Build Instructions for the Geiger Kit – v2 PCB

General tips:

- It's a good idea to print these instructions before you begin.
- Refer to the Parts List (below) as you build the kit. It describes orientations and options for various parts as you go along.
- Do want to mount the piezo and/or LED on the case? Run on AA batts or 9V? Use different headers? Again, the Parts List points some of these options out.
- Referring to the schematic (below) can be helpful.
- Follow the build sequence below and test as you go.
- Missing parts / extra parts - you are more likely to get an extra part. If something is missing let me know and I'll send it out.
- Take your time! It takes at least 2-3 hours to build this kit. Solder the right part, the right way, the first time. Parts are hard to desolder.

Soldering:

To cut down on noise, the PCB uses a "ground plane". So all of the lighter green on the bottom of the board is copper, and it is connected to the ground. (It's coated, but can't be trusted to insulate.) The reason for mentioning this, is so you understand why a neat soldering job is important. Joints that slop over the pad and on to the ground plane will cause a problem.

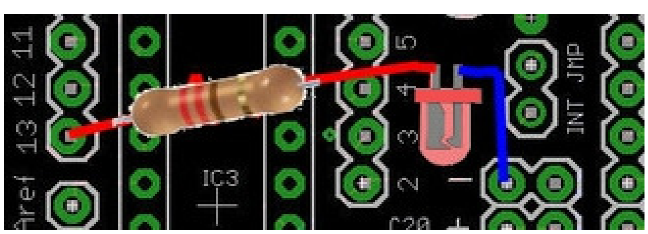
When you solder, start with a good iron, with nice clean tip that's freshly tinned. Solder the joint so that you have a nice round dot that stays inside the darker green. The clearance between the backplane and the pads is pretty close - do not use too much solder and add enough heat for a good flow.

You will notice some pads *do* connect to the back plane. These have 4 little traces from the back plane to the hole - like a "+" . These will take more heat.

Suggested build sequence:

- Add all resistors - except R5 - it's easier to work on a board that lays flat. It's not a bad idea to "Y" out the leads, tack solder, clip the leads, and resolder or reheat the joint.
- Add the diodes, then capacitors, and then transistors (C1 last). Refer to the photo below to make it easier to see the orientation of the diodes.
- Add R5 - R5 controls the high voltage, and comes preset to about 18Ω for the v2.0 boards which should give you good results from the tube without adjustment. (see **HV Test and Adjust** below)

- Decide on adding the regulator or not (see **Powering the Geiger** below)
- Complete the HV section (up to Q3) and test it. (see **HV Test and Adjust** below)
- Complete the detect and click section.
- Add the GM tube and power up the board. If you hear clicks congratulations! The hard part is over.
- Complete the ATmega section. You can add an LED (with resistor) between Pin 13 and ground to test. The default sketch will flash the LED 4-5 times at power-up or when the Reset button is pressed.



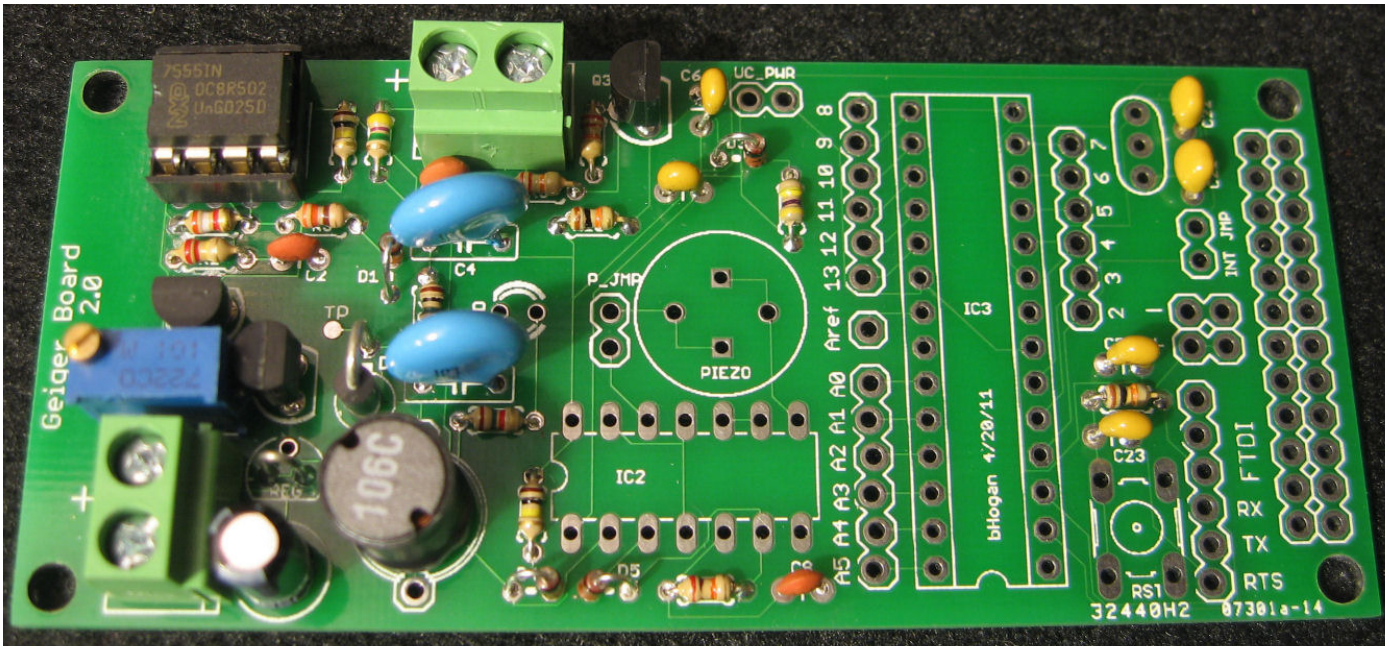
- Done – see **HV Test & Adjust** below.

Parts List for v2.0 Geiger Kits

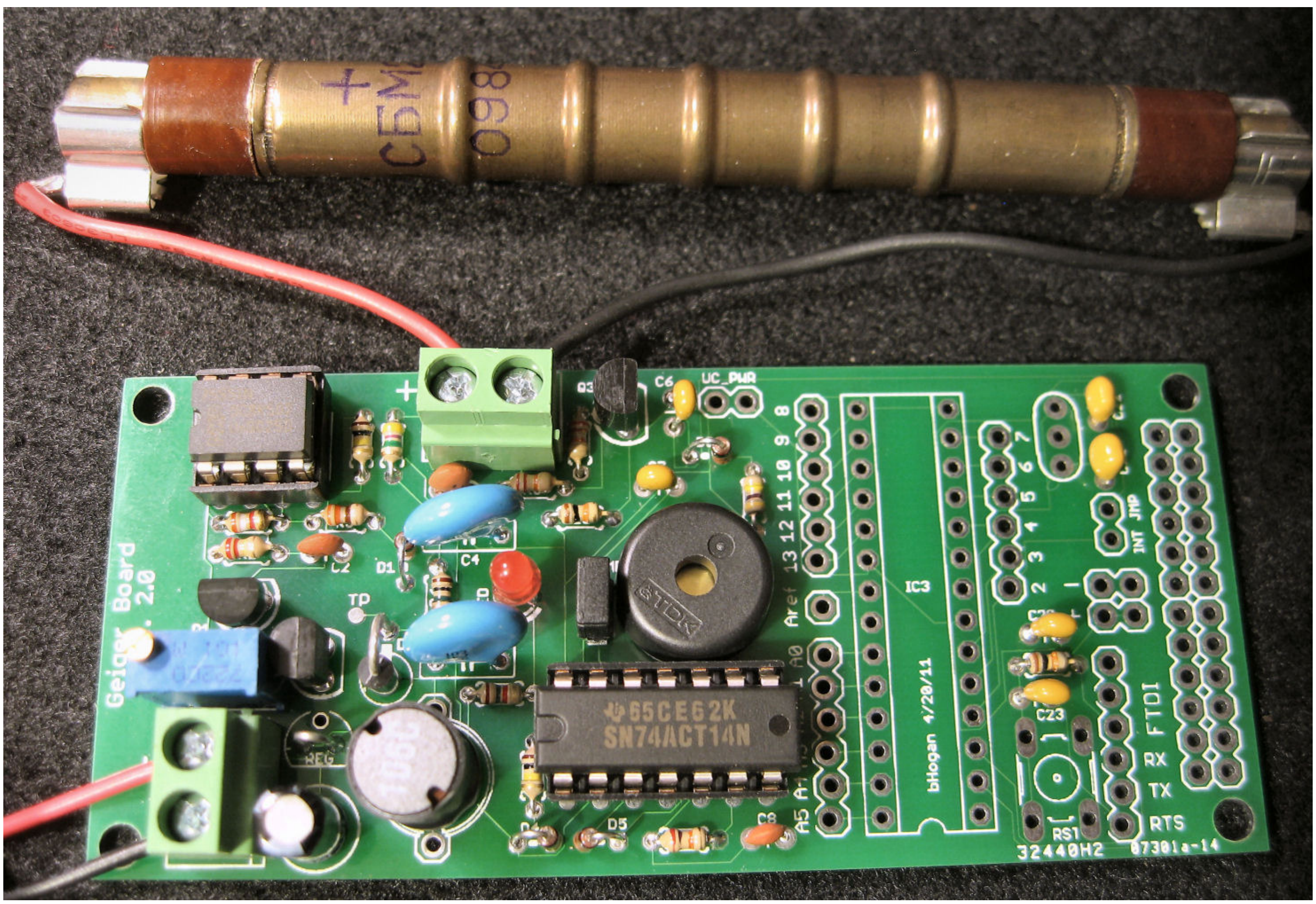
Ref #	Qty	Value	Description	Notes	#
R1	1	220KΩ	RD,RD,YL 1/8 W	**NOTE: Color bands on resistors are sometimes hard to distinguish (i.e. violet almost black). If in doubt it's best to check them with a meter before you solder.**	
R2	1	3.9KΩ	OR,WT,RD 1/8 W		
R3	1	330Ω	OR,OR,BN 1/8 W		
R4, R13	2	100KΩ	BN,BK,YL 1/8 W		
R5	1	100Ω pot	10 turn pot	**Note:** pot is preset to ~18Ω **polarity:** adj screw on left	
R6	1	1MΩ	BN,BK,GN 1/8 W		
R7	1	4.7MΩ	YL,VT,GN 1/8 W	**Option -** If using a LND-712 GM tube R7 should be higher but 4.7M is reported to provide better sensitivity.	
R8, R9	2	22KΩ	RD,RD,OR 1/8 W		
R10, R20	2	10KΩ	BN,BK,OR 1/8 W		
R11	1	470KΩ	YL,VT,YL 1/8 W	adjusts duration of the click - see schematic	
R12	1	1.5KΩ	BN,GN, RD 1/8 W		
R14	1	27KΩ	RD,VT,OR 1/8 W	adjusts pitch of the click - louder when resonant - see schematic	
C1	1	100uF	16V electrolytic capacitor	**polarity:** "-" stripe down.	
C2	1	.001uF	#102 (1nF) ceramic cap		
C3, C4	2	.01uF	#103 1KV ceramic cap	not polarized	
C5	1	330pF	#331 ceramic capacitor		
C6, C7, C20, C23	4	.1uF	#104 ceramic capacitor	reform leads (cut shorter and squeeze long ways with long-nose)	
C8	1	.022uF	#223 (22nF) ceramic cap		
C21, C22	2	22pF	#220 ceramic capacitor		
D1, D3, D4, D5	4	1N4148	signal diode	**polarity:** Cathode (banded side) goes up. **See pictures for orientation.**	
D2	1	MUR160[1]	600V 1A Ultra Fast diode	**polarity:** Cathode (banded side) goes into smaller silkscreen circle. Cathode (banded side) goes up. **See pictures for orientation**.	
L1	1	10mH	inductor	higher value = higher voltage	
Q1,Q3	2	2N3904	NPN BJT transistor	May need to reform the leads. Bend center lead back.	
Q2	1	KSP44	NPN HV transistor (same as MPSA44)	bend center lead back	
IC1	1	ICM7555[1]	CMOS 555 timer	**polarity:** Notch on left.	
IC2	1	CD74ACT14	Hex Inverter w Schmitt Trigger	**polarity:** Notch on left. **Option** - if removed stops click and led - interrupt still sent to uC.	
REG	1	78L05	5V / 100mA regulator	**Option** - if VCC 4.5-5V bypass with jumper IN to OUT	
OSC	1	16MHz	crystal	(not polarized) **Option** - can use 16MHz resonator & omit C21 and C22.	
PIEZO	1	~4 kHz	12 mm piezo - 5mm pitch	(not polarized) **Option** - for mounting piezo in case, jumper and use 2 pin connector.	
LED	1	red	3mm	**polarity:** Small flat on side, **or shorter lead**, goes left. May want to leave leads long so it's not hidden by the caps. **Option** - for mounting LED in case, replace with 2 pin conn.	
switch	1		push button	**"polarity:"** follow lead spacing. Snaps in and will lay flat.	
screw term	2	2 pin	5 mm pitch	**Option** - replace with connector of your choice	
socket	1	8 pin	IC socket	install with notch on left	
socket	1	14 pin	IC socket	install with notch on left	
socket	1	28 pin	IC socket	install with notch down	
header	3	6 pin fem.		Pads are offset for easier placement. **Option** – use other type of header	
header	1 1	6 pin male 6 pin male 90°	FTDI header	Pads are offset for easier placement. 90° male usually best for FTDI. (does cover some user pads) **Option** – use other type of header	
header	3	2 pin male		for piezo interrupt jumper, and uC power	
misc header	2 1	2 pin female 2x2 male	for user power pins	**Option** - use either female or male header depending on need.	
jumper	3	2 pin	for 2 pin headers	For more comfortable soldering - put jumper over pins first. :-)	
fuse clips	2	¼"	for HV conn. to GM tube	Reform tabs as needed. Solder stranded wire to these.	
PCB	1	v2.0		Version number on left side of board	
IC3	(1)	ATmega328	AVR microcontroller	**Kit B only** – pre-loaded with Arduino bootloader & demo ap.	

[1] May be a similar part with a different number.

Board built up to HV test. Modified for R5 pot and 5V regulator bypassed. . . .



Board built up to click test with tube added . . .



HV Test & Adjust

This is going to sound more complicated than it is, but this page will show you how to measure, test and adjust the HV section of the circuit.

Measuring the HV:

It's a bit tricky to measure the high voltage. The GM tube needs a lot of voltage but only a tiny amount of current. So the HV circuit only needs to provide a very small current, and that's what it does. This is good because the battery will last a long time, but it makes measuring the high voltage a bit more complicated.

When measuring voltage, a decent DVM puts a load on the circuit it's measuring of about 10MΩ (1MΩ for a cheap DVM). This load is far too much for the tiny amount of current available, and the DVM will read much lower than the actual voltage. I've read that you need 1 gig-ohm (1000MΩ) of input impedance to get accurate values of the HV for Geiger circuits.

One way to increase the input impedance of your DVM is to put large resistors in series with the probe and multiply the reading on your DVM. For example adding 9 10MΩ resistors adds 90MΩ. For a 10MΩ DVM you multiply the reading by 10. For a cheap 1MΩ DVM you multiply the reading by 91. If you want to a full gig-ohm of input impedance it's best to just buy a high value resistor - (example). There are good explanations of the subject with build instructions here or here.

I wanted all of the HV readings that I refer to on this site to be easy to duplicate without the expense of a 1GΩ probe. So for the HV readings mentioned here, I'm using a cheap 1MΩ DVM with a 90MΩ probe and multiplying by 91. (If you're using a 10MΩ DVM you should get better readings.) I'd be interested in hearing the readings people get from very high impedance DVMs.

The high voltage is measured from the cathode (band side) of D2 (HV TP) and the negative side of the GM tube.

Using the 90M probe I get 570V with the SMB-20 operating well, and the battery drain at a good point.

Testing the HV:

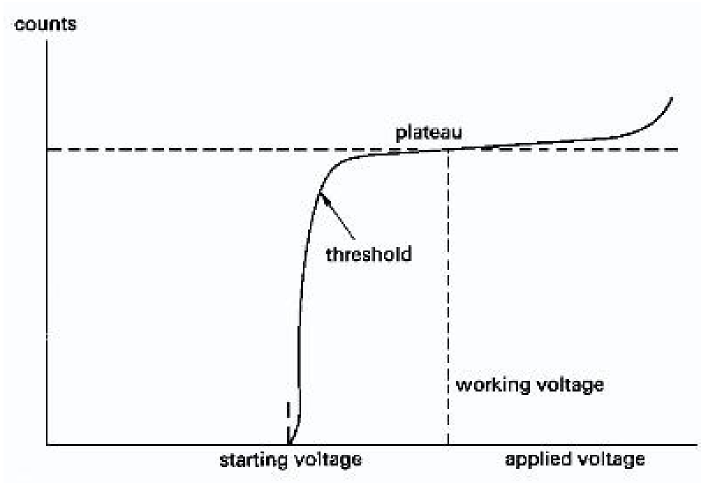
After building out the HV section of the board, it's a good idea to see if things are working up to that point. First read through the next section to get a sense of the final picture. Then, using the 90M ohm probe described above, or even without it, connect the battery, and measure the voltage between the cathode (band side) of D2 (HV TP) and the negative side of the GM tube. If you get something like 200V without the probe - congratulations! The HV circuit is working.

I can't recommend this (for obvious reasons), but I found that putting two fingers (same hand) across the wires that go to the GM tube will simulate an event on the tube pretty well. The current is so low, I can't feel it. I use this as an easy way to test the circuit without the tube. Again, I can't recommend this, however.

Adjusting the HV:

R5 controls the high voltage. (see Circuit Description on web site). It has been preset for about 18Ω. This should give you about 570V at the TP with the 90MΩ probe described above. The idea is to adjust the HV so it's working voltage is in the middle of the plateau as shown below. *Note that the circuit will work just fine with R18 set to 18Ω.*

Using some kind of active source, lower the HV (CCW) until you get no response. Note R5 is a 10 turn pot. Now increase the HV (CW) until you are getting a good response from the source. This point is the threshold in the graph above. Note the position of the pot or the HV at this point. Continue to increase the HV along the plateau. At some point the HV will drop to near zero. Note the position or voltage and set the pot CCW so it's between that point, and the threshold. (You'll probably wind up with the pot at 18 ohms again!) Adjusting the HV is probably more important when you use tubes other than the SBM-20.



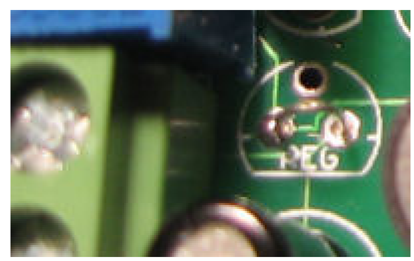
These are the results I got in my testing (1 MΩ DVM with 90M probe):

- R5 = 25R - 390V - detection just starts - threshold
- R5 = 23R - 440V - stable readings start- beginning of plateau
- R5 = 20R - 525V - (sweet spot) - about center of plateau
- R5 = 16R - 650V close to breakdown, battery current jumps up

Powering the Geiger

The Geiger kit will accept voltages between 4-9VDC. When applying voltage in the range of 5.5-9VDC (a 9V battery for instance) the voltage regulator must be installed. This will supply a working voltage of 5V to the circuit. Remember that the working voltage can not exceed 5.5V with ATmega328 installed.

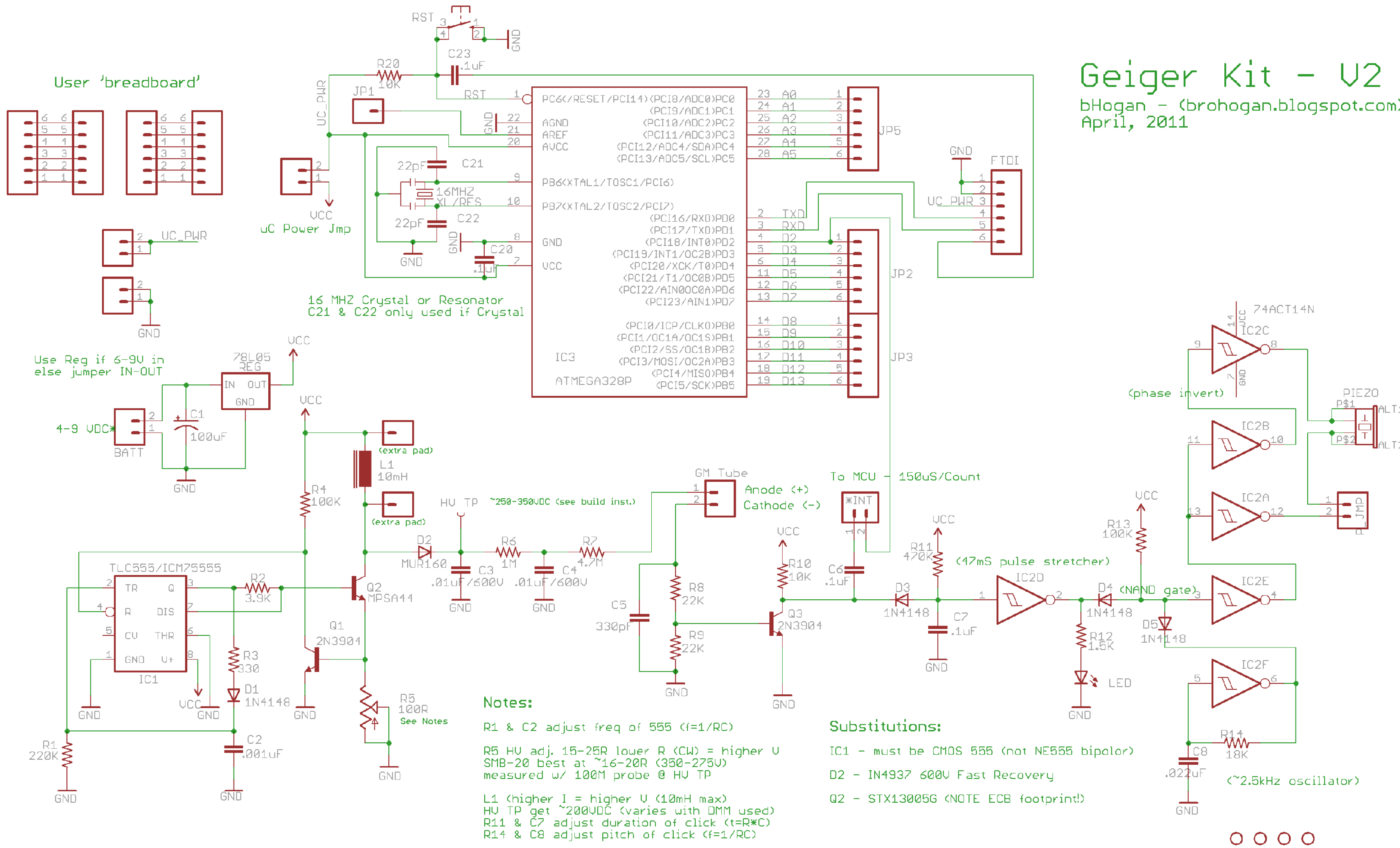
When powering the Geiger kit with a voltage in the range of 4-5.5VDC (3 AA or AAA ZnC batteries or 4 NiMH batteries) I recommend that the voltage regulator is not installed and is bypassed. This is easily done by soldering a straight piece of wire between the IN and OUT pads on the regulator like the picture on the right shows.



Since the HV circuit is self regulating, there is no increase in its output when higher voltages are used. For the best battery life, I'd suggest running without the voltage regulator, using 4 NiMH batteries, or 3 AAA ZnC batteries if case size is an issue.

Note that many FTDI cables also supply 5V power to what they are connected to. So when they are connected to the Geiger board, it's a good idea to disconnect the battery. The FTDI cable will then power the whole board just fine.

Beginning with the v2.0 board there is a jumper for "uC PWR" (top center of board). Removing this jumper separates the positive supply voltage (VCC) between the Geiger circuit and the ATmega328 (including it's FTDI connector, and user "+" header). This allows you to run the Geiger from one source, and the microprocessor from another (with the grounds in common). You could supply power for the Geiger through the screw terminals and supply power to the microprocessor from either the user "+" header, or the FTDI connector with the jumper removed.



Geiger Kit - V2

bHogan - (brohogan.blogspot.com)
April, 2011

Notes:

R1 & C2 adjust freq of 555 (f=1/RC)

R5 HV adj. 15-25R lower R (CW) = higher V
SMB-20 best at ~16-20R (350-275V)
measured w/ 100M probe @ HV TP

L1 (higher I = higher V (10mH max)
HV TP get ~200VDC (varies with DMM used)
R11 & C7 adjust duration of click (t=R*C)
R14 & C8 adjust pitch of click (f=1/RC)

Substitutions:

IC1 - must be CMOS 555 (not NE555 bipolar)

D2 - IN4937 600V Fast Recovery

Q2 - STX13005G (NOTE ECB footprint!)