Build Instructions for the GK-Plus v2.2 PCB Rev2b - part change

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.

Please Note: *When removing the TFT display from the board, do not pull it by the white plastic edge of the display itself. Grab it by its red carrier board. Otherwise you may loosen the display from the carrier board. If you do loosen it you can secure it again with a few drops of gel superglue or hot glue. Be careful not to get glue under the display however.*

Common Build Problems:

These are the most common problems builders have had . . .

- **Parts orientation wrong** - 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** (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* 3-4 hours to build this kit. Solder the right part, the right way, the first time. Parts are hard to unsolder.

Soldering:

Use a good iron, with a tip that's freshly tinned. Solder the joint so that you have a nice flow between the lead and the pad. 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.

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.

Do not use any flux paste or 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. The only exception is for IC4, the surface mount part. If you use a flux pen there, clean the area afterwards.

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 may not do any board repair if lead free solder was used.

Soldering Surface Mount Parts:

There are two SMD (surface mount) parts - IC4 and the battery holder. IC4 is not a super fine pitch component, but soldering SMD uses a little different technique. There are lots of tutorials on this subject on the web. Sparkfun has a good one. I’ve found that a flux pen makes the solder flow nicely and avoids bridging. I dab it on the pads before soldering and then over the pins and pads after tacking down a couple of pins. Note that some water soluble flux pens like the Kester #2331-ZX can leave a conductive film. If you use a flux pen like that, clean the area with alcohol after soldering.

Build Sequence

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

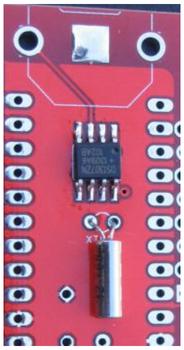
Step 1 - Building the kit:

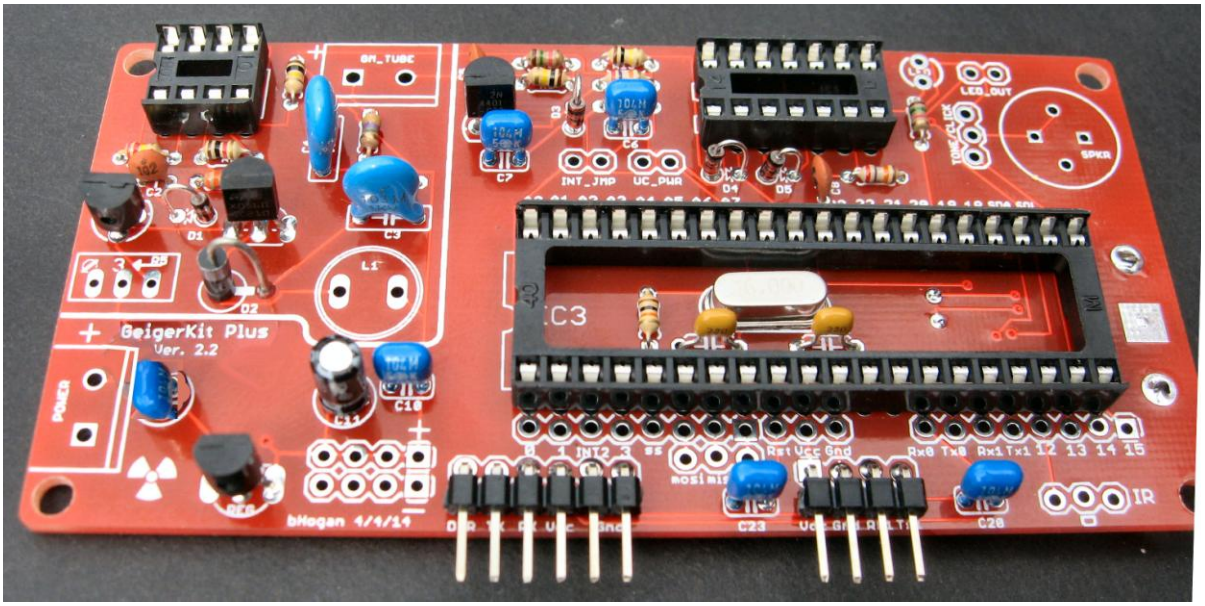
Use the tables on the next pages as your guide. The approach taken is to build the board by height – starting with the shortest components. It's easier to work on a board that lays flat and holds the parts in place when you flip it over to solder. Personally I find this technique faster than using a vise and spreading the leads to hold the part while it is being soldered. First you will build the main board, then, optionally, the display adapter board. While working, refer to images on the page after the table so you can double check orientations of parts, etc.

Build Sequence for v2.2 GK-Plus – main board					
Ref #	**Qty**	**Value**	**Description**	**Notes** polarized? ->	Y N
PCB	1	v2.2	1.95" x 3.90" (~5 x 10 cm)	**Orientation:** Rad symbol on **lower left** side of board.	N
start with the 3 components that mount on the bottom of the PCB . . . (See picture below)					
IC4	1	DS1307	RTC	**SMD part - polarity:** notch towards top. **See pictures.**	Y
XTL2	1	32.768kHz	watch crystal	leave leads long enough to bend over and lie flat - solder from top. **see pic**	N
BATT Holder	1	12mm	for CR1225 battery	**Before soldering** – cover the center contact with solder to a *slight* bulge. This makes better contact with the battery. With the open side towards the right side of the board, solder the two tabs that fit in the holes in the board.	Y
solder the rest on top . . .					
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, use magnifying glass or 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		N
R6	1	1MΩ	BN,BK,GN		N
R7	1	4.7MΩ	YL,VT,GN	**Note:** *Anode Resistor* – Value good for most tubes.	N
R8, R12	2	1.5kΩ	BN,GN, RD		N
R10, R14	2	27kΩ	RD,VT,OR		N
R20	1	10kΩ	BN,BK,OR	Be sure last band is orange – not red (mounts under IC3)	N
header	1	6 pin 90°	FTDI header	To connect FTDI dongle for serial output or programming.	N
header	1	4 pin 90°	GPS	**Note:** you can omit if no GPS or wired in.	N
OSC	1	16MHz	crystal	(mounts under IC3)	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		N
C8	1	.022uF	#223 (22nF) ceramic cap		N
C21, C22	2	22pF	#**220** / #22 ceramic cap	These mount under IC3 - mount close to board	N
C6, 7, 10, 20, 23	5	.1uF	#104 ceramic capacitor	blue in color	N
C1	1	.1uF	#104 ceramic capacitor	blue in color (Note: silkscreen is different than other .1uF caps above)	N
C11	1	2.2 or 3.7uF	50V electrolytic capacitor	**polarity:** "-" stripe to right	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 circle. **See pictures.**	Y
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	40 pin	IC socket	notch on left - **Note**: check that no pins are bent under before soldering.	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	STX0560	NPN HV transistor	Spread leads as necessary.	Y
REG	1	L4931CZ50-AP	5V / 250mA LDO Voltage Regulator	Spread leads if necessary. A step-up/step-down regulator can be fitted instead - see ***Appendix II – Powering the Geiger***	Y
C3, C4	2	.01uF	#103 HV ceramic cap		N
D2	1	UF4007	1000V 1A Ultra Fast diode	**polarity:** Bend over the lead on the **banded** side (cathode) anode goes into larger circle, cathode goes into the hole on the right. **See pictures.**	Y
header	1	2x4pin fem.	"user power header"	solder to **+** and **-** pads **Suggestion:** Tack headers in first, then solder all.	N
I/O header	1	11 pin female	I/O pins	Header for I/O pins on **lower left** - "0" to "gnd".	N
headers	2	3 pin fem.	"IR" + "mosi, miso, sck" header	IR header is also on Display Adapter so it's optional here. The "mosi, miso, sck" header goes in front of 11 pin header, is also optional.	N
I/O headers	3	8 pin female	I/O pins	Headers for other I/O pins - "Rx0" to "15", "23" to "SDL", and "A0" to "A7"	N
header	1	2 pin fem.	LED Out	Passes LED signal to Display Adapter board	N
2 pin male headers	2 ea	with jumper blocks	See ***Jumpers*** section	**Suggestion:** Put the jumper block on the header before soldering (easier on fingers ;-).	N
3 pin male header	1	with jumper	Click / Tone select for speaker	Put the jumper on the upper 2 pins for CLICK for initial use. Remove jumper if using Display Adapter. See ***Appendix I – The Jumpers.***	N
speaker	1	**AC1205G**	**Does not have paper seal!**	Install in vertical pads. (+ side up). **Be sure to use the right one!**	Y
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 ~21Ω for ~450V **polarity:** adj screw on left	Y
L1	1	15mH	inductor	**Trim leads**	N
LED	1	red	3mm	**Note:** This part is also duplicated on the Display Adapter. **polarity:** Small flat on side, **or shorter lead**, goes up.	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	Schmitt trig. hex inverter	Bend all pins inward a bit on a flat surface. **polarity: Notch** (not dot) on left. (Polarity important - else it gets hot and lets out the magic smoke!)	Y
IC3	1	ATmega1284P	AVR microcontroller	Bend all pins inward a bit on a flat surface. (I may have done this for you.) **polarity:** Notch on left	Y
IR sensor	1	38kHz	similar to TSOP4838	Mounts in 3 pin female header. **polarity:** Bulge faces "down" **See picture.**	Y
battery	1	CR1225	for RTC	slide into battery holder on bottom + side away from board	Y
SD header	1	4 pin male	on TFT Display	Solder this header to the SD side of the TFT display	Y
fuse clips	2	¼"	for HV conn. to GM tube	Reform as needed. Solder stranded wire to these. **Tube is polarized**	Y
alarm	1	piezo w/ osc	has paper seal on top	For Display Adapter, or use as needed to add the alarm to your project.	Y

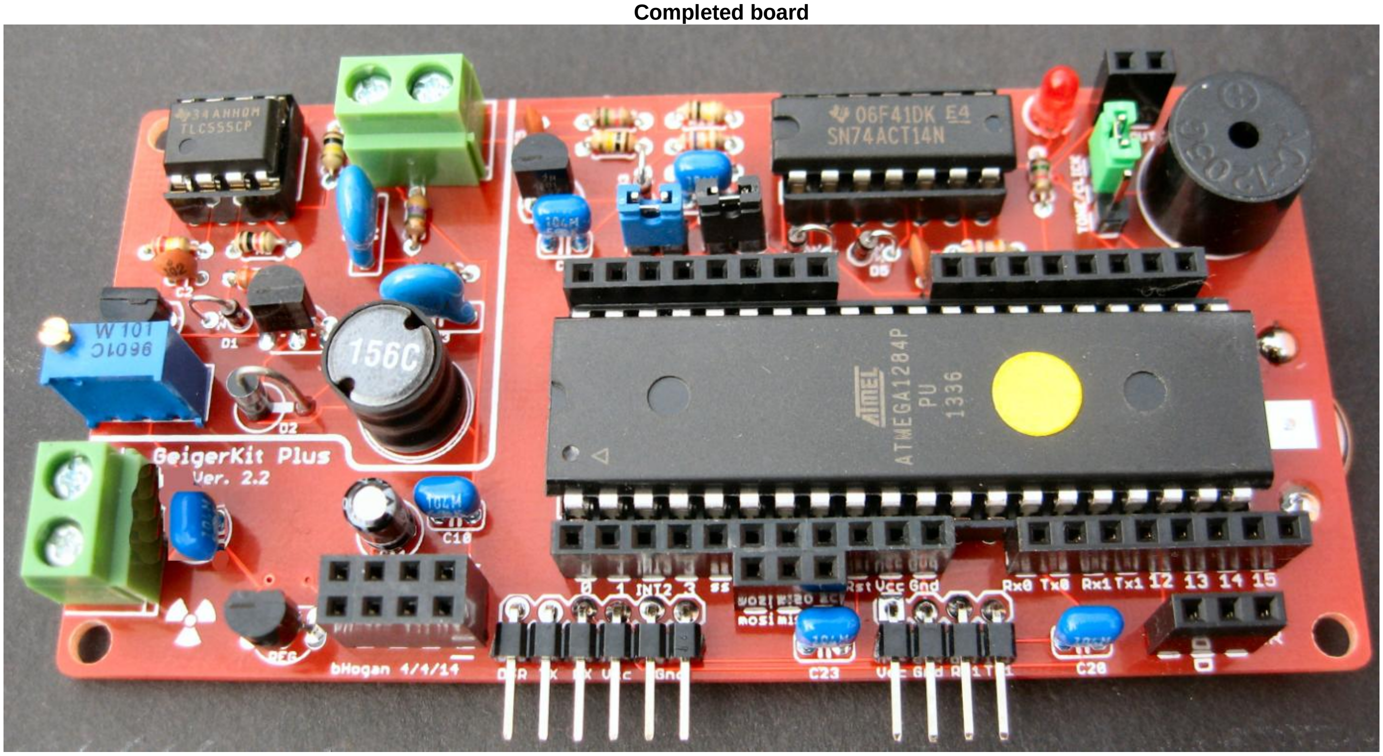
Assembly Images – main board:

Small components added – note diode positions.

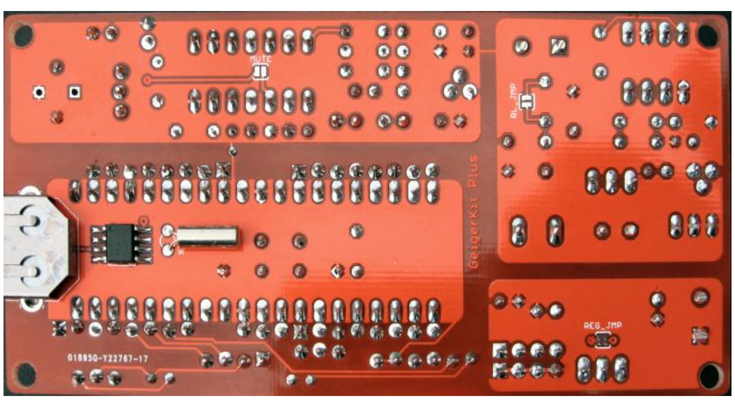


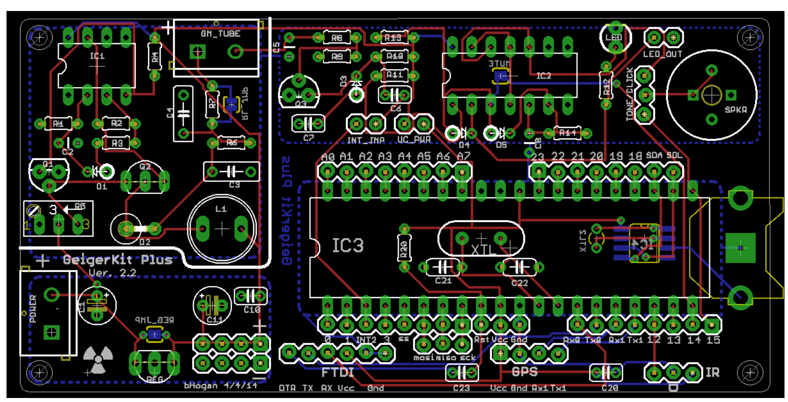


Completed board



v2.2 PCB layout . . . (larger image in browser here)





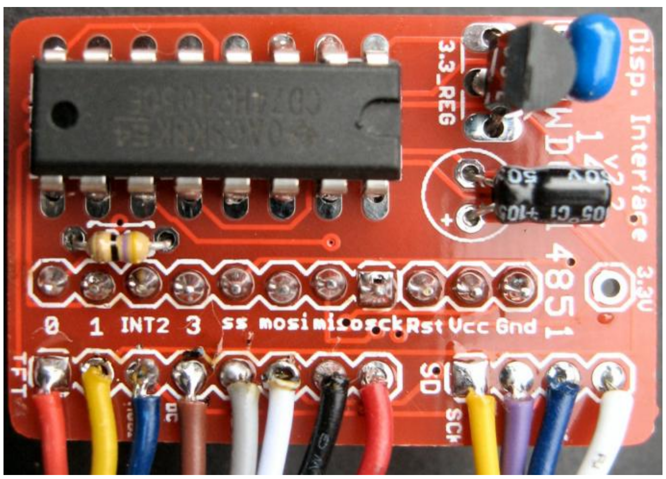
Step 1 - Building the kit - continued: (Display Interface Board)

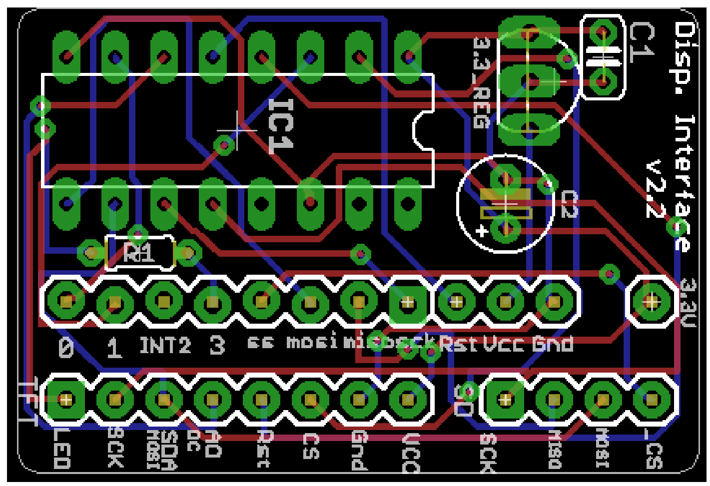
At this point you should have a working Geiger counter with no display. Generally you should go on and build this Display Interface board.

The Display Interface is a small board that plugs into the 11 pin “0 – Gnd” header on the main board. Its purpose is to interface the 5V board to the 3.3V TFT display. (While it’s possible to run the display on 5V, it’s not recommended.) Pads on the other side of this board are used to solder the wires that will go to the display.

If you’re going to build the optional **Display Adapter** you don’t need to build this Display Interface Board right now since the Display Adaptor also provides the level shifting needed for the display. But eventually you *may* want to switch to the Display Interface board when it comes time to put your kit in a case. It’s a simple build . . .

Build Sequence and Parts List for v1/v1.1 Display Interface board					
Ref #	**Qty**	**Value**	**Description**	**Notes** polarized? ->	Y N
PCB	1	v2.2	1.43”x2.06” (~3.6 x 5.3 cm)	**Orientation:** IC1 on top left.	N
R1	1	47Ω	YL,VT, BK	extra safety to limit backlight current at > max setting	N
IC1	1	**CD74HC4050E**	hex buffer for level shift	Bend pins in a bit on a flat surface. **polarity:** Notch right. Not socketed.	Y
C1	1	.1uF	#104 ceramic capacitor	blue in color	N
REG	1	78L33	3.3V Regulator	Spread leads if necessary.	Y
C2	1	2.2 or 3.7uF	50V electrolytic capacitor	**polarity:** "-" stripe to top - bend over before soldering for lower profile	Y
header	1	11 pin male	I/O header	insert in main board into: **0-gnd**, put Display Interface on top, and solder.	N





The Display interface board is wired to the display by matching the labels on the two boards (LED to LED, etc.) Note that the “TFT” and the “SD” sections refer to the same sections on the TFT. The choice of wire or ribbon cable to use is left up to you.

On the display side, the 8 pin male header is soldered in from the factory. An extra 8 pin female and 4 pin female header is provided if you want to use the display headers. Later, you may want to remove the display headers and solder the wires directly to the display when you case the project.

Keep the wires that go to the display as short as possible. When the wires are too long you may see strange problems on the display - rainbow patterns, missing pixels, and white screens. If long wires *must* be used, it may be necessary to shield them in some way. I had good results by winding them in aluminum foil and then insulating the shield. Grounding the shield may also improve its effectiveness.

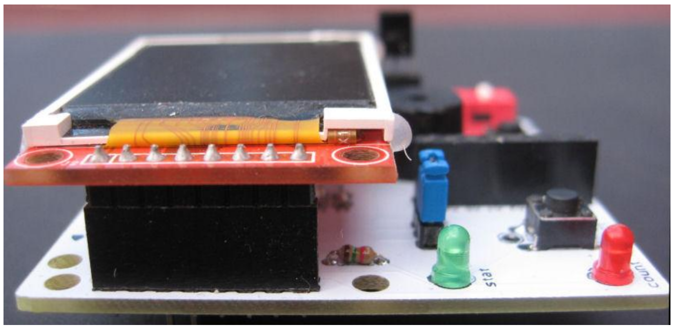
Step 1 - Building the kit – continued (Optional Display Adapter)

The optional adapter makes it easy to mount the display to the main board and gives you some push buttons, switches, and alarm piezo that you can use to try out the features. In addition it provides the necessary level shifting for the display.

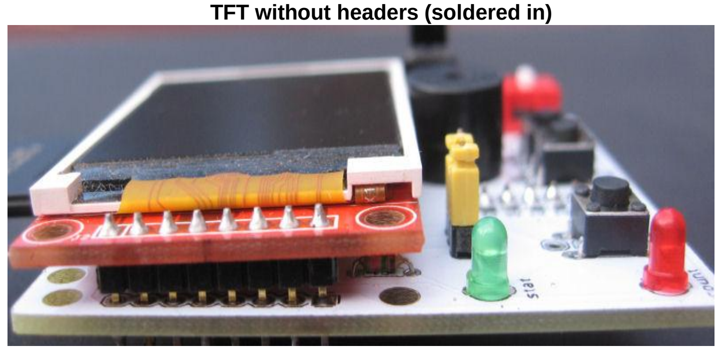
If, or when, you put your kit in a case, you can go back to using the Display Interface board, and mount the display and switches to the case with wires going to the I/O pins. A wiring diagram is provided below. Note that the extra pads by many of the components can be used for wiring them off the Display Adapter.

You can also build your case around using display adapter. If so, it is usually best to plug the display into the 8 and 4 pin headers as per these instructions. *However for a lower profile, you could solder the display directly to the adapter board as shown on the right below. This makes it about 5/16" (8mm) thinner. If you do this, be sure to cover the back of the SD card holder with electrical tape. Also clip the tops of the header pins it will rest against, and tape them too*.

TFT in header



TFT without headers (soldered in)



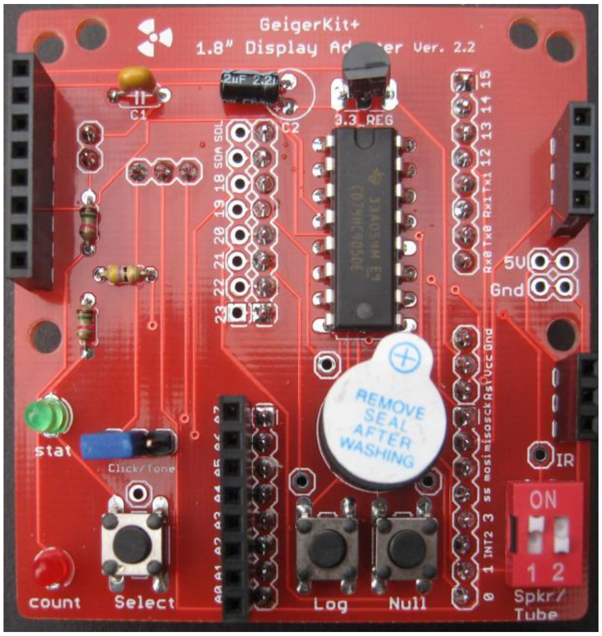
<u>Please note:</u> This board has a lot of headers. If you are not careful, it is easy to solder one in the wrong place. Prevention is best - **Be sure you refer the pictures before you solder.** If you do need to remove a header the easiest way is to remove the plastic 'case' and pull each pin at a time. On female headers the plastic case can be carefully pried up. Same with male headers, but it easiest just to cut the plastic between the pins.

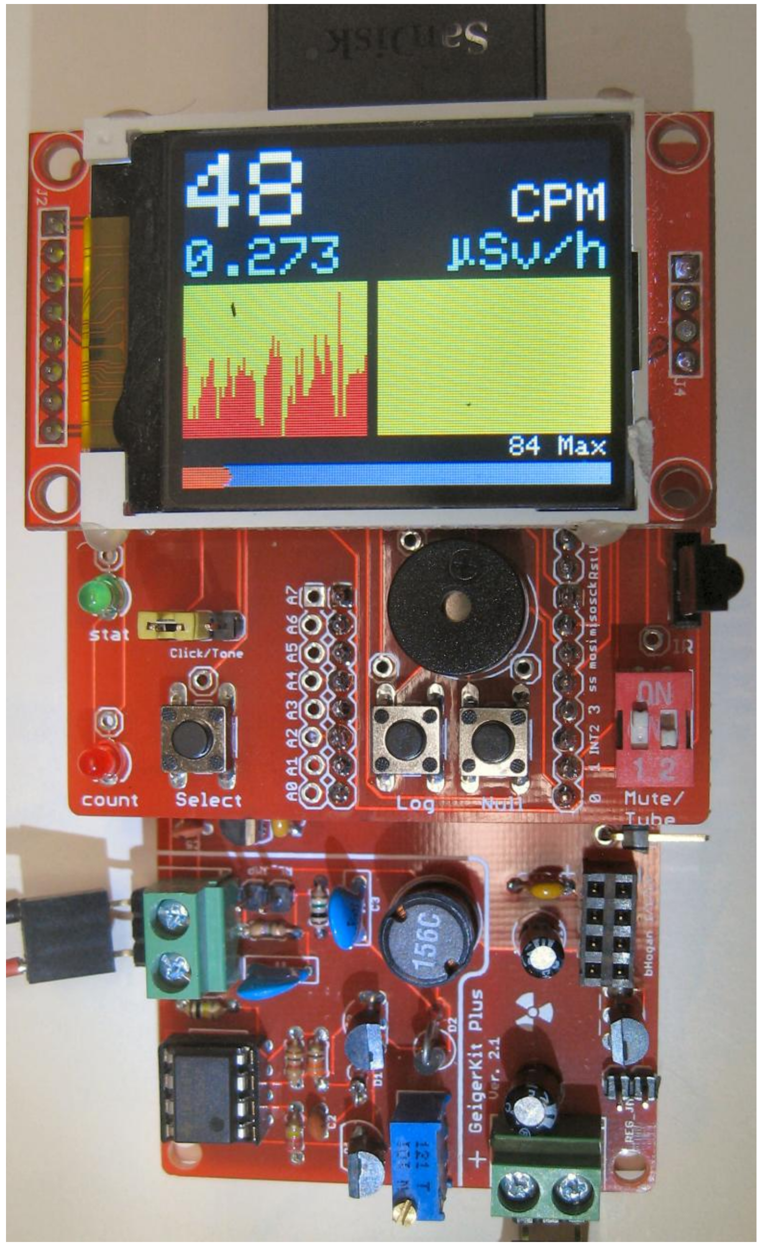
Build Sequence and Parts List for v2.2 Display Adapter

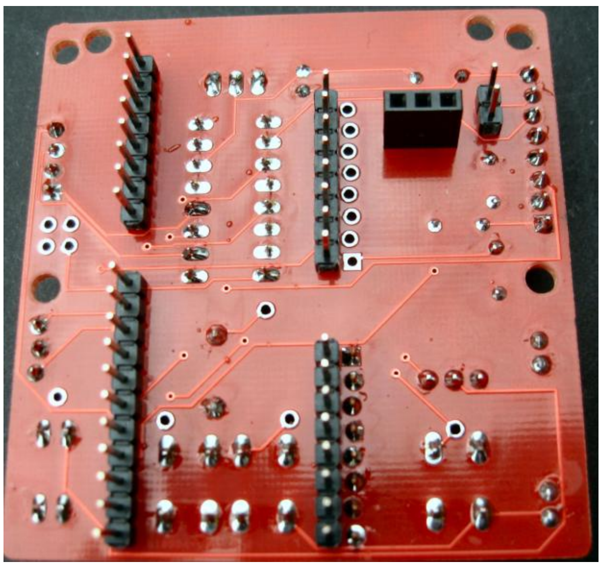
Ref #	Qty	Value	Description	Notes — polarized? ->	Y N
PCB	1	v2.2 or v2.1	2.3"x2.4" (~5.8 x 6.1 cm)	**Orientation:** HV boxed in area is on **upper left** side of board.	N
R1, R2	2	1.5kΩ	BN,GN, RD	(limits current to LEDS)	N
R3	1	47Ω	YL,VT, BK	V2.2 only - extra safety to limit backlight current at > max setting	N
IC1	1	**CD74HC4050E**	hex buffer for level shift	Bend pins in a bit on a flat surface. **polarity:** Notch up. Not socketed.	Y
REG	1	78L33	3.3V / 100mA regulator	If not using display headers - mount bent over to keep low - see pictures. Spread leads if necessary.	Y
C1	1	.1uF	#104 ceramic capacitor	If not using display headers - mount bent over to keep low - see pictures.	N
C2	1	2.2 or 3.3uF	50V electrolytic capacitor	If not using display headers - mount bent over to keep low - see pictures. **polarity:** "-" stripe to top	Y
LEDs	2	red / green	3mm	Red LED above "counts" label, green above "stat" label. **polarity:** Flat on side, **or shorter lead**, goes down for both.	Y
switch	1	DIP - 2 pos.	Mute (Spkr) / Tube	**polarity:** "ON" label up - set "Spkr" (left) switch to ON	Y
The headers below mount to the top of the display board. There are many wrong places to solder them!					
header	1	3 pin male	Click / Tone	use jumper from main board - put on click side for first tests	N
header	1	3 pin female	IR	vertical set of pads on right side	N
header	1	4 pin female	SD on TFT	TFT (right side) header **Not added if display soldered in - See above**	N
header	1	8 pin female	TFT Display	TFT (left side) header - **Not added if display soldered in - See above**	N
header	1	8 pin female	I/O header	A0-A7 header **Optional:** just used for future expansion by user.	N
The remaining headers are used to mate the main board to this display board. They mount to the <u>bottom</u> of the display board. (soldered from the top) Alignment is important, so you will first place the headers into its mate on the main board. Then you will put the display board over them and solder from the top of the display board. Don't try to do all at once – do in groups as indicated below.					
header	1	11 pin male	I/O header	insert into female header on **main board** labeled: **0-gnd**	N
header	3	8 pin male	I/O headers	insert into main board headers: **Rx0-15, 23-SDL, A0-A7** – **now solder all 4 in**	N
header	1	3 pin female	Click/Tone	insert into main board header labeled: **Click/Tone** – **solder in**	N
header	1	2 pin male	LED_OUT	insert into main board header labeled: **LED_OUT** – **solder in**	N
Finish up					
switch	3	push button	Select, Log, & Null	Snaps in and lays flat.	Y
alarm	1	piezo w/ osc	has paper seal on top	Supplied with main board - Install in vertical pads - **+ side up**	Y
IR Sensor	-	-	-	Move it from main board – bulge faces out	Y
jumper	-	Click/Tone	-	Move it from main board	N

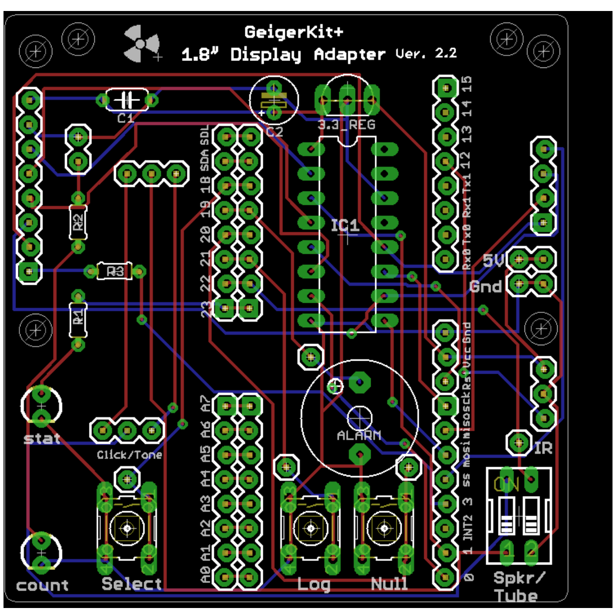
Note: When removing the display on this board, be sure to pull it on the red part not the white top and bottom edge.

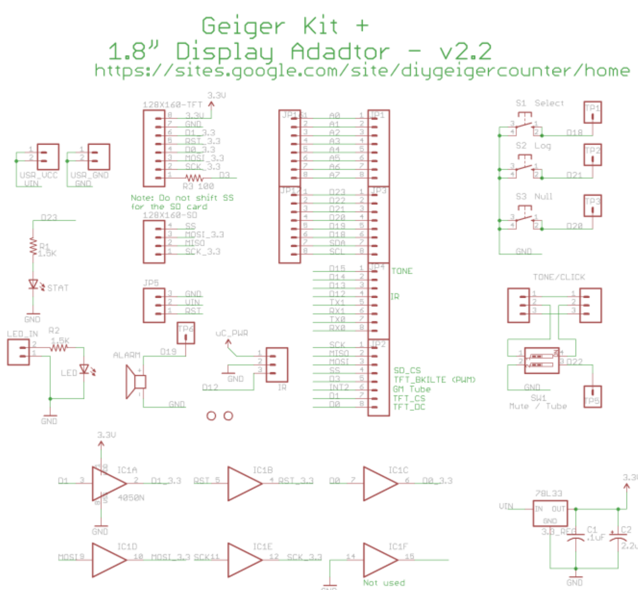
Assembly Images – display adapter board:











Step 2 – Inspect and Clean the main 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, (say for the RTC chip) be sure to clean the areas where it was used with alcohol.

Step 3 - Configure the jumpers:

Make sure that the UC-PWR and INT_JMP jumpers are installed and that the CLICK/TONE jumper is in the top 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 on jumpers.

Step 4 – Add the TFT LCD:

If you built the display adapter and used the headers for the display simply plug in the display. If instead you will be wiring the LCD to the Display Interface board, see the ***Wiring Diagram*** below.

You can also add an SD card to the socket on the display. A 2 Gb card works fine. By default the SD card will stick out of the top of the display. However, note that a menu option in the software allows you change the orientation of the screen, so you may be able to configure things so that the card is accessed from the side.

Step 5 - Power up the board:

Decide on how you will power your Geiger Kit. Refer to ***Appendix II - Powering the Geiger*** below to help you decide. Observe polarity. It should click once.

For the ultimate in flexibility you might consider replacing the on board regulator with step up / step down voltage regulator at some point. This allows for battery voltage above and below the 5V needed. Here is one I would suggest . . . http://www.pololu.com/product/2119/ . It takes an input of between 2.7- 11.8V and can output least 500mA at 5V with a typical efficiency of 90%.

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 100kΩ . (I just use my finger, but can't suggest that you do that.)

If you hear clicks congratulations! Now you might look at ***Appendix III - HV Test & Adjust***. The HV should already be adjusted to ~450V which will work with most tubes.

Step 7 – Adding a GPS:

A GPS is optional but if you will be traveling with your Geiger it can be very handy to have the location of where you got your readings. You must supply your own GPS module, and it's best to use one of the proven ones described in ***Appendix IV – GPS Wiring***. Here are some tips on GPS:

- Make sure the power connections to the GPS are correct. Wrong connections can fry your GPS.
- The GPS module will be powered at 5V. Make sure your GPS is 5V tolerant. (The supported modules are.)
- Different models have different init sequences. Using a model not listed may require coding work on your part.
- You will know you have a fix when speed and altitude is displayed on the status line of the main display.

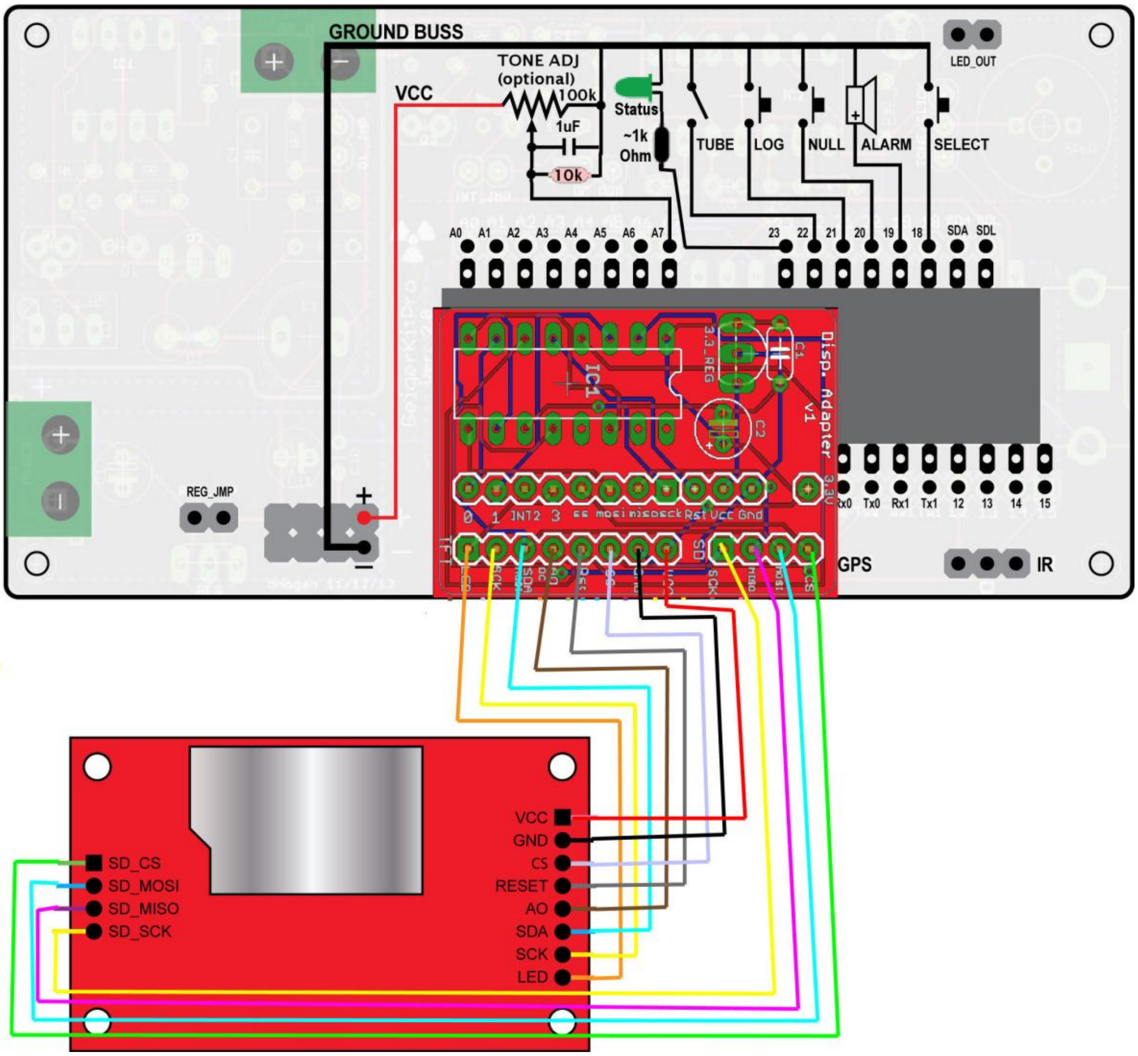
Step 8 – Understanding the features:

More than likely you're going to explore the kit at this point. Have fun. But there are a fair amount of hidden features, so when you have an opportunity, like when you're on the toilet, read the separate ***GK-Plus User Guide*** for more info.

Enjoy the kit!

Wiring Diagram for Display and Control Connections (if not using the Display Adapter board):

(larger image in browser here)



Pin Map for ATmega1284P with GK+ Software

define	pin	function
TFT_RESET	-1	connected to RST header pin (***chip*** pin 9) - must define as -1
TFT_DC	0	aka "A0" data/command signal for display
TFT_CS	1	CS for the TFT
INT2	2	Events from Geiger circuit
TFT_BKLITE	3	PWM pin to power TFT backlight
SD_CS	4	(hardware) SPI - CS pin - for SD card
MOSI	5	(hardware) SPI - MOSI pin - for SD card & TFT
MISO	6	(hardware) SPI - MISO pin - for SD card
SCLK	7	(hardware) SPI - SCLK pin - for SD card and TFT
Tx0	8	used for serial comm
Rx0	9	used for serial comm
RX1 / INT0	10	used for GPS comm
TX1 / INT1	11	used for GPS comm
IR_PIN	12	PCI Interrupt for IR sensor on this pin
free	13	PWM pin open
SPKR_MUTE	14	signal pin to mute speaker - board mod required
TONE_PIN	15	PWM pin output to speaker or piezo for tone mode
I2C SCK	16	for RTC
I2C SDA	17	for RTC
SEL_BUTTON	18	Select button to toggle scaler display, set and silence alarm
ALARM_PIN	19	Outputs HIGH when Alarm triggered
NULL_BUTTON	20	Null button used to set the null point in tone mode
LOG_BUTTON	21	log to card and mark it if this pin goes low
TUBE_SEL	22	jumper LOW to select alt conversion to uSv
LED_PIN	23	for “Stat” LED
free	24-30	A0 to A6 open
TONE_POT	31	A7 tone adjustment pot - must be analog pin

Appendix I - The Jumpers:

There are 3 jumpers on the top of v2.2 board and 3 solder jumpers on the bottom. Normally all three jumpers on the top of the board have a jumper block installed and the solder jumpers are not soldered over (open).

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 ATmega1284 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)**

Opening this jumper removes power from the microprocessor, display, SD card, IR sensor, and GPS. This puts the kit into a "click / flash only" mode and will bring the power consumption down to about **9mA**. Note however, that the **MUTE** solder jumper (described below) must be left open, otherwise the click sound will be diminished.

Another use for this jumper is to allow for 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 Vcc pin on the 11 pin header or FTDI connector. Putting a HIGH on the output pin will turn on the HV and a LOW will turn it off.

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

Shorting this jumper with a blob of solder allows the MUTE button on the IR Remote to mute the sound coming from the speaker. This might be useful if you keep your remote handy but a better method is to simply wire a switch to the CLICK/TONE jumper and use the “center position off” to mute the speaker. A disadvantage of shorting this jumper is that the UC_PWR will no longer provide a loud click when opened, so in effect you sacrifice the advantage of the UC_PWR jumper.

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 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.

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

This jumper bypasses the 5V voltage regulator. Do not short this jumper when running at voltages of 5V or greater. This jumper could be shorted if you are running on 5V or a bit less to save a tiny amount of power. However, since the GeigerKit Plus uses a "low dropout regulator" (LDR) power savings are minimal if any. In short, this jumper is normally not used.

Appendix II - Powering the Geiger

Power Consumption:

Because the GeigerKit Plus uses a TFT graphic display, it uses more power than the basic kit with a 2x16 character display. Adding a GPS increases power consumption even more. This should be kept in mind when considering a power source. An approximate breakdown of power consumption is as follows:

- Base operation (HV, click & LED, microprocessor): **40mA**
- with TFT display at default backlight setting: **50mA** (Running the backlight at 100% uses up to 10mA more over the default setting)
- with MTK3339 GPS: **90mA.**

As you can see, you will save a lot of power running the backlight at lower levels and selecting a GPS that uses less.
The "uC_PWR" jumper can be used to put the unit in a "click / flash only" mode. This is described in the section on jumpers above. With the jumper open, the microprocessor, display, SD card, IR sensor, and GPS are disconnected. In this state the kit will draw only **9mA**.

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. If the battery pack will supply more than 5.5V the REG_JMP should not be shorted to let the voltage regulator do its job (20V max). 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 6 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 contrast), 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 would 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%.This part can replace the voltage regulator on the kit. Mount it horizontally, parts side up, on a 3 pin male header soldered into the REG pads as shown on the right. Do not short the REG_JMP with this part installed!.

Powering with the FTDI

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.

Power usage and the Backlight

The backlight on the TFT display is driven by PWM with the duty cycle being adjusted by the menu setting. The maximum duty cycle is capped in the software at 200 (out of 255). This limits the average current going to the backlight.

Beginning with v1.1 of the Display Interface board and v2.2 of the Display Adapter, an additional current limiting resistor was added. Those with boards before these versions should avoid the two higher settings above the default setting of 7 bars (14).

The backlight can consume a lot of power. The following table shows how much current is used at the 16 different display brightness settings:

setting:	16	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1
mA:	19.4	13.7	9.7	6.8	4.9	3.5	2.5	2.1	1.9	1.7	1.4	1.2	1.0	0.8	0.6	0.4

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 :

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

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 21Ω 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.

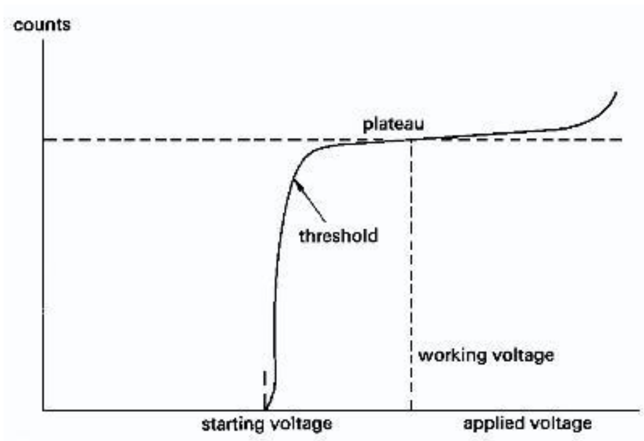
~HV	Ω at R5	~HV	Ω at R5	~HV	Ω at R5	~HV	Ω at R5
55V	100	400V	19.9	575V	14.6	800V	11.1
100V	58.4	425V	18.9	600V	14.1	900V	9.7
200V	32.9	450V	18.1	625V	13.6	1000V	8.4
300V	24.4	475V	17.3	650V	13.2	1100V	6.9
325V	23.0	500V	16.5	675V	12.7	1125V	5.3
350V	21.8	525V	15.8	700V	12.2	may go higher -	depending on parts
375V	20.7	550V	15.2	750V	11.5	HV crash	5.1

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 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 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.**

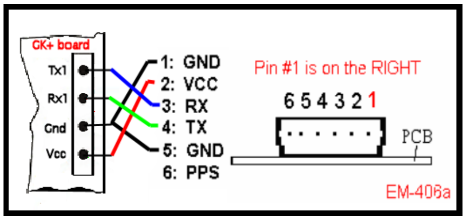
To get the values for <u>your meter</u> do the following:
Using some kind of active source, lower the HV (CCW) until you get no response. (R5 is a 25 turn pot). Now slowly increase the HV (CW) until you *just start* to get clicks. Record the initial voltage. Now slowly increase the HV again until you 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 – GPS Wiring

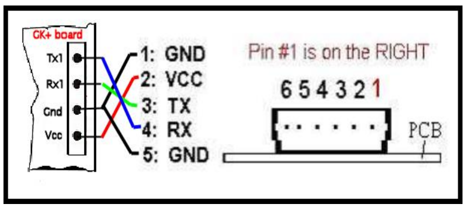
Different models have different connections. Refer to the wiring diagram on the left of the description for each GPS. Comments regarding opinions and pricing are my own.

The EM-406a: (Google for sources – it can now be had at very reasonable prices.)



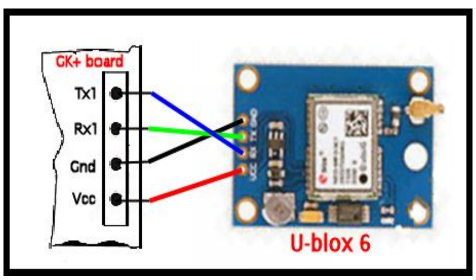
Select **EM406/408A** from the GPS menu.
(The EM-411, PMB 648 and EM-506 are also *reported* to work with this setting.)
It's usually best to cut the supplied cable and solder the ends to a header that will connect to the kit's GPS header.
Fix LED: steady = no fix - 1 flash /sec = have fix
The PPS signal is not used.

The EM-411: (Google for sources – it is generally priced less than the EM-406a and has similar capabilities.)



Select **EM406/408A** from the GPS menu.
Fix LED: steady = no fix - 1 flash /sec = have fix
It's usually best to cut the supplied cable and solder the ends to a header that will connect to the kit's GPS header.
Note the Tx and Rx pins are reversed on the GPS side compared to the EM-406a.

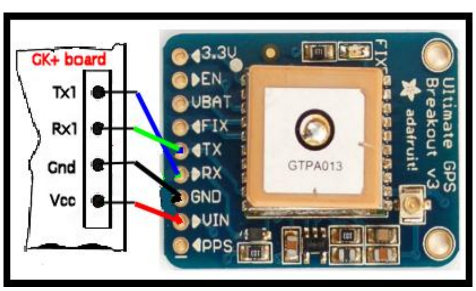
U-blox 6: (This GPS can be found on eBay).



Select **U-blox 6** from the GPS menu.
A new kid on the blox! Lowest price and excellent at getting a fix indoors with its included chip antenna. External antenna connector nice to have.
Fix LED: not lit = no fix - 1 flash /sec = have fix

Recommended. Normally this GPS draws about 30mA more power than the MTK3339 GPS listed below. However, the SW puts this GPM in low power mode when 5 or more satellites are fixed. This saves most of the 30mA.

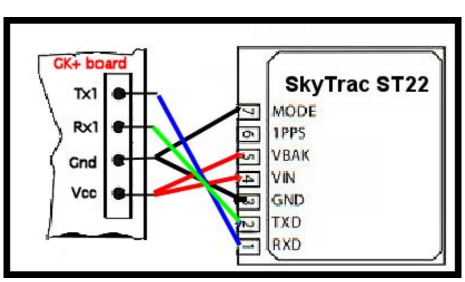
Adafruit "Ultimate GPS Breakout" (MTK3339):



Select **MTK3339** from the GPS menu.
There are "advanced" connections that can be made - breaking out the "Fix" LED for example. It can accept an external antenna and works at high altitudes (27km!). It is also capable of doing its own logging if you want to experiment with that. (not supported in SW)
Fix LED: 1 flash /sec = no fix - 1 flash /10 sec = have fix

You should do the Adafruit mod to install the battery holder and backup battery on this GPS.

SkyTraq ST22: (This GPS can be found at http://www.mr-lee-catcam.de/pe_cc_i6.htm).



Select **SkyTraq 22** from the GPS menu.

The price has come down on better GPS modules making this one much less attractive. Retained in code only for legacy purposes.

There is no on board LED that indicates a fix. Having one is especially helpful during initial setup. There is no on board battery or SuperCap to hold the last fix. This means longer searches (~25 min) at startup.
Not suggested.

Appendix V - Main Board Schematic