Build Instructions for the Geiger Kit – V4.0.2 PCB

Common Build Problems:

These are the most common problems I've noticed a few builders had . . .

- **Solder dust / flux on board** – HV is very easy to short – see ***Cleaning the board*** below.
- **Too much solder** - This can cause shorts between the lead and the ground plane - Read ***Soldering*** below.
- **Parts orientation wrong** - generally RTFM issues, and not referring to the "Notes" and pictures.

General tips:

- "***Sometimes just a few hours of trial and error debugging can save minutes of reading instructions.***" Even if you're experienced, you run the risk of wishing you had considered something beforehand.
- Use the **Build Sequence and Parts List** (below). It describes orientations and options as you go.
- Use the pictures and schematic (below) to help you.
- Missing parts / extra parts – You are more likely to get an extra part, but if something is missing, let me know.
- 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 unsolder. Try to enjoy the journey.

Soldering:

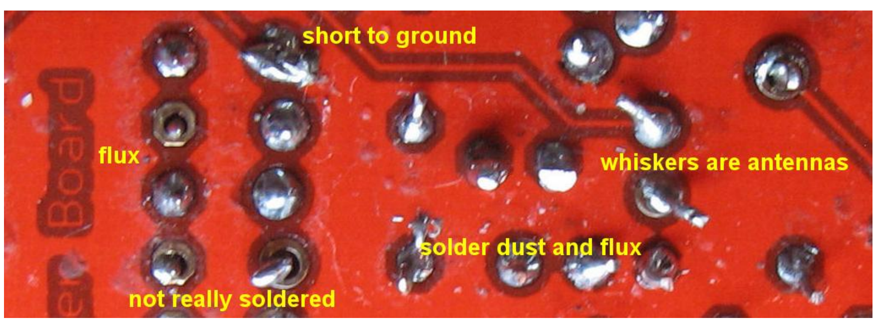
To cut down on noise, the PCB uses a "ground plane". So all of the lighter red on the bottom of the board is copper, and it is connected to the ground. The reason for mentioning this is so you understand why a neat soldering job is important. Joints that slop over the pad and on to the ground plane will cause a problem.

When you solder, start with a good iron, with a good tip that's freshly tinned. Solder the joint so that you have a nice round dot that stays inside the darker red. Do not use too much solder, and add enough heat for a good flow. The holes are plated through so don't worry about getting solder up to the top of the board.

Sometimes it's best to shorten long leads before you solder them, or re-solder them after they are cut. You will notice some pads will connect to the back plane. These have 4 little traces from the back plane to the hole, like a "+" . These pads will require more heat. I usually solder that side of the part last.

I do not recommend using lead free solder for the kit. In my experience, it makes parts even harder to unsolder and more heat is needed which may damage the pads. I will not do any board repair if lead free solder was used.

Here is a picture of what your board should not look like . . .



Build sequence and startup:

These steps will take you through building and starting up your kit for the first time. Please follow all of these steps!

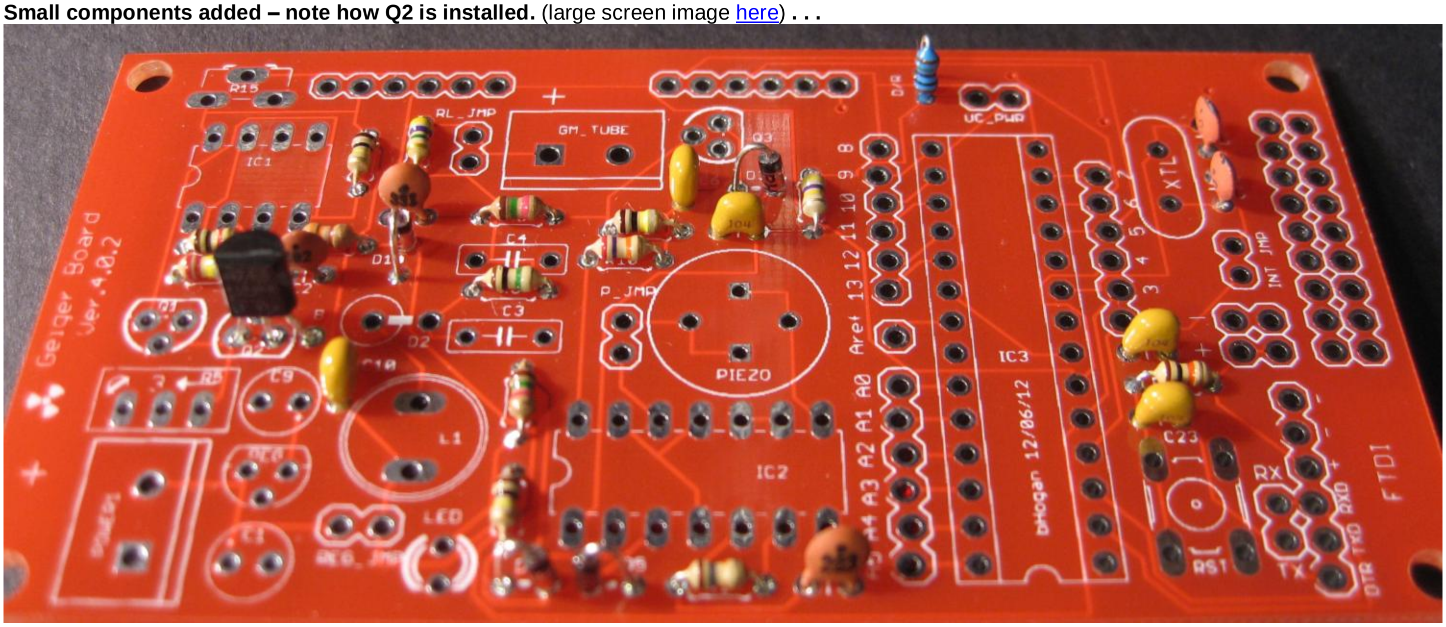
Step 1 - Building the Kit:

Use the table on the next page as your guide to building the kit. Its approach is to build the board by height – starting with the shortest components. It's easier to work on a board that lays flat and holds the parts in place when you flip it over to solder. While working, refer to pictures on the page after the table, and the completed board on the last page, to double check orientations of parts, etc.

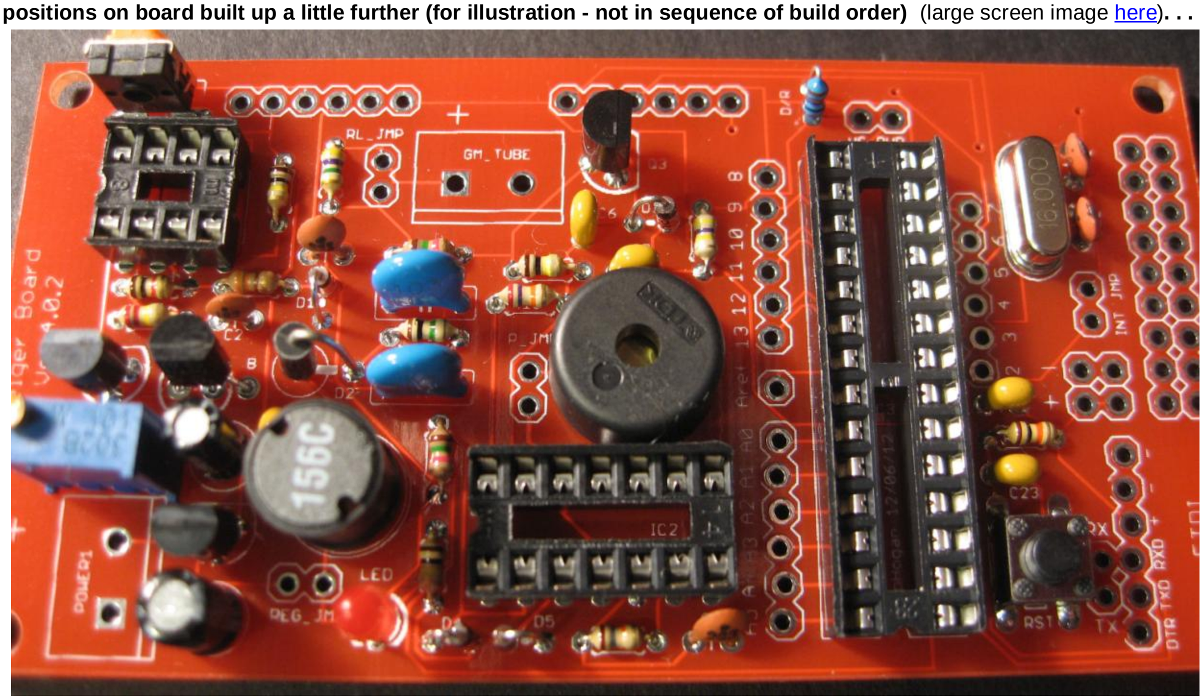
Build Sequence and Parts List for v4.0.2 Geiger Kits					
Ref #	**Qty**	**Value**	**Description**	**Notes** polarized? ->	Y N
PCB	1	v4.0.2	1.85" x 3.60" (~4.7 x 9.14 cm)	**Orientation:** "Geiger Board – Ver. 4.0.2" is **left** side of board.	N
R1	1	220kΩ	RD,RD,YL (all fixed resistors are 1/8W)	**Color bands on resistors may be hard to distinguish (i.e. violet almost black) If in doubt, it's best to check with a meter before soldering.**	N
R2	1	1kΩ	BN,BK, RD	Careful! Be sure last band is red – not orange (R20)	N
R3	1	330Ω	OR,OR,BN		N
R4, R9, R13	3	100kΩ	BN,BK,YL	**Note**: hard to read "R8" and "R9" on board – **R9** closest to piezo.	N
R6	1	1MΩ	BN,BK,GN		N
R7	1	4.7MΩ	YL,VT,GN	**Note:** *Anode Resistor* – Value good for most tubes. Some may work better with other values (i.e. 10MΩ). Check the data sheet for your tube. If you need more resistance you can also add it right at the tube.	N
R8, R12	2	1.5kΩ	BN,GN, RD	**Note**: hard to read "R8" and "R9" on board – **R8** closest to GM_TUBE term.	N
R11	1	470kΩ	YL,VT,YL	R11 sets duration of the click - see schematic	N
R10	1	27kΩ	RD,VT,OR		N
R20	1	10kΩ	BN,BK,OR	Careful! Be sure last band is orange – not red (R2)	N
R14	1	15kΩ	BN,GN,OR	Sets the pitch of the click - louder when resonant.	N
C2	1	.001uF	#102 (1nF) ceramic cap		N
C5	1	330pF	#331 ceramic capacitor		N
C6, C7, C10, C20, C23	5	.1uF	#104 ceramic capacitor	(C20 is just above C23)	N
C8	1	.022uF	#223 (22nF) ceramic cap		N
C21, C22	2	22pF	#220 / #22 ceramic cap		N
OSC	1	16MHz	crystal		N
socket	1	8 pin	IC socket	notch on left **Suggestion:** tack in all 3 sockets, then solder all pins at once.	Y
socket	1	14 pin	IC socket	notch on left	Y
socket	1	28 pin	IC socket	notch down	Y
D1, D3, D4, D5	4	1N4148	signal diode	**polarity:** For all diodes - bend over the lead on the banded side (cathode). **See pictures below for orientation.**	Y
"D/R" (diode or resistor)	1	150Ω resistor supplied	BN,GN,BN (blue body)	(Located just to the right display header.) Current limiting for the backlight. 150Ω uses minimum current with reasonable brightness. A smaller value, or even a diode could be used instead of a resistor. (If a diode is used, the banded side goes to the upper pad). **To add a switch for the backlight you can use these connections with the resistor in series with the switch.**	Y
Q1,Q3	2	2N4401	NPN BJT transistor	Bend the center lead back – don't try to push in all the way to PCB.	Y
Q2	1	STX13005	NPN HV transistor	**Mounting is different than Q1 & Q3!** The STX13005 uses the "B" pad for the rightmost lead. Top most hole not used. Refer to pictures below.	Y
REG	1	78L05	5V / 100mA regulator	Bend the center lead back. – see REG_JMP under **Powering the Geiger** for usage.	Y
switch	1		push button	**"polarity:"** follow lead spacing. Snaps in and lays flat. **Trim leads**	Y
PIEZO	1	~4 kHz	5 x12 mm - 5mm pitch	Note: *partially* covering the hole with tape can make it louder	N
C3, C4	2	.01uF	#103 HV ceramic cap		N
R15	1	vertical 10KΩ	1 turn pot for LCD	**polarity:** orange adj screw toward top of board **Trim leads**	Y
D2	1	UF4007[1] MUR160	600V 1A Ultra Fast diode	**polarity:** Bend over the lead on the banded side (cathode) The anode goes into larger silkscreen circle, and the cathode goes into the hole on the right. **See picture below for orientation.**	Y
header	5	6 pin fem.	I/O pins & LCD	**Suggestion:** tack in all 5 headers, then solder them all pins at once. Plug in LCD if possible to line up LCD headers before soldering.	N
header	1	2x2 female	for user power pins	2 Vcc & 2 Gnd pins for powering add on devices like the LCD	N
header	1	2 pin fem.	Rx & Tx pins breakout	Installs near FTDI header. Used for serial device – (i.e. GPS) or the Geiger Shield with a GPS.	N
header	1	6 pin 90°	FTDI header	To connect FTDI cable for serial output or programming.	N
jumper blocks and 2 pin headers	5 ea	assorted colors for 2 pin male headers	These jumpers provide the options described in the ***Jumpers*** section that follows	**Suggestion:** Put the jumper block on the header before soldering (easier on fingers ;-). **Normally the RL_JMP is not left in - remove this jumper block after soldering. You must configure REG_JMP for your battery type before powering on the kit. This is all described in Step 3 below.**	N
C1	1	100uF	16V electrolytic capacitor	**polarity:** "-" stripe to the right.	Y
C9	1	4.7uF	50V electrolytic capacitor	**polarity:** "-" stripe to the left.	Y
screw term	2	2 pin	5 mm pitch	**Trim leads Option** - replace with a connector of your choice	N
R5	1	100Ω pot	blue 25 turn HV pot	**Note:** pot is preset to ~26Ω **polarity:** adj screw on left	Y
L1	1	15mH	inductor	Orientation shouldn't matter but maybe less noise if installed as pictured	?
LED	1	red	3mm	**Note:** This is the little red LED – not a larger yellow one. You can mount this directly on the board, or you can extend the LED in various ways. **polarity:** Small flat on side, **or shorter lead**, goes down.	Y
IC1	1	TLC555	CMOS 555 timer	Bend all pins inward a bit on a flat surface. **polarity:** Notch on left.	Y
IC2	1	CD74ACT14	Hex Inverter w Schmitt Trigger	Bend all pins inward a bit on a flat surface. **polarity:** Notch on left. **Option** - if removed stops click and led - interrupt still sent to uC.	Y
IC3	1	ATmega328P	AVR microcontroller	Reform pins **polarity:** Notch down	Y
fuse clips	2	¼"	for HV conn. to GM tube	Reform as needed. Solder stranded wire to these. **Tube is polarized**	Y

Assembly Images:

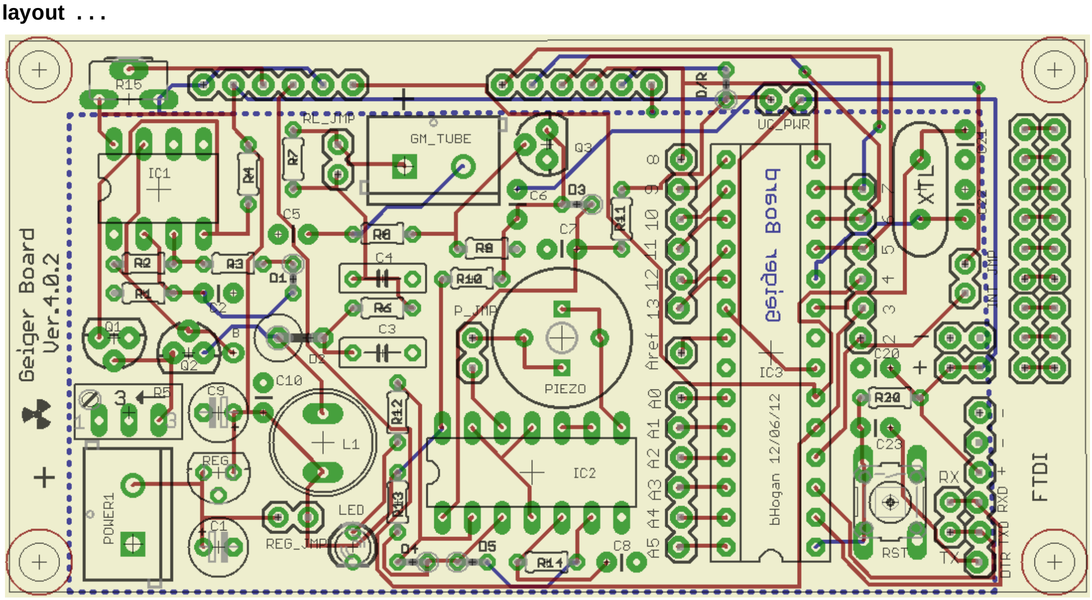
Small components added – note how Q2 is installed. (large screen image here) **. . .**

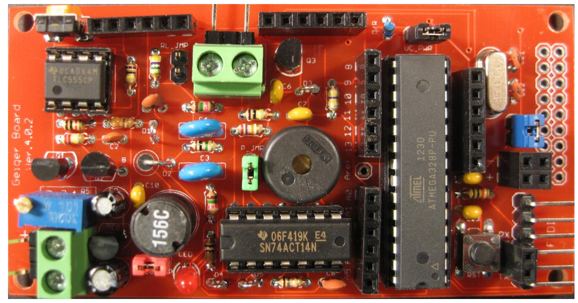


Diode positions on board built up a little further (for illustration - not in sequence of build order) (large screen image here)**. . .**



V4.0.2 board layout . . .





Congratulations!

That took some work.

There is a larger picture of the completed board on the last page, and a large screen image [here](#).

Step 2 - Cleaning the board:

It doesn't take much of a conductor for 500V to get around on. So a clean board is important. At a minimum, brush the bottom of the board with an old toothbrush to remove any solder dust when you're finished soldering.

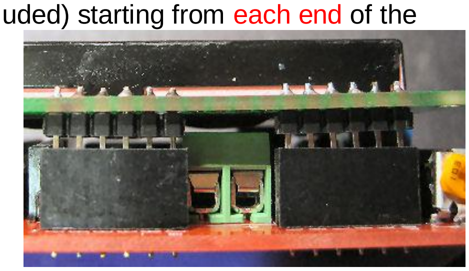
If you search, you'll find many methods for removing excess flux. One of the simplest is to use alcohol – the kind from the hardware store - and a toothbrush. Be sure to blow off the water that is created or at least let it dry well before powering up the board. There are better solvents like commercial flux removers. Just be careful not to use something that removes more than the flux! When in doubt use alcohol or a commercial flux remover.

Step 3 - Configure the jumpers:

It is important that you configure the jumpers on the kit before you power it on. Please refer to the ***Jumpers*** section below to help you decide which jumpers to leave in, and which jumpers to remove. As a quick safety note, the **RL-JMP** is normally not installed, and if you use 9V, be sure to remove the REG jumper. (Normally the P_JMP, INT_JMP, and UC_PWR jumpers are left in.)

Step 4 – Add the LCD:

If an LCD display was included with your kit, solder the two 6 pin male headers (included) starting from each end of the pads on the LCD. This leaves four pads in the center free. If you use your own LCD, you may want to cut the center 4 pins. This makes it easier to run the wires to your tube and prevents possible shorts. (You can also extend the wires to the LCD by either using those LCD female headers, or the connection diagram on the web site page for ***Adding an LCD***.)



By the way, if you connect that LED between I/O pin 13 and ground, it will flash a few times at power-up or when the Reset button is pressed. You can use this to test your ATmega section if you don't have an LCD yet.

Step 5 - Power up the board:

Decide on how you will power your Geiger Kit. Refer to the ***Powering the Geiger*** section below to help you decide. Powering with around 5V with REG jumper in is the best way to go in my opinion. Observe polarity! It should click once. You will need to adjust the contrast pot for the LCD (usually almost fully clockwise) before you can read the display.

Step 6 – Connect your GM tube:

Now you can add your tube. If you don't have one yet, you can test the HV and the click circuit by quickly shorting the tube wires across a resistance like a pencil line. (I use my finger, but I can't suggest that you do that.)

If you hear clicks congratulations! Now you might look at the other sections below on ***HV Test & Adjust***. The HV should already be adjusted to work with most tubes.

Step 7 – Adding the alarm and second screen button:

A "second screen" and an alarm feature are included in the software. Using these features requires two additional parts – a push button, and some type of a warning device. For testing, your kit includes a simple push button (like the reset button) and a special "test" yellow LED. See ***Connecting an Alarm*** below for details.

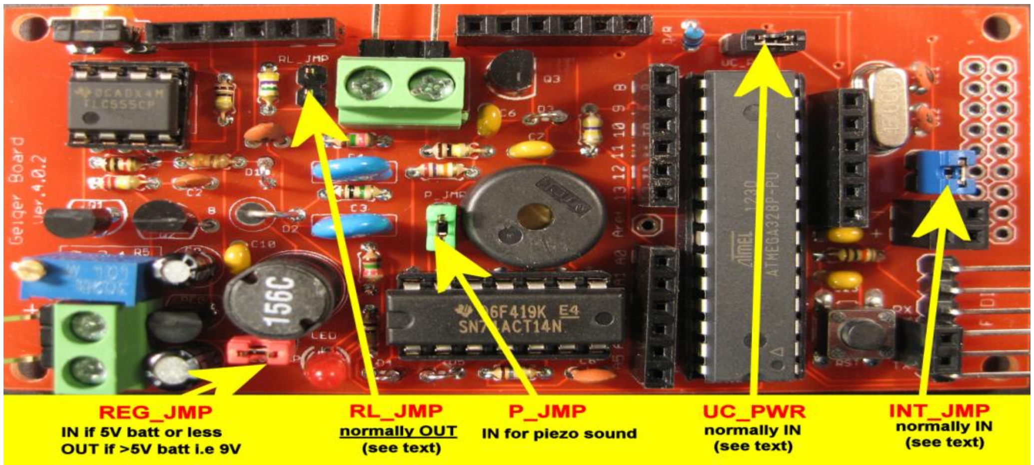
Step 8 – More info:

You will find more a lot more information on [DIY Geiger Counter web site](#) including troubleshooting and projects.

Enjoy the kit!

The Jumpers:

There are 5 jumpers on the Ver.4.2 board. You should configure them properly *before* powering up the board.



REG_JMP **(important)**

This jumper bypasses the 5V voltage regulator. This jumper *should* be installed if you are running on 5V or less. When running at voltages of 5.5V or higher (9V for instance) this jumper ***must not*** be installed. Refer to the ***Powering the Geiger*** section below for more details.

RL_JMP **(important)**

When installed this jumper bypasses (shorts out) R7 which is the anode ("load") resistor for the GM tube. Normally this jumper is not installed. It is only used if you will be adding the anode resistor directly at the GM tube. **Caution:** With this jumper in, the anode resistor is bypassed, and the HV from the kit can give you a bit of a bite! Be sure to use an anode resistor either on the board (jumper out) or at the tube. Running a GM tube without an anode resistor may cause the tube to avalanche problems and shorten it's life.

P_JMP

This jumper disconnects the piezo. You can replace the jumper block with female 2 pin header and wire it to a switch. The switch will turn off the sound from the Geiger so that the other people on the airplane won't notice it. Note the LED will continue to flash with this jumper removed.

INT_JMP

This jumper connects the ATmega328 ("uC") to the Geiger circuit. This is the only connection between the two. Each *event* creates a negative going pulse that is sent to the "Interrupt 0" (pin 2) of the uC via this jumper. So when this jumper is removed, the microcontroller will stop counting. This jumper might come in handy if you wanted to run the events into a different microcontroller. Also, as you will see below, it plays a part in powering off the microcontroller completely.

UC_PWR

As mentioned in Powering the Geiger, this jumper separates the positive supply voltage (Vcc) between the Geiger circuit and the ATmega328. This includes the FTDI connector Vcc, and the two "+" pins on the 4 pin power header. This jumper gives you several options.

It allows you to run the Geiger without the microprocessor and display. This will conserve a large amount battery power. One note on this is that if you remove or put a switch on this jumper, you may also need to remove or switch the INT JMP as well. Otherwise the event pulse will go to an un-powered microcontroller pin and the response from the piezo and LED could be reduced.

This jumper also allows you to run the Geiger from one source, and the microprocessor from another. Be sure to keep the grounds common. With the jumper removed, 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. It's not a good idea to leave the jumper *in* and do this. In fact, it's a bad idea. However, I've done this accidentally several times with no damage.

Finally you can take advantage of the jumper to control the HV from your sketch. Simply remove the jumper and run a wire from an output pin you are controlling to the positive screw terminal. Then power the board from the 4 pin power header. Putting a HIGH on the output pin will turn on the HV and a LOW will turn it off.

Powering the Geiger

You have options when powering the Geiger kit. The two *main* options are AA/AAA batteries or a 9V battery.

Powering at less than 5.5V:

When using AA or AAA batteries, they will typically be used to supply less than 5.5V. Since alkaline cells deliver ~1.5V and NiMH cells deliver ~1.2V you can use 3 alkaline or 4 NiMH (AA or AAA). The key is to stay under 5.5V - the max for the microcontroller. Without a microcontroller and display, the Geiger kit draws less than 5mA. However, with the microcontroller and backlight LCD, the Geiger can draw over 100mA. Because of this, I prefer to run with 4 NiMH since they can provide ~2000mAH of power.

When running under 5.5V, the voltage regulator should be bypassed by installing the REG_JMP jumper. The regulator will always drop about 1.4V so if it's not bypassed, you would also be running at a lower voltage than expected.

Note: Never power the Geiger with 9V if the REG_JMP jumper is installed!

Powering at more than 5.5V:

This typically means using a 9V battery. On the plus side they are small and easily changed. On the minus side, a typical 9V battery only supplies ~600mAH. **When using 9V, the REG_JMP jumper must be removed so that the voltage regulator can lower the voltage to ~5V.**

Other Options:

For the ultimate in flexibility you might consider step up / step down voltage regulator. When used between your batteries and the kit, it allows for battery voltage above and below the 5V needed. Here is one I would suggest. It takes an input of between 2.7- 11.8V and can output least 500mA at 5V with a typical efficiency of 90%.

You could of course supply 5V regulated power from a power cube (bypass the regulator). I did a short test with LiPO batteries. 3.7V is just under the ATmega328 spec, but some do run at that without problems. I had to adjust the contrast and backlight on the LCD but the ATmega328 was working. The HV circuit works fine at that voltage.

Powering with the FTDI and the uC PWR Jumper:

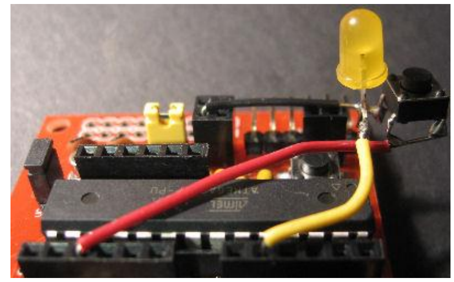
Most FTDI cables can also supply 5V power to what they are connected to. So when they are connected to the Geiger board, it's a good idea to disconnect the battery. The FTDI cable will then power the whole board just fine.

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 its FTDI connector, and the power 2x2 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 "+" power header, or the FTDI connector with the uC_PWR jumper removed.

Note that the jumper can also be removed in order to run the Geiger without the microprocessor to conserve battery power. However, with the microprocessor shut down, you should also remove the INT jumper. Otherwise, the response from the piezo and LED may be reduced.

Connecting an Alarm

Beginning with Version 8 of the Geiger Kit software, an over-threshold alarm is provided. When the CPM exceeds a set value, I/O pin A1 will be pulled high. It will go back to low (Gnd) when the CPM drops back under the alarm threshold. This alarm is optional, you can connect one or not.



The choice of final parts to use for your alarm is up to you. However, for testing your kit includes a simple push button (like the reset button) and a special “test” yellow LED. This LED has a resistor built in, so unlike a regular LED, it can be powered directly with 5V.

The push button connects with one side on ground and the other to I/O pin 10. You install the LED with the flat side or shorter lead to ground, and the other longer lead to I/O pin A1. The picture on the right illustrates the connection. Note how the switch is wired. The terminals on *opposite* sides are in common. (You can check continuity with your meter to be more clear.)

If you want to use an *audible* alarm you can connect a "piezo buzzer" to I/O pin A1 pin and ground instead of the LED. The piezo alarm must be the type with a built in oscillator such as http://www.radioshack.com/product/index.jsp?productId=2062397 or http://www.allelectronics.com/make-a-store/item/SBZ-40/1-12-VDC-PIEZO-BEEPER/1.html These are sometimes referred to as a “buzzer” rather than a piezo. Do exceed 40mA for any device connected to an I/O pin! (You will have to use the pin to trigger a transistor if you need more than 40mA.)

For setting the alarm, and switching to the second screen, you’ll want to use a better push button than the test button supplied. It is connected between ground and I/O pin 10.

The alarm threshold is configured during power up. After the first few screens you will see a "Set Alarm" prompt. You can let it time out (3 sec.) to keep the current setting, but pushing the button at this point start a list of alarm settings. Keep pushing the button (or hold it down) until find a setting you want, then pause to set that last value. After the 600 CPM setting, the choices repeat. The possible settings increment by 10 until 100 CPM and then by 50 until the 600 CPM setting. There is also a "No Alarm" setting. The chosen setting is retained in eeprom so it does not need to be set each time. See the ***Software*** page on the Geiger site for more information on the alarm.

Software Switches

These are notes about switches or jumpers that can be connected to certain I/O pins to make the software behave differently. In all cases, a no connection means "off" and connecting to ground (-) means the switch is "on".

CPM->uSv/h (Pin 9)

If you are using a LND 712 tube you should change to the built in rate (100 CPM = 1 uSv/h) for that tube. Connecting I/O **Pin 9** of the ATmega328 to ground will use that rate.

Alternate Display / Second Screen (Pin 10)

A push-button switch is connected between I/O Pin 10 and ground. When pressed once, the second screen will remain on until the button is pressed again. The first time the mode is entered, the totals will start fresh. After that, reentering the mode will display the totals that have accumulated.

The second screen shows values for 1 minute and 10 minute periods. While the period is in *progress* (time not reached) the accumulated counts for the period are displayed. (On the 10 minute line, the elapsed time is also shown on the right side.) When a given period is *complete*, the format of the display changes. It then becomes the running average of CPM and uSv/h for the last 1 minute and 10 minute periods (as in ver. 7). The alternate display is updated every 5 seconds.

Connecting I/O **Pin 10** of the ATmega328 to ground activates the display. The 10 minute count can be useful in testing food, for example. Comparing counts over this longer period may tell you more than the faster display of CPM.

This same switch is also used for setting the alarm threshold during startup of the Geiger Kit. See ***Connecting an Alarm*** for more details.

HV Test & Adjust

Please Note: ***There is a Japanese summary of these instructions on the website. (links also work there)***

This page will show you how to measure, test and adjust the HV section of the circuit. If you are happy with the way your board is working, you could skip this.

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 tiny current, and that's what it does. This is good because the battery will last longer - and it won't kill you! However, 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 10-11MΩ (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. A meter with an 11MΩ input impedance may read 214V when the voltage is closer to 420V. You need several gig-ohms (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 in series adds 90MΩ. If you want to a full gig-ohm of input impedance it's best to just buy a high value resistor - (example). Once you have the resistors added in series with the meter, you have to multiply the reading by some factor. The formula for this is

$$V_{actual} = V_{read} \times R_{meter} + R_{probe} / R_{meter}$$

So for example, my meter has an 11MΩ input impedance. If I use a 1000MΩ "probe" I have **11 + 1000 / 11 = 92.** This means that I multiply my reading by 92. If you built a 90MΩ probe for a typical 10MΩ meter, you'd have **10 + 90 / 10 = 10** so you multiply your reading by 10.

What's the difference between using a 90MΩ vs. a 1000MΩ probe? Here is what I saw:

460V with a 1000MΩ probe, 358V with a 90MΩ probe, and 222V with no probe (11MΩ meter).

Even with the 1GΩ probe, the HV may read a bit low. Testing against calibrated sources I now use a conversion factor of 100 instead of 92. There is more on this subject at http://ea4eoz.ure.es/hvprobe.html .

Don't sweat all this unless you want to! In the **Adjusting the HV** section below I describe a method that doesn't depend on being able to measure the actual HV. And by the way . . .

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

Testing the HV:

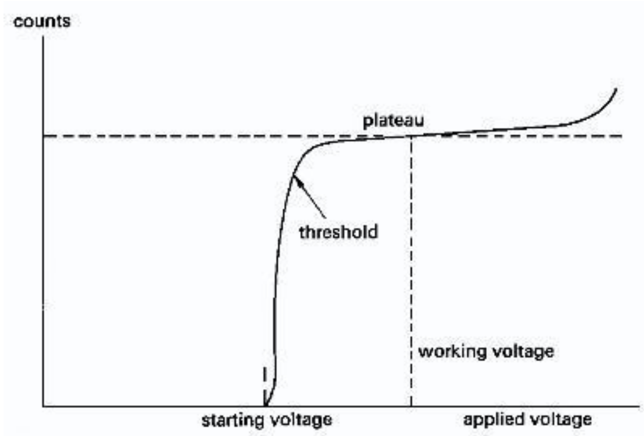
After building out the HV section of the board, you can see if things are working up to that point. First read through the next section to get a sense of the final picture. Then, using the 90M ohm probe described above, or even without it, connect the battery, and measure the voltage between the cathode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

You can test the click circuit by quickly shorting the tube wires across a resistance like a pencil line. (I use my finger, but I can't suggest that you do that.)

Adjusting the HV:

R5 controls the high voltage. (see Circuit Description on web site). It is preset for about 26Ω which should give you about 450V. This is about right for most tubes. Turning R5 clockwise will increase the voltage (by decreasing the resistance). However after a certain point, (~9.5 Ω) the circuit will crash, and the HV will fall off almost completely. If you want to adjust the HV, the idea is to adjust it so the voltage is *about* in the middle of the tubes operating range ("plateau" as shown below). Note that the Geiger should work fine with R5 set to its default, but it may be better for the tube (and battery) if you don't apply more voltage than you need.

Also remember that as long as the voltage in the operating range, the tube will have about the same sensitivity. The tube does not get more sensitive with higher voltages. **Put another way, once the tube is in its operating range, the HV you run at is not critical, and has very little effect on accuracy**.



So how do you do this? First let's look at the data sheet for two of the most common tubes. The readings in red are what I actually measured with my 1000MΩ probe and multiplying by 92. With a 90MΩ probe your readings will be different but the technique will be the same.

Tube	Initial Voltage (just get counts)	Operating Range	Recommended Voltage
SBM-20 spec	260-320	350-475	400
my readings for SBM-20	340	365-510	**430**
LND 712 spec	325 (max)	450-650	500
my readings for LND 712	440	475-675 (675 is over max HV)	**575**

To get the values for your meter do the following:
Using some kind of active source, lower the HV (CCW) until you get no response. (R5 is a 25 turn pot). Now slowly increase the HV (CW) until you *just start* to get clicks. Record the initial voltage. Now slowly increase the HV again until you are getting a good response from the source that doesn't seem to change as you go higher. Record the low end of operating range. By now you will have an idea how far your readings are from the data sheet.

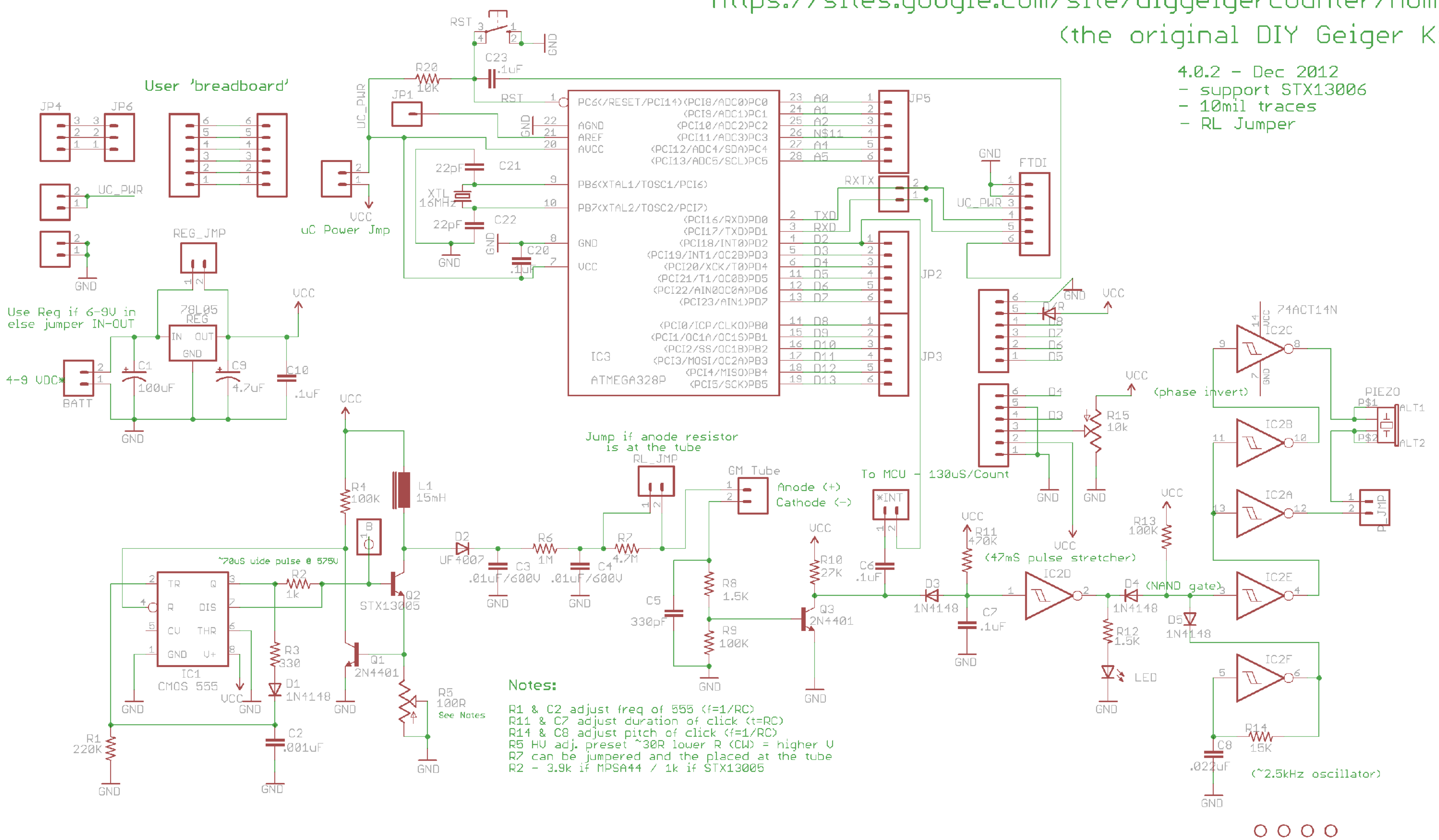
Notice the operating range for the tube. It's 125V wide for the SBM-20 and 200V wide for the LND 712. If your readings sort of followed the spec. you can assume about the same range and figure your high end of operating range. Now take the center of your operating range as your recommended voltage and set your pot to that. Finally, take two aspirins, and quit messing with it. It's not *that* critical, GM tubes have a wide range of operating voltage.

Geiger Kit – V4.0.2

https://sites.google.com/site/diygeigercounter/home

(the original DIY Geiger Kit)

4.0.2 – Dec 2012
- support STX13006
- 10mil traces
- RL Jumper



Notes:

R1 & C2 adjust freq of 555 (f=1/RC)
R11 & C7 adjust duration of click (t=RC)
R14 & C8 adjust pitch of click (f=1/RC)
R5 HV adj. preset ~30R lower R (CW) = higher V
R7 can be jumpered and the placed at the tube
R2 – 3.9k if MPSA44 / 1k if STX13005

Completed board. (large screen image here.)