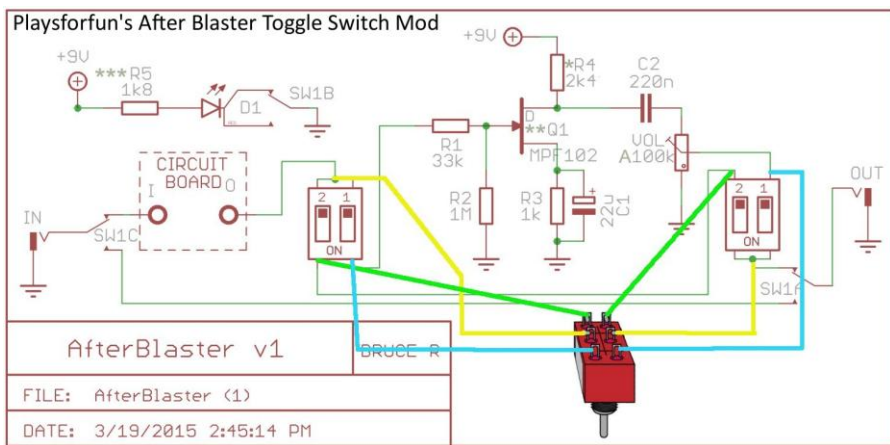
If you wish to try a J201 (even more available grit) you will have to socket and adjust R1 and increase as well as possibly socketing R4 and adjusting till you can attain in roughly 4.5v to 6v. on the Drain leg of each transistor.

As always "socket and see" what you like. When using sockets this allows you to easily make changes without ruining the board by desoldering etc...

Here are two pictures to help assist with DPDT and Potentiometer Mod with video demo:

http://www.youtube.com/watch?feature=player_embedded&v=VtEKqTsHnmo

AFTERBLASTER TOGGLE SWITCH MOD



Add-On Build Guides for all GuitarPCB Builds

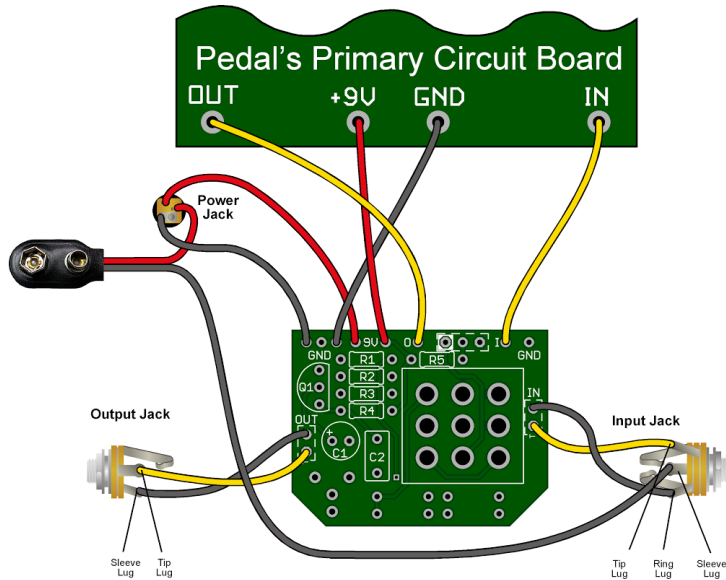
[Soldering Tutorial on Youtube](#)

[Crash Course \[Basic\]](#) - Guide #1 for all things GuitarPCB.

[Crash Course \[Level 2\]](#) - Guide #2 for all things GuitarPCB.

[Tips, Tricks and Tutorials](#) section of the forum contains many innovative pedal building tips and ideas.

WIRING DIAGRAM



This document, PCB Artwork and Schematic Artwork © GuitarPCB.com. Schematic, PCB and this document by Bruce R. All copyrights, trademarks, and artworks remain the property of their owners. Distribution of this document is prohibited without written consent from GuitarPCB.com.