

Building and Operating:  
the Bug Descratcher III kit  
from Jackson Harbor Press

## Introduction:

In 1992, WX7G (Dave Cuthbert) described a circuit in the Hints and Kinks section of QST (September 1992, page 87) entitled: A Debouncer for Semiautomatic Speed Keys. The idea of the circuit was to get rid of scratchy leading edge of transmitted dits and dahs caused by contact bounce with a semi-automatic key (aka bug). The circuit employed was basically a retriggerable one-shot (monostable multivibrator) which would extend the dit or dah by about 10 milliseconds after the end of the dit or dah while providing a continuously low (keyed) output without any of the bounces due to the mechanical bug contact. The Bug Descratcher III does basically the same thing with a different part lineup (although less weight is added to the code elements) and offers the option of a SSR (Solid State Relay) output for universal keying (pos/neg HV tube keying or LV solid state keying) which protects the bug contacts. The Bug Descratcher III can also be used with a straight key.

## General notes on building the Bug Descratcher III

The integrated circuit (U2) and the optional Solid State Relay (U3) are MOS devices. This means that they should be handled as little as possible to prevent static damage. The builder should use a grounding strap and anti-static mat if available or at the very least, work on a grounded metal surface and be sure to touch ground prior to touching these parts.

One decision the builder should make before starting construction of the Bug Descratcher III kit is how the project will be mounted in the case. Ideally the Bug Descratcher III will be mounted in an all metal case to minimize RF pickup - an Altoids tin will work fine. The circuit board can be mounted to the case with 4-40 (1/8 inch) sized hardware.

Since there is a 5 Volt regulator IC in the Bug Descratcher III, it can be powered from either a battery or power supply of at least 6 Volts to 14 Volts.

The components should be inserted a few at a time, soldered in place and then clip the leads. The pads and traces are small and delicate - a small tipped, low power (25 watts or less) soldering iron should be used.

## Building the Bug Descratcher III

Step 1) Get the parts together: All of the board mounted components have been supplied but you will still have to provide off-board items to fully implement the kit. These items include:

- Output connector to external keyed device
- metal case, an Altoids or other candy tin will work fine
- mounting hardware, 4-40 sized
- external power connector or battery holder

Step 2) Identify and orient the components: Most of the components should be fairly easy to identify and place - see the parts list and the parts placement diagram for descriptions. Suggestion: use a magnifying glass to read the tiny printing on the components.

step 3) Insert and solder the components on the main circuit board: Use the parts placement diagram for information on the placement and orientation of the parts. Clip the leads after soldering. The general idea is to start with the lower profile parts and build up to the higher parts. Here is a suggested sequence for installing the parts:

- a) 8 pin DIP socket - should be inserted roughly in the center of the board with the notch up (towards the Jackson Harbor Press legend at the top of the board).
- b) C1 – small yellow .1 uF capacitor marked 104 with long leads - located just above the 8 pin socket top left corner.
- c) C2 – small yellow .1 uF capacitor marked 104 with long leads - located at the top left corner of the circuit board.
- d) C6 – small yellow .01 uF capacitor marked 103 with short leads - located just below the 8 pin socket bottom left corner.

## **Building and Operating the Bug Descratcher III kit from Jackson Harbor Press**

- e) C3 – small yellow .01 uF capacitor marked 103 with short leads - located bottom right quadrant of the circuit board.
- f) R1 – 4.7 k ohm (yellow-violet-red-gold) – form the leads of R1 by bending both leads down 90 degrees and then insert R1 just below C3 in the bottom right quadrant of the circuit board.
- g) R3 – 100 ohm (brown-black-brown-gold – small 1/8 watt) – form the leads of R3 by bending both leads down 90 degrees and then insert R3 just below C6 in the bottom left quadrant of the circuit board.
- h) R4 – 240 k ohm (red-yellow-yellow-gold) – form the leads of R4 by both leads down 90 degrees and then insert R4 above R3 in the bottom left quadrant of the circuit board.
- i) C5 – dark red .01 uF polyester cap marked 103 – insert C2 just above R4
- j) C4 – yellow Tantalum cap marked 2u2 – insert just above the 8 pin socket top right corner with the positive, bumped out side oriented down towards the socket – there is a + sign on the board next the the hole, there is also a + sign on the side of C4 with the positive lead. If this cap is inserted backwards bad things may happed, double check the orientation before soldering in place.
- k) U1 – LM2936 low power 5 volt regulator in TO-92 package – insert between C1 and C2 located at the top left of the circuit board – orient the flat side of the regulator towards the top of the board (with the Jackson Harbor Press legend).
- l) Q1 – Pn2222 or 2n3904 NPN transistor in TO-92 package – insert at the bottom right of the circuit board – orient the flat side of the transistor towards right of the board.
- m) R5 – 820 k ohm (gray-red-yellow-gold) – form the leads of R5 by bending one lead around 180 degrees and then insert R5 just to the left of R4

optional high voltage keying components:

- e) R2 – 1 k ohm (brown-black-red-gold) - form the leads of R2 by bending one lead around 180 degrees and then insert R2 located just to the right of C4 (top right quadrant of the circuit board).
- f) 6 pin DIP socket insert with the notch facing the top of the board located just to the right of R2 (top right quadrant of the circuit board).

Step 4) Check the board: Before proceeding, take the time to check the bottom of the board for solder bridges. Use the bottom view diagram as a guide to visually check for these shorts. It may help to clean the flux from the board and then use a strong light in conjunction with a magnifying glass to see these problems. Soldering problems are the main source of most problems with kits. Also, double check the orientation of the critical components such as the capacitor C4 and the regulator U1. After you are convinced that the board is OK, form the leads of IC U2 to fit in the socket, insert the IC the socket, being sure to follow the parts placement diagram for proper orientation (pin 1 indicated by a notch or dimple should be towards the left side of the board. If the optional high voltage keying option was purchased, form the leads of the SSR (6 pin DIP) and insert the SSR into the socket with the notch or dimple towards the top edge of the board.

Step 5) Solder connecting wires from the board to the input and output jacks and 13.8 volt supply connector. See the parts placement diagram and the hookup diagram for a visual of all these connections.

Operation:

Connect the Bug Descratcher III input to the bug (semiautomatic key) or straight key. Connect the output of the Bug Descratcher III to the transmitter or code practice oscillator. Powerup the Bug Descratcher III and then key the bug, the output of the Bug Descratcher III should follow the input except that the contact bouncing or scratchiness should be removed.

## **Building and Operating the Bug Descratcher III kit from Jackson Harbor Press**

Notes:

If the optional high voltage keying option is installed, use it alone, don't attempt to use both the low voltage and high voltage outputs at the same time. Adding the SSR affects the keying characteristic of the the low voltage output negatively.

One untried modification idea: use a 470 k ohm resistor in place of R4 and a 1 megohm pot instead of R5 – this will allow the user to vary the debounce length from (guesstimated) 6 mS to 18 mS (the nominal debounce is about 12.5 mS).

Please feel free to email with any questions, comments, suggestion or problems with this kit. My email address is:  
wb9kzy@wb9kzy.com

Thanks for choosing the Bug Descratcher III kit and  
Best Regards,

Chuck Olson, WB9KZY

Copyright © 2014 by Charles J. Olson

## Building and Operating the Bug Descratcher III kit from Jackson Harbor Press

### List of parts included with the Bug Descratcher III kit

Ref	marking	Description
C1	104	.1 uf multilayer ceramic .1" lead space capacitor
C2	104	.1 uf multilayer ceramic .1" lead space capacitor
C3	103	.01 uf multilayer ceramic .1" lead space capacitor
C4	2u2	2.2 uf tantalum .1" lead space capacitor
C5	103	.01 uf polyester .2" lead space capacitor
C6	103	.01 uf multilayer ceramic .1" lead space capacitor
Q1	Pn2222 or 2n3904	TO-92, NPN keying transistor
R1	yellow-violet-red-gold	4.7 k ohm 1/4 watt resistor
R3	brown-black-brown-gold	100 ohm 1/8 watt resistor
R4	red-yellow-yellow-gold	240 k ohm 1/4 watt resistor
R5	gray-red-yellow-gold	820 k ohm 1/4 watt resistor
U1	LM2936	TO-92, 5 Volt low power regulator
U2	12F509	8 pin DIP, PIC microcontroller CMOS 8 pin DIP machined pin socket (for U2) circuit board

### optional items for high voltage keying:

R2	brown-black-red-gold	1 k ohm 1/4 watt resistor
U3		6 pin DIP solid state relay (SSR) 6 pin machined pin socket (for U3)

### Items you'll need to provide to complete the Bug Descratcher II kit

optional Metal case (an Altoids tin is fine)  
4-40 sized (1/8 inch) mounting hardware  
input jack  
output jack  
13.8 volt power jack, plug, cable OR 9 Volt battery connector  
solder, wire