Build Instructions for the Geiger Kit – v5.6 PCB

Getting Started

Congratulations! This kit is culmination of the experience gained in making Geiger Kits over the past several years. I hope you enjoy building it as well as using it. Try to take your time, and enjoy the journey.

Common Build Problems:

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

- **Parts orientation or wrong part used** - generally RTFM issues, and not referring to the “Notes” and pictures.
- **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.
- **Forgotten solder joints** - Particularly on headers and sockets after they are tacked in.

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 the part orientation and options as you go.
- Use the assembly 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.

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.

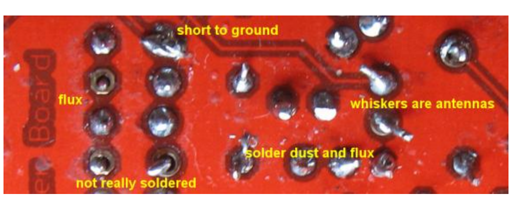
When you solder, start with a good iron, with a good tip that's freshly tinned. Solder the joint so that you have a nice round dot that stays inside the darker red. Do not use too much solder, and add enough heat for a good flow. The holes are plated through, so don’t worry about getting solder up to the top of the board. A “3rd hand” with a piece of solder in one of the alligator clips can be handy when tacking in IC sockets, etc.

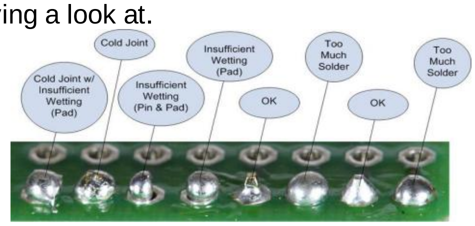
You will notice some pads will connect to the back plane. These have 4 little traces from the back plane to the hole, like a "+”. These pads will require more heat. I usually solder that side of the part last.

Do not use flux paste and avoid flux pens – especially in the HV area! Many will leave a residue that is slightly conductive. External fluxes can cause wacky problems. Simply use rosin core solder.

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.

Below is a picture of a terrible soldering job on the kit, and to the right are great examples from the Adafruit Guide to Excellent Soldering which I recommend having a look at.





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.

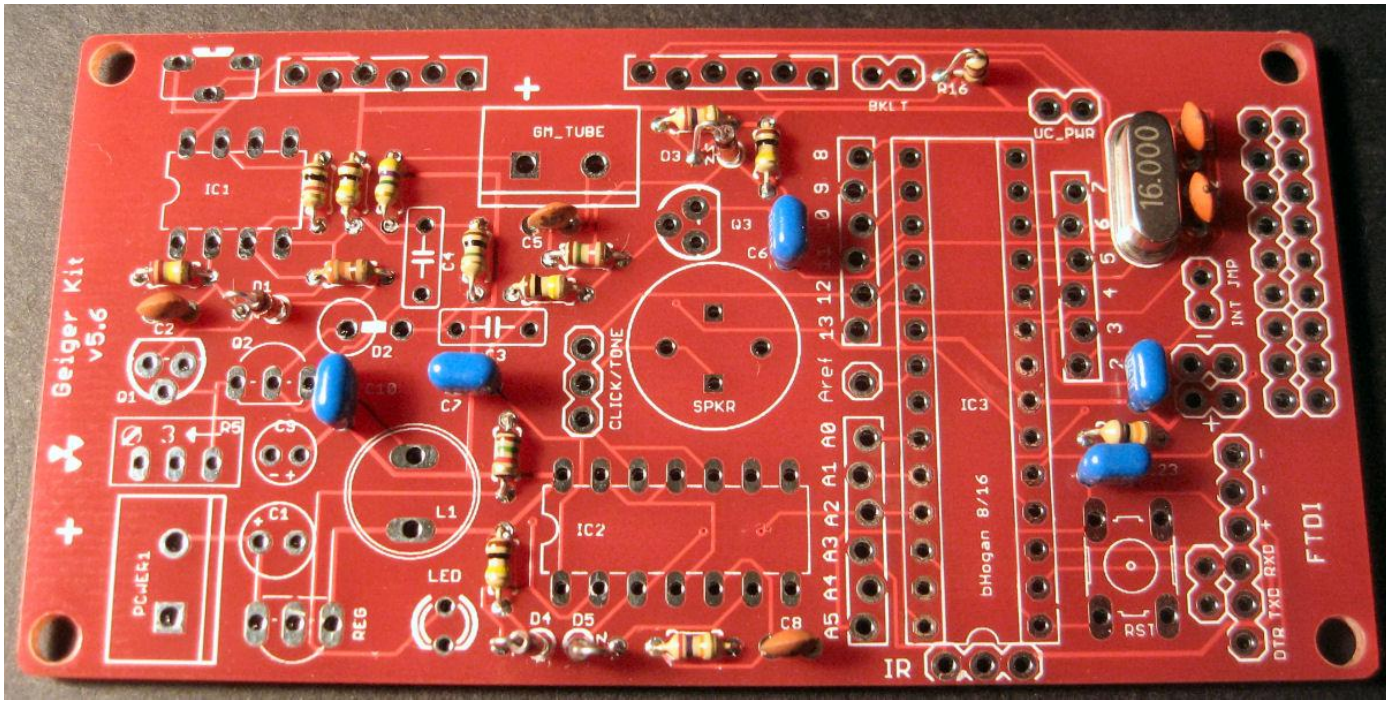
Step 1 - Building the Kit:

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 images on the page after the table to double check orientations of parts, etc.

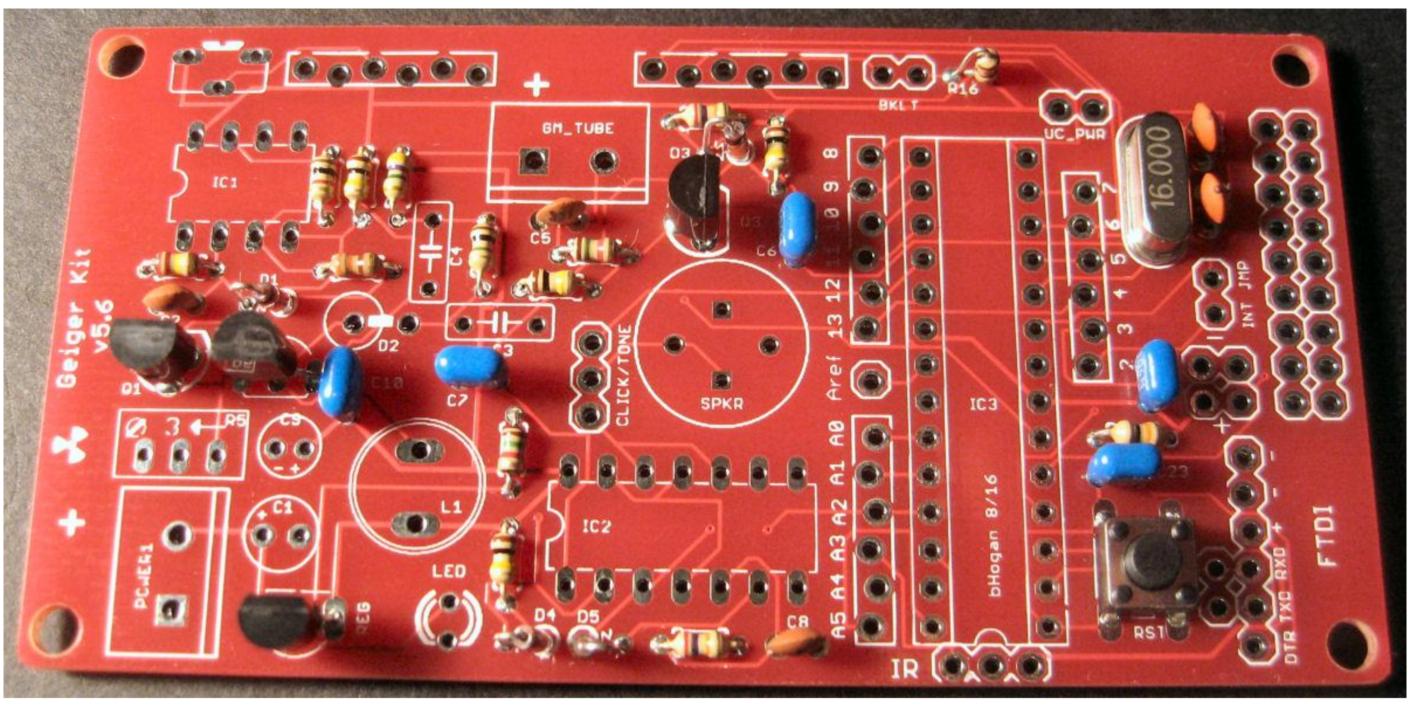
Build Sequence and Parts List for v5.6 Geiger Kits					
Ref #	**Qty**	**Value**	**Description**	**Notes** polarized? ->	Y N
PCB	1	v5.4	1.85" x 3.60" (~4.7 x 9.14 cm)	**Orientation:** "Geiger Board – Ver. 5.6" is the **left** side of board.	N
R1	1	220kΩ	RD,RD,YL (all fixed resistors are 1/8W)	**Color 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	Be sure last band is red – not orange	N
R3	1	330Ω	OR,OR,BN		N
R4, R9, R11, R13	4	100kΩ	BN,BK,YL	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	"R8" and "R9" look similar. R8 is above R9 - closest to GM_TUBE term.	N
R10, R14	2	27kΩ	RD,VT,OR		N
R20	1	10kΩ	BN,BK,OR	Be sure last band is orange – not red	N
C2	1	.001uF	#102 (1nF) ceramic cap	On rare occasions the markings may be partially worn off this cap.	N
C5	1	330pF	#331 ceramic capacitor	(right below the GM_TUBE term)	N
C8	1	.022uF	#223 (22nF) ceramic cap		N
C21, C22	2	22pF	#220 / #22 ceramic cap		N
C6, C7, C10, C20, C23	5	.1uF	#104 ceramic capacitor		N
OSC	1	16MHz	crystal		N
socket	1	8 pin	IC socket	**Note:** the 3 sockets are stacked together. Separate them. 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). The body will go in the hole with the white circle. **See pictures below.**	Y
R16	1	150Ω	BN,GN,BN (stands up)	Located to the right display header. **See pictures below.** For LCD backlight. 150Ω uses minimum current with good brightness. **To add a switch for the backlight, see *Appendix IV*.**	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 mounts without bending the center lead back. **See pictures below.**	Y
REG	1	L4931CZ50	5V / 250mA LDO reg	Spread leads if necessary. **Can substitute Pololu Reg – see *Appendix II*.**	Y
switch	1		push button	**"polarity:"** follow lead spacing. Snaps in and lays flat. **Trim leads**	Y
C3, C4	2	.01uF	#103 HV ceramic cap		N
R15 (not labeled)	1	vertical 10kΩ	1 turn pot for LCD	**polarity:** white adj. screw toward top of board **Set the pot fully CCW then back CW about 1/8 turn so to see the display when you first power on.**	Y
D2	1	UF4007	1000V 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 pictures below.**	Y
C9	1	4.7uF	50V electrolytic capacitor	**polarity:** "-" stripe to the left.	Y
C1	1	100uF	16V electrolytic capacitor	**polarity:** "-" stripe to the right.	Y
header	5	6 pin fem.	I/O pins & LCD	**Suggestion:** tack in all 5 headers, then solder them in all pins at once.	N
header	1	2x2 female	"user power header"	Supplies 2x Vcc and 2x Gnd pins for powering add on devices	N
header	1	3 pin fem.	IR sensor socket	Installs at bottom edge of 28 pin socket for IC3.	N
header	1	6 pin 90°	FTDI header	To connect FTDI dongle for serial output or programming.	N
2 pin male headers	3 ea	with jumper blocks	See ***Jumpers*** section	**Suggestion:** Put the jumper block on each header before soldering - (easier on fingers ;-). Leave the jumpers in place.	N
3 pin male header	1	with jumper	Click / Tone select for speaker	Put the jumper on the lower 2 pins for CLICK for initial use. See ***Jumpers*** section.	N
speaker	1	AC1205G	no paper seal 40Ω	**Note:** install with pins horizontal in horizontal pads. Polarity is not critical.	N
screw term	2	2 pin	5 mm pitch	**Suggestion:** if you trim the leads don't use your precision cutters for this.	Y
R5	1	100Ω pot	blue 25 turn HV pot	**Note:** pot is preset to ~26Ω for ~450V **polarity:** adj screw on left	Y
L1	1	15mH	inductor	Orientation shouldn't matter but maybe best if installed as pictured	?
LED	1	red	3mm	You can mount this directly on the board, or 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 or dot 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. (**Very important with this chip!** Else it will get hot and let out the magic smoke!) **Option** - if removed stops click and led - interrupt still sent to uC.	Y
IC3	1	ATmega328P	AVR microcontroller	Bend all pins inward a bit on a flat surface. (I may have done this for you.) **polarity:** Notch down	Y
IR sensor	1	38kHz	similar to TSOP4838	Mounts in 3 pin female header. **polarity:** Bulge faces "down" **See picture.**	Y
6 pin male header	2	6 pin	LCD Headers	These male headers are soldered to the LCD as described in ***Step 4***.	N
fuse clips	2	¼"	for HV conn. to GM tube	Reform as needed. Solder stranded wire to these. **Tube is polarized**	Y
alarm piezo	1	piezo w/ osc	has paper seal on top	Gift - to be used for the alarm. **see *Appendix IV* for use**	Y

Assembly Images: (For clarity these do not follow assembly order.)

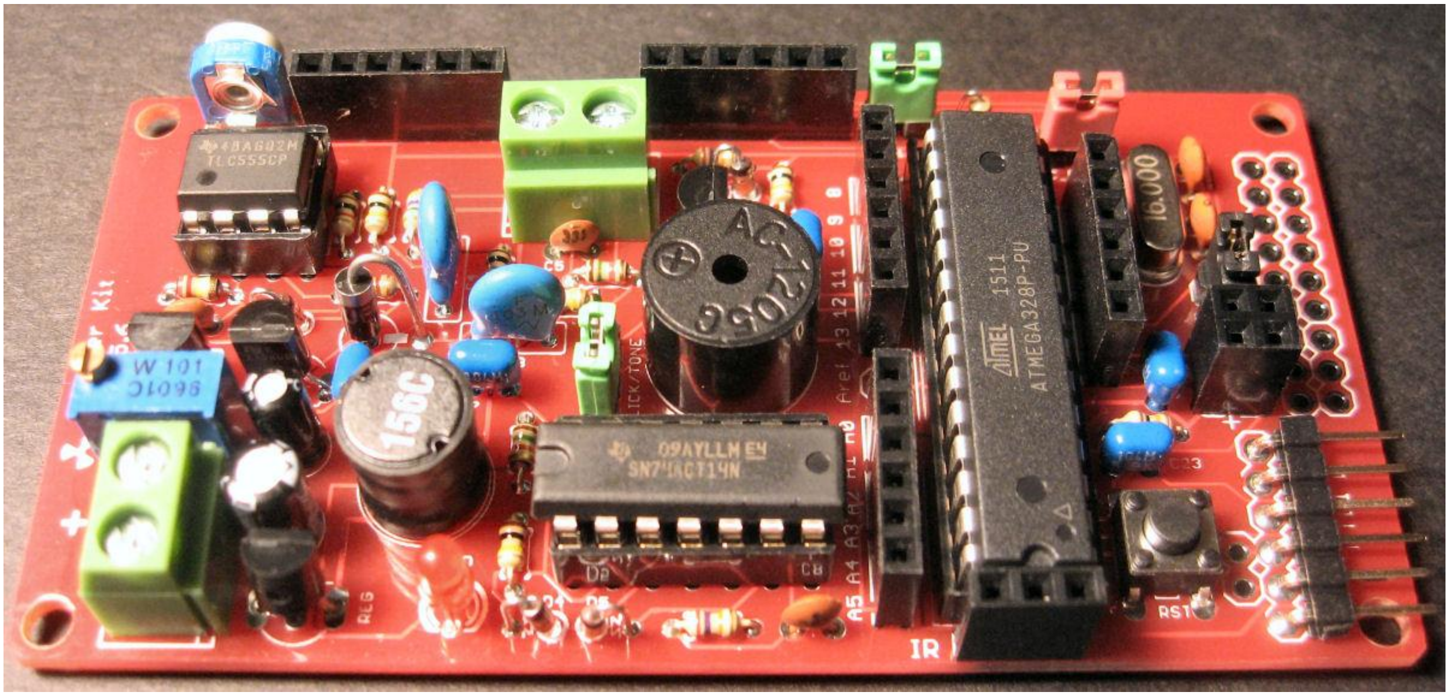
Small components added – note diode positions.



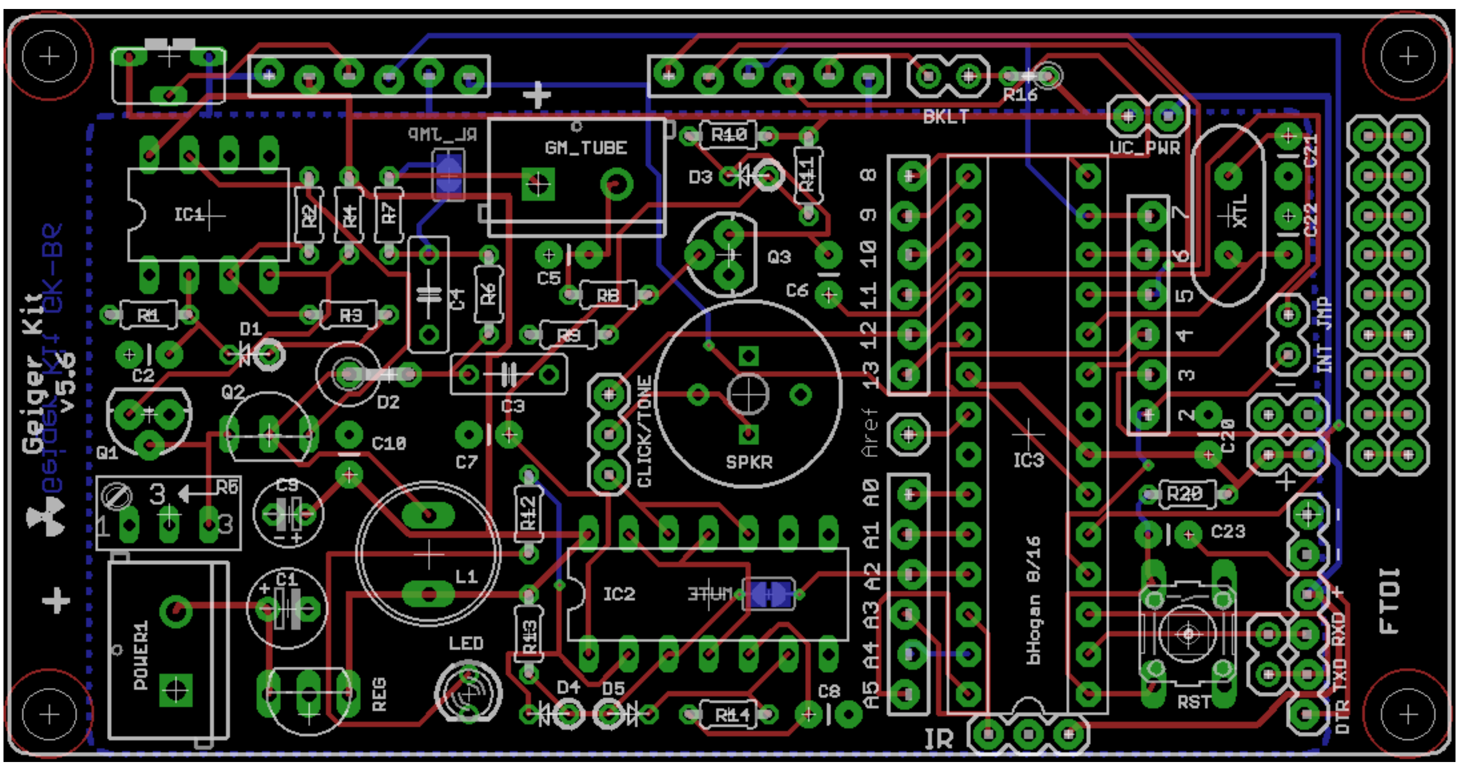
Board built up a little further . . .



completed board . . .



PCB layout . . .



Step 2 – Inspect and Clean 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 main board with an old toothbrush to remove any solder dust when you're finished soldering.

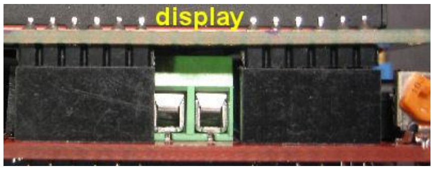
If you want to remove excess flux, one of the simplest ways is to use alcohol – the kind from the hardware store - and a toothbrush. Be sure to blow off the water that is created or at least let it dry well before powering up the board. However, there are better solvents like commercial flux removers. Just be careful not to use something that removes more than the flux! Some water soluble flux pens like the Kester #2331-ZX can leave a conductive film. If you used a flux pen like that, be sure to thoroughly clean the areas where it was used.

Step 3 - Configure the jumpers:

Make sure that the UC-PWR, INT_JMP, and BKLT jumpers are installed and that the CLICK/TONE jumper is in the lower click position. There are also solder jumpers on the bottom of the board which are not normally used.
Please refer to ***Appendix I - Jumpers*** for more information.

Step 4 – Add the LCD:

Two 6 pin male headers are included in the parts bag. Solder these headers starting from each end of the pads on the LCD. This leaves four pads in the center free. You can extend the wires to the LCD when you put it in a case – see ***Appendix V – LCDs and Wiring***. depending on the display, you may want to bend the center lower tab on the back of the LCD just slightly if it doesn't clear the speaker.



Step 5 – 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 100kΩ. (I just use my finger, but can't suggest that you do that.) Note that due to the very low current supplied you do not need any special wire for the GM tube. (24AWG silicone stranded core wire is nice to use though.)

Step 6 - Power up the board:

Decide on how you will power your Geiger Kit. Refer to ***Appendix II – Powering the Geiger*** to help you decide. Observe polarity. It should click once. You will need to adjust the contrast pot for the LCD (usually almost fully counter clockwise) before you can read the display.

If you hear clicks congratulations! Now you might look at ***Appendix III – HV Test and Adjust***. However, the HV should already be adjusted to ~450V which will work with most tubes.

Step 7 – Understanding the features:

See the **Geiger Kit - GK-B5** page for a link to the **User Guide** for your software release. It provides information on how to setup and use various features of the kit. Some of them are "optional" meaning that to add them you must supply some of your own components such as switches, etc. Also see ***Appendix IV – Wiring the External Controls*** here for a wiring diagram of the switches, buttons, and indicators that you may want to add.

Enjoy the kit!

Appendix I - Jumpers:

There are 4 jumpers on the top of v5.6 board and 2 solder jumpers on the bottom. Normally all four jumpers on the top of the board have a jumper block installed and the solder jumpers are not modified.

CLICK/TONE **(normally jumpered to CLICK)**

This is a 3 way jumper for the speaker. With the jumper on the "click" side of the two pins you hear the classic Geiger click sound. When the jumper is on the "tone" side, it is set for "Tone Mode" where activity is indicated by the pitch of the sound. If you put the kit in a case, you can wire these 3 pins to an ON-OFF-ON switch. The speaker will be off when the switch is in the center position. The LED will continue to flash with the speaker off.

INT_JMP **(normally jumpered)**

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 our use the kit as a development board.

UC_PWR **(normally jumpered)**

When this jumper is removed you can run the Geiger from one source, and the microprocessor from another. (Keeping the grounds common.) You would power the Geiger circuit from the screw terminals, and power the ATmega328 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 occasionally with no damage).

A nice way to take advantage of this jumper is by controlling the HV from your sketch. Simply remove the jumper and run a wire from an output pin you are controlling in software, 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.

Note that when the microprocessor is powered off, and the MUTE_JMP is not cut (see below) the sound from the speaker is diminished. If you want to conserve battery power by shutting off the microprocessor, you can sacrifice the IR mute ability and not jumper the MUTE_JMP.

BKLT **(normally jumpered)**

This jumper connects the LCD backlight to power. If you want to have a Backlight ON/OFF switch on your case, simply remove this jumper and add your switch across these two pins. If you have a display with a green background and black characters you can still read the display with the backlight off. Running without a backlight saves about 6-8mA.

MUTE_JMP **(solder jumper on bottom – normally closed)**

By default, this jumper allows the MUTE button on the IR Remote to mute the click sound coming from the speaker. (The MUTE button will always mute the tone sound.) The only reason to cut this jumper is to have a normal click sound when the microprocessor is not powered on.

RL_JMP **(solder jumper on bottom – normally open)**

Shorting, this jumper bypasses (shorts out) R7 which is the anode ("load") resistor for the GM tube. Normally this jumper is not shorted. It is only used if you will be adding the anode resistor directly at the GM tube. With this jumper in, the 4.7mΩ 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 (not shorted) or at the tube. Running a GM tube without an anode resistor may cause the tube to avalanche and shorten its life.

Appendix II - Powering the Geiger

Onboard Voltage Regulation:

The onboard voltage regulator allows for a 20V input and is capable of a maximum of load of 300 mA. It has very low dropout of 0.4V.

Power Consumption:

The current drawn by the kit depends on several factors such as the HV setting, display type, and CPM. The current measurements below were taken on this version of the PCB.

- **HV = 450V (as shipped), background CPM ~32mA** *(at 12,000 CPM add ~7mA) (at 940V HV max HV add ~16mA)*
- **with backlight off ~24mA**
- **with display removed (i.e. a balloon flight) ~22mA**
- **with microprocessor off (UC_PWR jumper open) ~5mA**
- Note: Tone mode increases power used by 30-40mA.

Powering with AA Batteries:

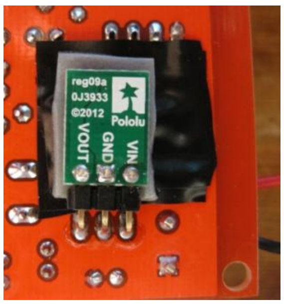
This is a common way to power the kit. The batteries can be configured so that they will supply about 5V. I prefer to run with 4 NiMH since they can provide ~2000mAH of power at close to 5V. A 9V battery is generally not suggested since they only supply ~600mAH. This would only supply about 16 hours of use. However they can be used during testing.

When running under about 4.5V, the voltage should be stepped up – see the LiPO section below. If the kit is powered with Vcc below about 4.2V the HV circuit will work, the display will work (but will need more backlight), and the click sound will be quieter.

Powering with LiPO Batteries:

LiPO batteries typically supply 3.7V so a step-up or booster module must be used to bring this voltage up to 5V. 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. I suggest the Pololu #2119. It takes an input of between 2.7- 11.8V and can output at least 500mA at 5V with a typical efficiency of 90%.

The Pololu regulator replaces the voltage regulator in the kit. It can be mounted vertically on the top of the board in the same pads the standard voltage regulator uses. When mounting this way, the parts side of the Pololu faces toward the bottom of the board.



Another option is mount the Pololu regulator on the bottom of the board. as shown in the picture on the left. Again, it is soldered to the same pads as the voltage regular supplied with the kit. To keep a low profile, the 90° header supplied has another 90° bend on the long pins soldered to the part. If you want to do this, make sure it is orientated as shown, and that it is well insulated (good electrical tape, or heat shrink). You can secure it with double sided tape as shown.

In either case, be sure it is orientated correctly. It does not have reverse voltage protection and you may let out the magic smoke.

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, you should disconnect the battery. The FTDI cable will then power the whole board just fine.

Appendix III - HV Test & Adjust

There is a Japanese summary of these instructions here.

This section will show you how to measure, and adjust the HV section of the circuit. If you are happy with the way your kit is working, you could skip this. You can do a quick test of the HV and click circuit by quickly shorting the tube wires – preferably across a high resistance. (I use my finger, but can't suggest that you do that.) If you hear clicks, or the speaker screams, your HV is probably OK.

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 typical DVM will put a load on the circuit it's measuring of about 10MΩ. 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 a 10MΩ input impedance may read 214V when the voltage is closer to 420V. You need at least a 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 you get. Adding 9 10MΩ resistors in series adds 90MΩ. If you want a full gig-ohm of input impedance, it's best to just buy a single 1GΩ 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:

$$V_{\text{actual voltage}} = V_{\text{reading}} \times (R_{\text{probe}} + R_{\text{meter}}) / R_{\text{meter}}$$

So for example, if you built a 90MΩ "probe" for a typical 10MΩ meter, you'd have **90 + 10 / 10 = 10** so you'd multiply your reading by 10. If you used a 1 GΩ resistor (1000MΩ) with the same meter it would be **1000 + 10 / 10 = 101** so you'd multiply your reading by 101.

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

222V with no probe (10MΩ meter), 358V with a 90MΩ probe, and 460V with a 1000MΩ probe.

Adjusting the HV:

The blue pot (R5) controls the high voltage. (see Circuit Description on web site). It was preset to 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, the circuit will crash, and the HV will fall off almost completely. In general, the kit can produce HV from 50->1100V.

The high voltage is best measured from the cathode (band side) of D2 and ground. Connect your meter to a ground on the board, and if using the 90M ohm probe described above, put it in series with the positive probe of your meter. Touch the other end on D2. It's OK to measure without the 90M ohm probe you will just get low readings and maybe a whine from the speaker. If you get something like 200V without the probe - congratulations! The HV circuit is working.

You can get an *approximation* of the HV by measuring the resistance of R5. (An easy way to connect to R5 is one probe to ground and the other to the base of Q1 or emitter of Q2.) The chart below shows the HV at various resistances of R5. Voltage was measured with a 1 GΩ probe. Note however, that since small changes in resistance make large changes in voltage and the specs on the individual parts may vary it is only a guideline.

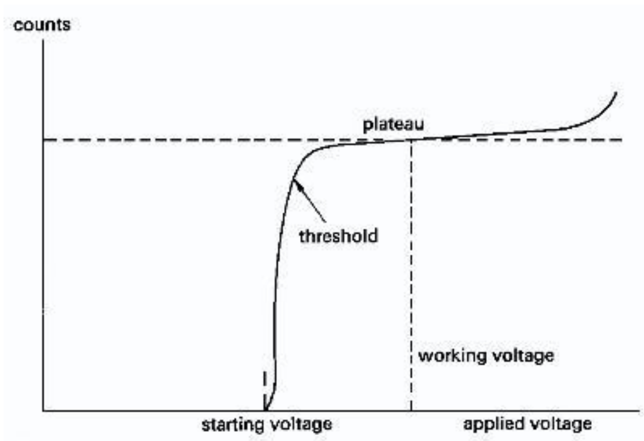
~HV	Ω at R5	~HV	Ω at R5	~HV	Ω at R5	~HV	Ω at R5
84V	100	400V	27.1	575V	20.0	800V	15.1
100V	91.8	425V	25.7	600V	19.2	900V	13.7
200V	49.9	450V	24.5	625V	18.6	950V	12.0
300V	34.7	475V	23.4	650V	18.1	unstable	<12.0
325V	32.4	500V	22.4	675V	17.4	HV crash	6.0
350V	30.4	525V	21.6	700V	16.9		
375V	28.5	550V	20.7	750V	16.0		

Setting the HV:

While the Geiger should work fine with R5 set to its default, it may be better for the tube if you don't apply more voltage than you need. Each type of GM tube has its own operating voltage range. If you know it you can just set the HV to middle of this range by one of the methods above.

However sometimes the operating range is not specified, or it may have changed with age, or you simply want to set the HV "dynamically" and not by a measured HV. This section describes a method that doesn't depend on being able to measure the actual HV.

The idea is to adjust the HV so that it is in about in the middle of the tubes operating range – this is the "plateau" as shown below. Within this plateau the tube will have about the same sensitivity regardless of the voltage. **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 counts**.



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 101. With a 90MΩ probe your readings will be different but the technique will be the same.

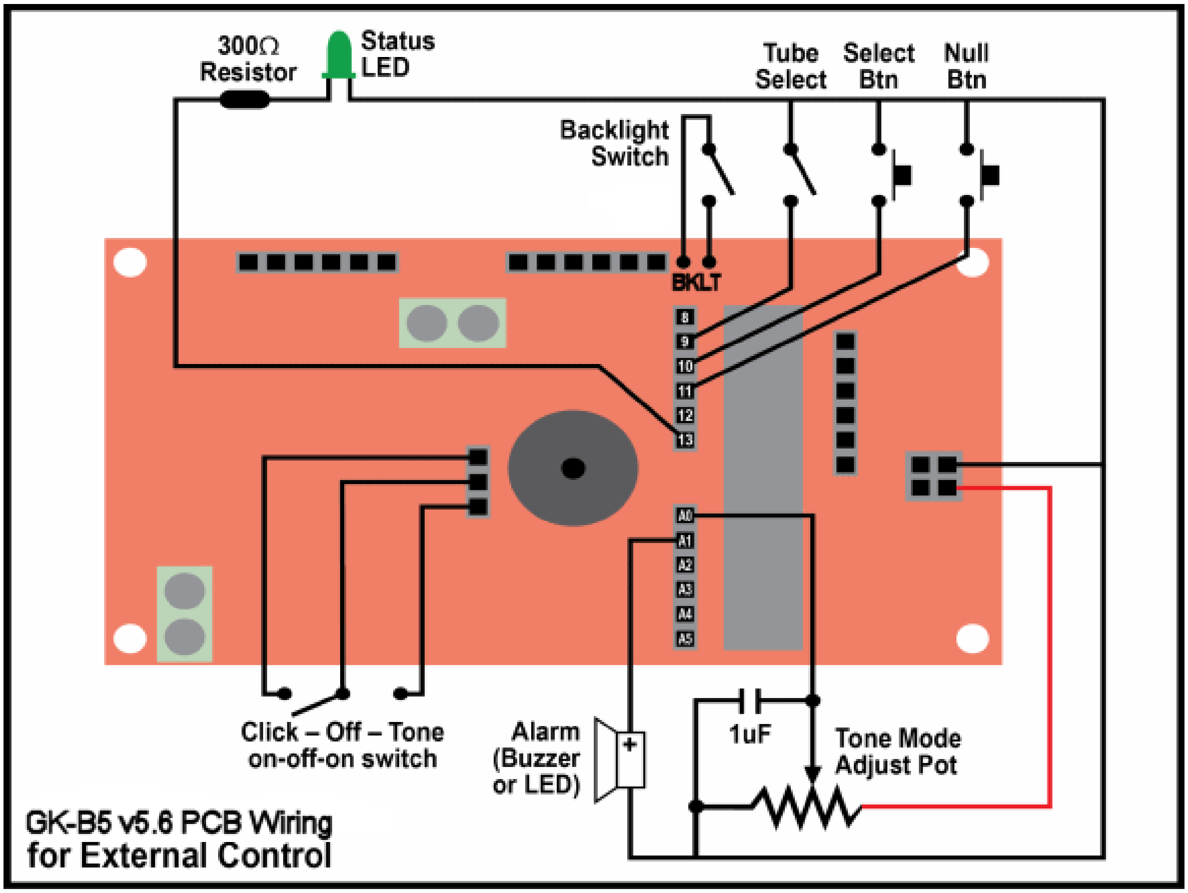
Tube	Initial Voltage (just get counts)	Operating Range	Recommended Voltage
SBM-20 spec	260-320	350-475	400
my readings for SBM-20	340	365-510	**430 calc.**
LND 712 spec	325 (max)	450-650	500
my readings for LND 712	440	475-675	**575 calc.**

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 (475-350) 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.

Appendix IV - Wiring the External Controls:

The diagram below shows how to connect the switches, buttons, etc. that are supported by the current board and software. Note that the controls you choose add are up to you.



Note on the Backlight Switch - Wiring a switch to control the backlight depends on the version of the PCB you have. Beginning with PCB v5.6 you simply remove the jumper and wire in a switch as shown.

Appendix V – LCDs and Wiring:

Selecting an LCD:
An LCD display is included with the kit but you may want a certain color or size - i.e. 2x8, mini, etc.

When selecting an LCD the main requirement is that it uses the extremely common HD44780 (or compatible) chipset. It should also be a 5V display (again, most are). Most have a backlight. If they do, some require a current limiting resistor for it. See the specs for your particular LCD.

A consideration might be the ability to still read the display with the backlight turned off. (A switch to turn off the backlight can save about 8mA.) For this, you want a "transflective" display that shows the characters in black against the background - usually green. These might not look as "cool" as other LCDs, but they can be read with the backlight switched off. Here is an example of this type of display.

There are many places to get an LCD such as Sparkfun, Adafruit, etc. There are also many listed on eBay.

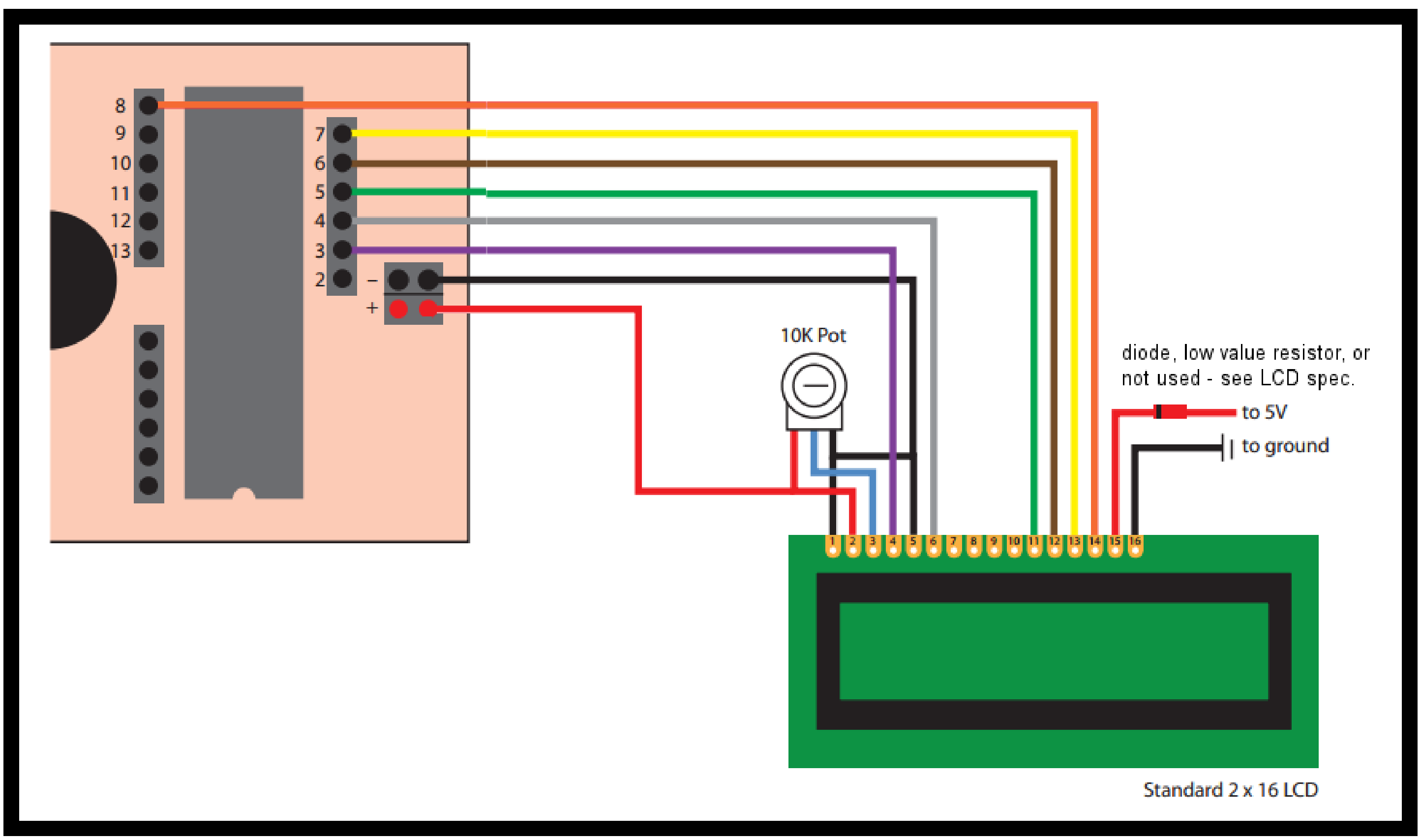
Other LCD Types:
Something a little smaller than a standard display is this New Haven mini display. It can also be read without the backlight. Unfortunately, the pins are on the bottom of the display so it hangs over the top of the board on the GK-B5 kit.

Finally, with a software change you can use a DOGM character display . These are small, very bright, and offer a lot of options. On the other hand they generally require more power when the backlight is on, and are somewhat expensive. If you are interested in this display, see this project.

Moving the LCD off the board:
On the GK-B5 kit, the LCD simply plugs right in and the contrast pot and the backlight resistor are already on the board.

The simplest way to extend the LCD to your case is to wire 1:1 from the display headers on the board to the LCD. You can use 90° male headers to plug into the display headers and on the LCD side, either use female headers or solder directly to the LCD.

For some (rare) situations you can also connect the LCD directly to the appropriate I/O pins. In that case you will need to supply the contrast pot and the backlight resistor. The following diagram shows those connections.



Appendix VI – Schematic