RC Tractor Guy Controller V2.1 Assembly Guide

Features

- 20 Push button inputs
- 2 Dual axis thumb sticks with built-in push button
- 2 Rotary encoders with built-in push button
- MCU Socket to suit Meduino Mega 2560 Pro Mini
- Standard Arduino shield socket
- SPI Socket to suit NRF24L01 footprint
- UART Socket to suit XBee footprint

Assembly Hints

Required Tools

- A soldering iron (25-40W) with a small tip
- Wet sponge to clean soldering iron tip
- Thin solder
- Wire snips/diagonal cutters to trim component leads
- Tweezers to aid in positioning of SMD components
- Philips screwdriver

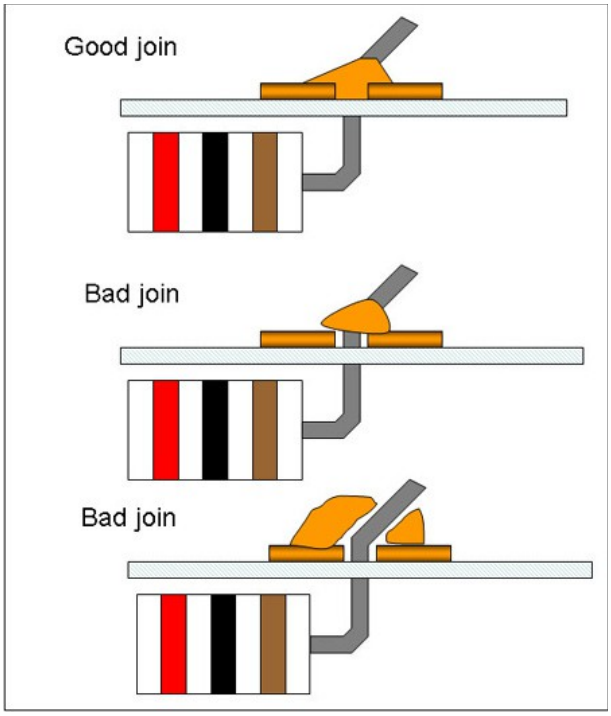
Assembly Tips

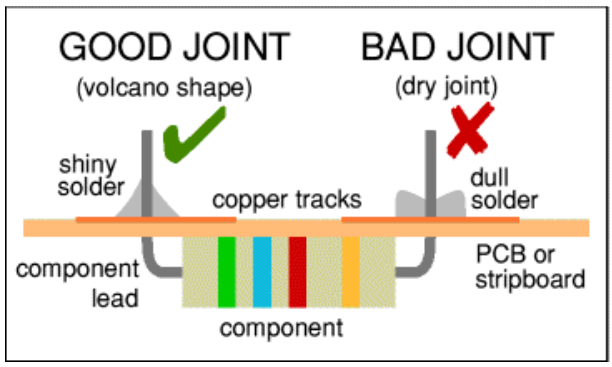
- If you are unsure of your soldering skills it is best to get some assistance
- Children should be assisted by an adult
- Perform assembly in the order stated below to ensure ease of assembly

Soldering Tips

Watching the EEVBlog soldering tutorials on YouTube is highly recommended

- Clean the soldering iron tip regularly and apply solder to the tip to give it a wet look
- This is called thinning and will make the soldering process easier
- It is occasionally useful to thin component legs to make soldering easier e.g. SMD components
- Mount the components flat against the surface of the PCB
- Apply solder to one components leg, choose a corner leg for multi leg components
- Hold the soldering iron against the joint long enough for the solder to flow properly across both the component and PCB surfaces
- Remove the soldering iron shortly after the solder flows to prevent damaging components with excessive heat
- Ensure the solder joint is cone shaped and shiny
- Assess the components positioning
- Solder the leg furthest away from the first solder joint
- Reassess the component positioning
- Solder the remaining legs
- Trim excess leads as close as possible to the solder joint



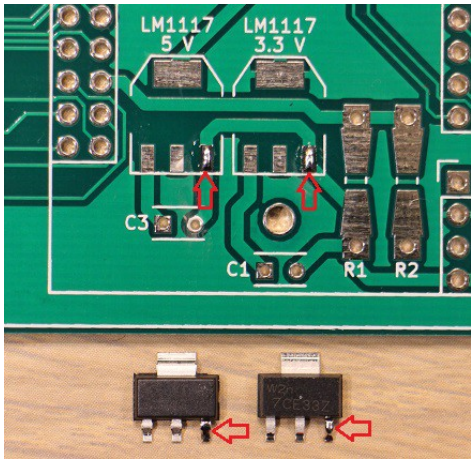


Kit Components Checklist

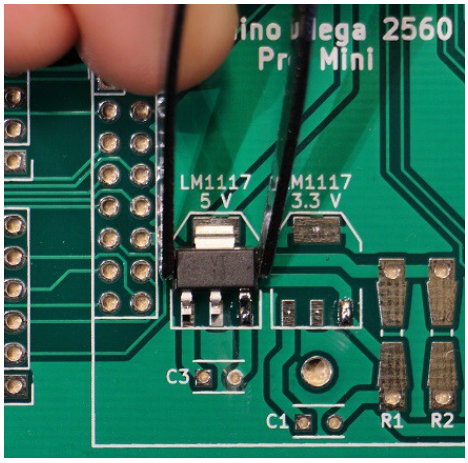
Part	Quantity	Check
Main PCB	1	
Front PCB	2	
Rear PCB	2	
Joystick Gimbal	2	
Joystick Cap	2	
Female Header 2x16	1	
Female Header 2x8	2	
Stackable Header 1x6	2	
Stackable Header 1x8	2	
Female Header 2x4	3	
Push Button 24mm	20	
DC Jack Socket	1	
Rotary Encoder	2	
Encoder Cap	2	
Power Switch	1	
6xAA Battery Holder	1	
NPN Transistor	1	
IR Emitter	2	
1k Resistor	4	
2.2k Resistor	1	
3.3k Resistor	1	
20 Ohm IR Resistor	2	
1x8 2mm Header (Xbee)	2	
DIP Switch	1	
Diode	2	
15mm Standoff	6	
14mm Standoff	6	
Mounting Screws	12	
3.3 V Regulator	1	
10uF 35V	4	
5V Regulator for IR LEDs	1	
Jumpers	8	
Battery Mounting Screws	2	

Assembly Procedure

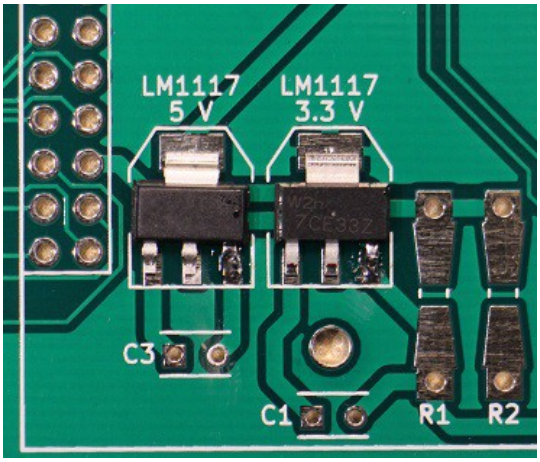
1. SMD Voltage Regulators



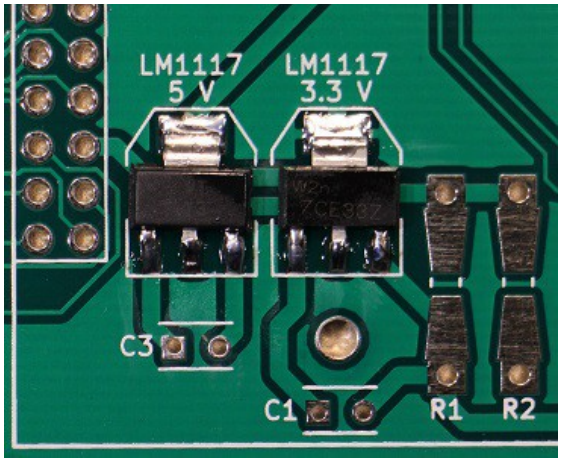
Soldering the will be easier if you first thin one pad on the PCB and one leg on the regulator



Align the regulator above the pads using a tweezers to hold the component

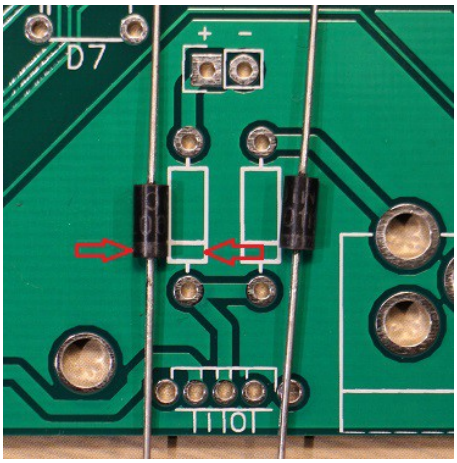


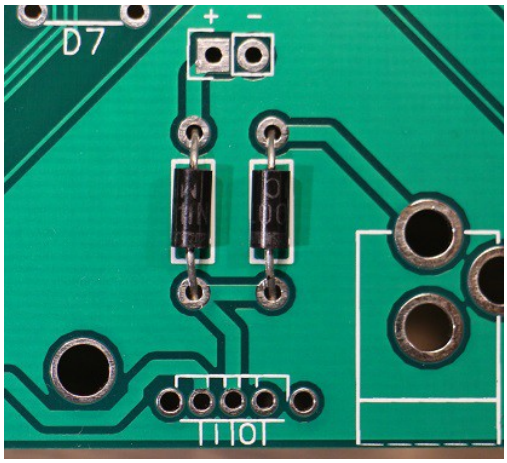
Solder the first leg of each regulator and check the alignment. Reheat the joint to adjust the position if necessary



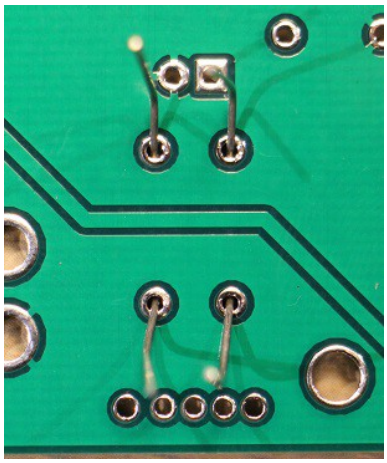
Solder the remaining legs

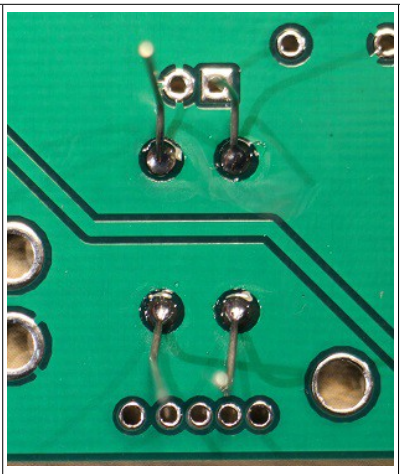
2. Diodes





Diodes are polarity sensitive, ensure that the line on the diode is in the same orientation as the symbol on the silkscreen.

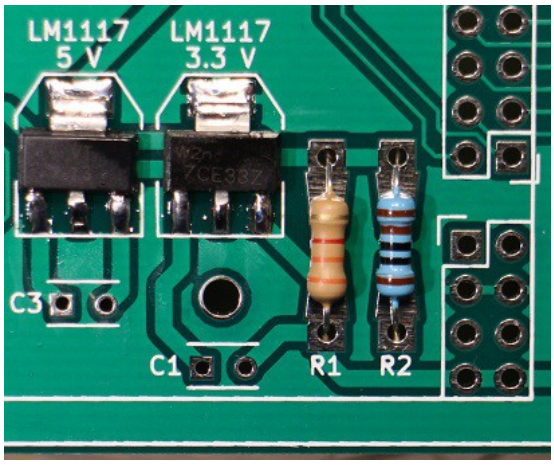




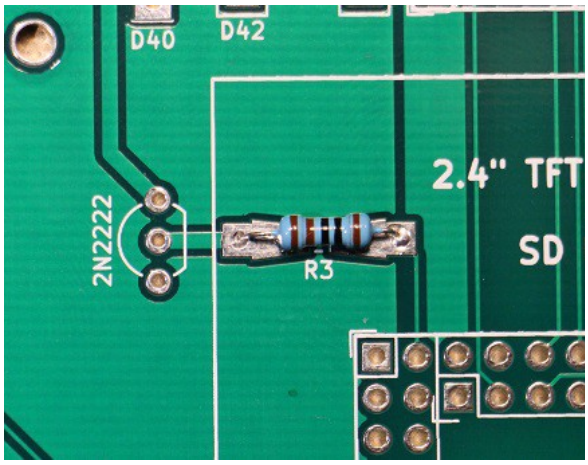
Bend the legs of the diodes out to secure them in place. Then turn the PCB over and solder the legs in place. Trim the excess off the component legs.

3. <u>Resistors</u>

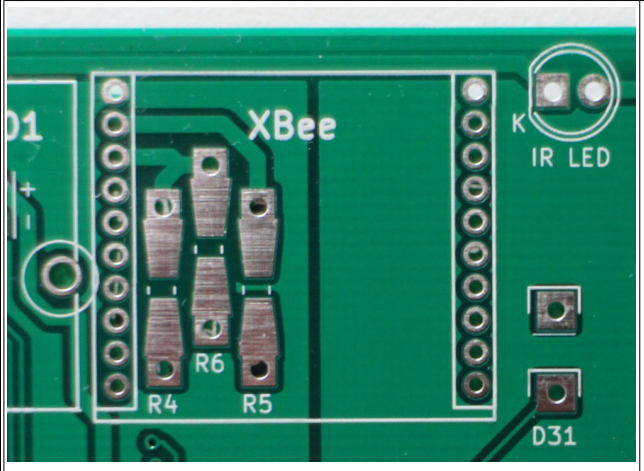
The resistors are not polarity sensitive so you don't need to worry about their orientation. Push the legs through the PCB, bend the legs slightly to hold them in place and solder them as you did with the diodes.



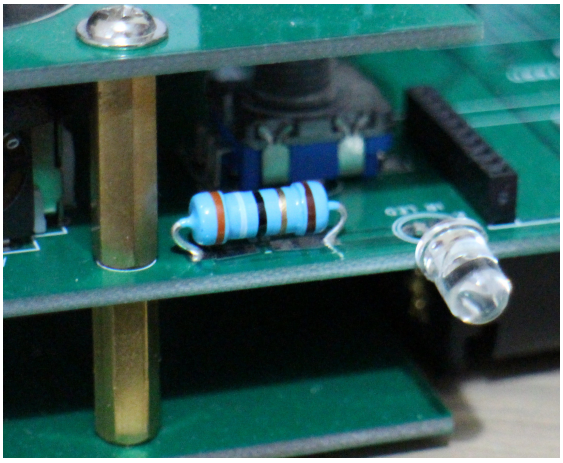
R1 and R2 are a voltage divider used to safely measure the controller supply voltage. In this picture R1 is 3k3 Ohm and R2 is 1k Ohm.



The 1k Ohm R3 resistor limits the current drawn from the MCU output to switch the IR LED transistor.

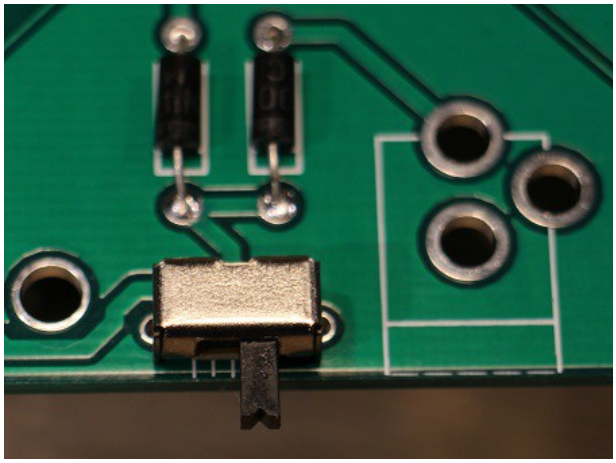


R4 and R5 are both 1k Ohm resistors used to give priority to the FTDI chip during sketch upload. R6 is a 2.2k resistor which reduces the Arduino TX from 5V to under 3.3V

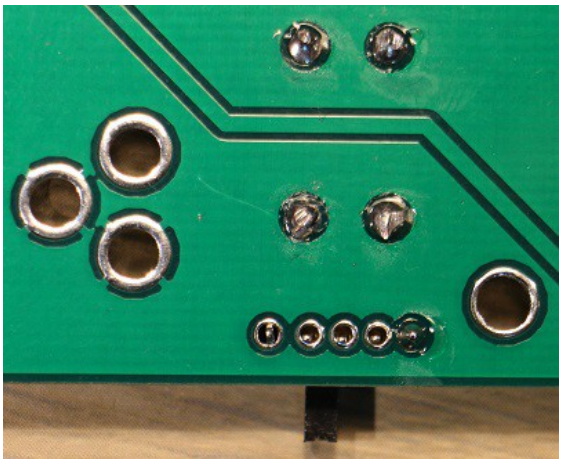
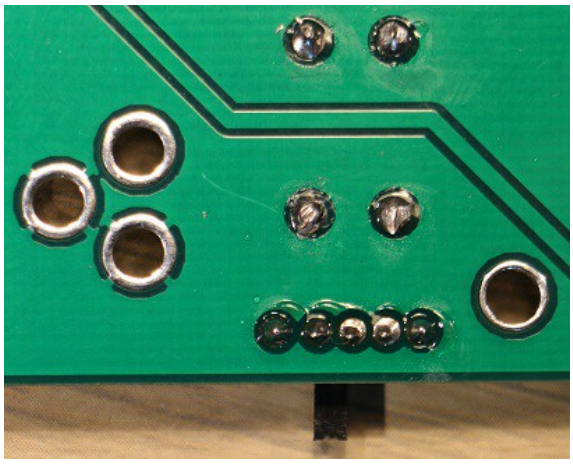


R7 and R8 are the resistors used to limit the current used to drive the IR LEDs. The resistance depends on the LED spec, usually ~10-20 Ohms. 1W 39 Ohm resistors are supplied with the kit. I left the resistor foot print the same and bent the legs as I taught raising the resistor up would help to dissipate heat.

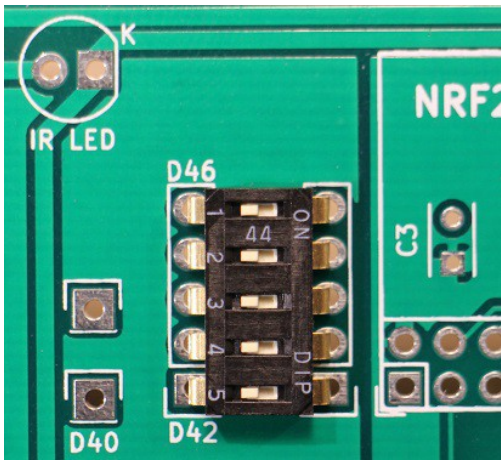
4. Power Switch



The power switch is located in the bottom right hand corner of the PCB.

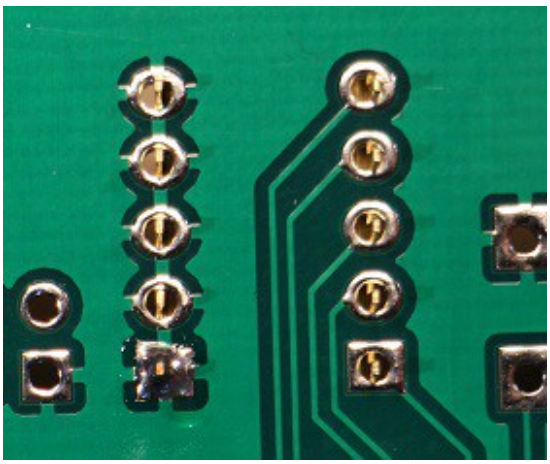
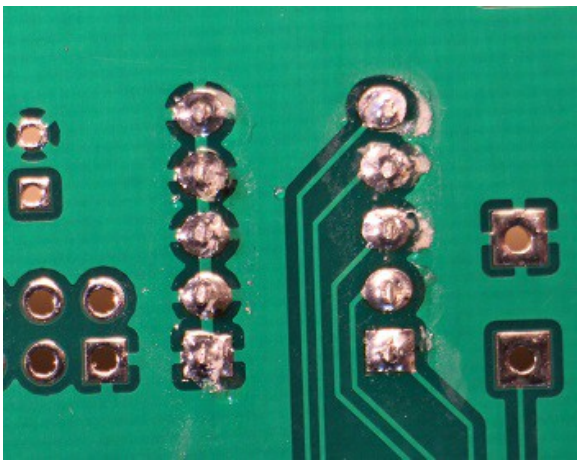
	
Solder a pin on one side switch then check the switch alignment.	Solder the remaining pins if you're happy with the alignment of the switch.

5. DIP Switch

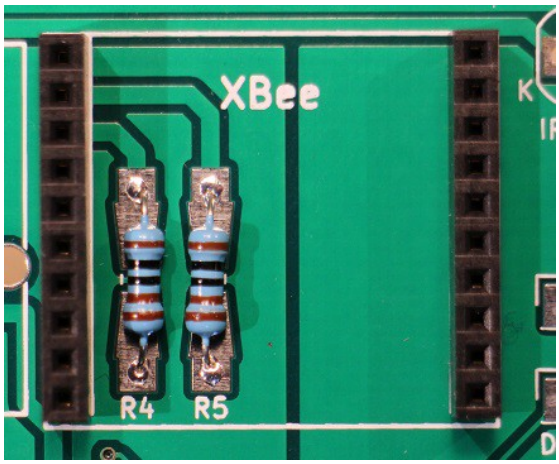


The DIP switch is located along the upper edge of the PCB, to the left of the NRF24L01.

This switch could be used for adjusting settings like channel selection of a radio.

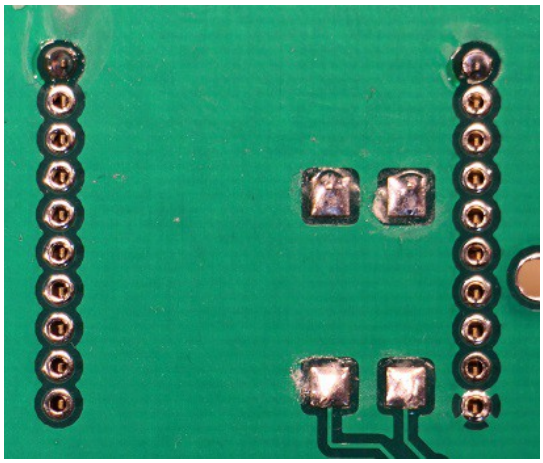
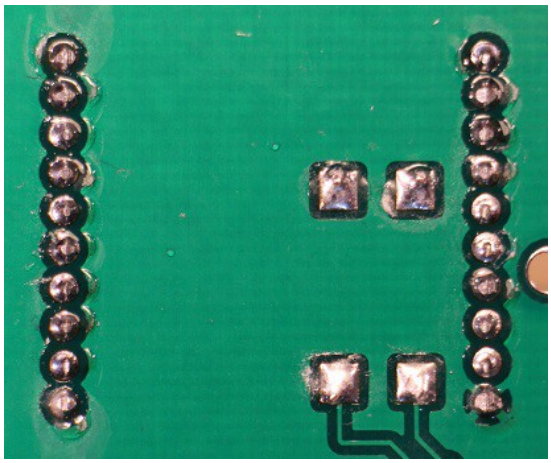
	
Solder a corner pin and check alignment.	Solder the remaining pins.

6. <u>XBee Header</u>



Xbee modules use a header with 2mm pin spacing.

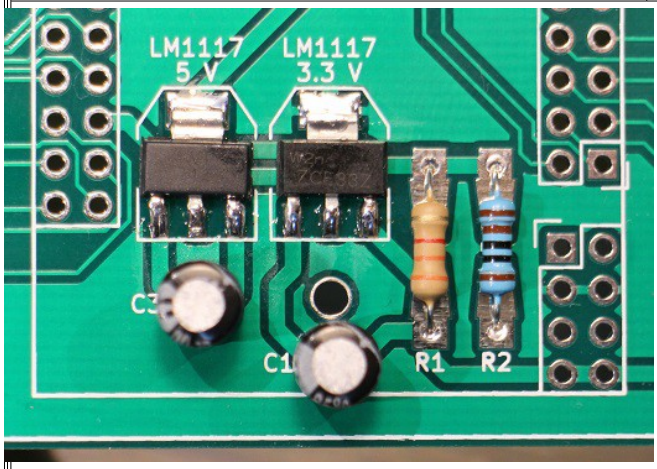
This header is located along the upper edge of the PCB.

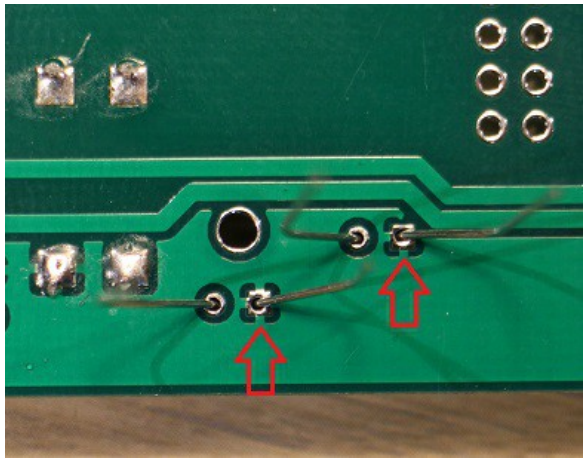
	
Solder an edge pin on each female header section and check the header alignment.	If happy with the alignment solder the remaining header pins.

7. Capacitors

There are 4 capacitors on the controller, these are all 10µF electrolytic capacitors.

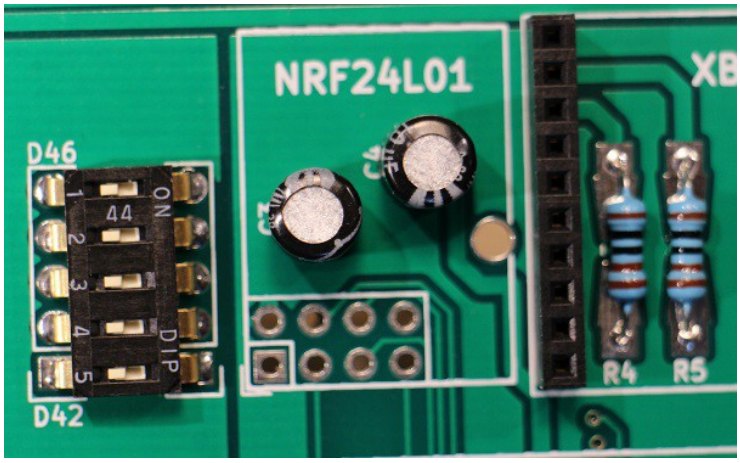
Electrolytic capacitors are polarity sensitive, the negative leg is shorter than the positive leg. The body of the capacitor also has a white stripe with a negative symbol to indicate the negative side.

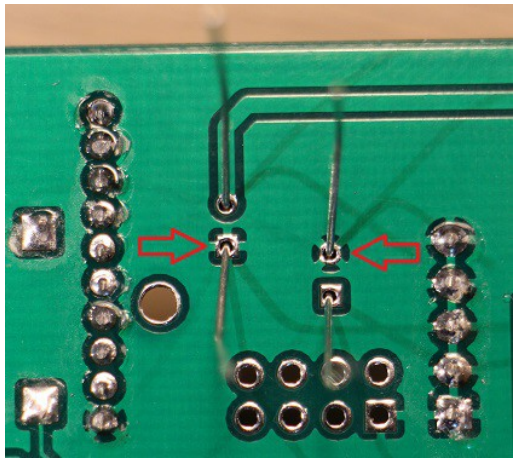




C1 and C2 are located at the input to the voltage regulators along the bottom edge of the PCB.

The connection to the ground plane is clear on the underside of the PCB. Ensure the shorter leg is connected to the indicated pad.



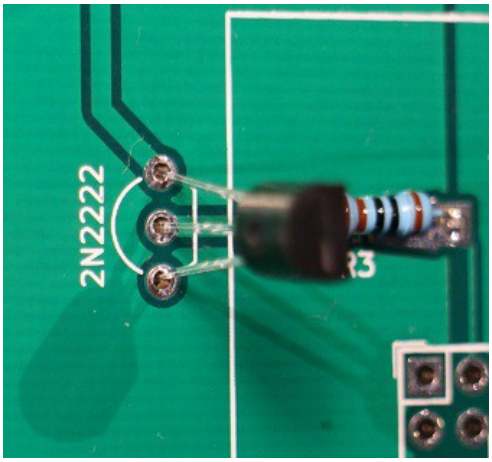
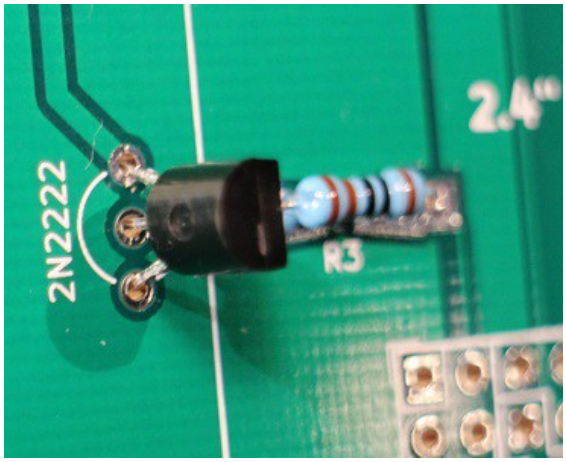


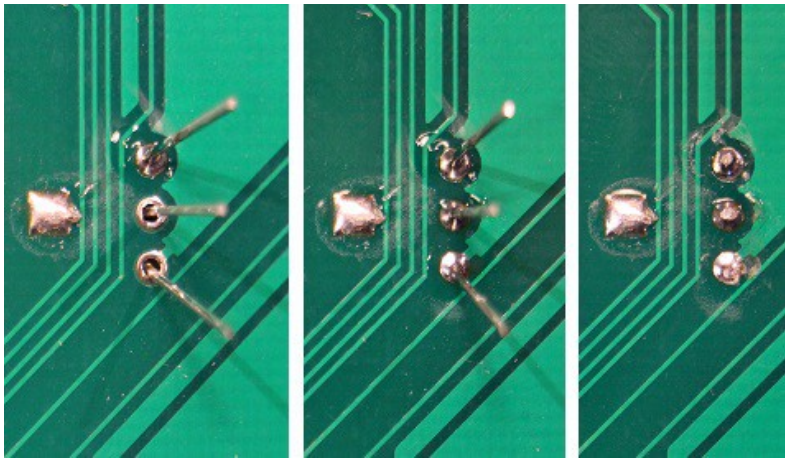
C3 and C4 are located beside the radio modules along the upper edge of the PCB.

Again ensure the shorter capacitor leg is connected to ground as indicated in the picture.

8. <u>Transistor</u>

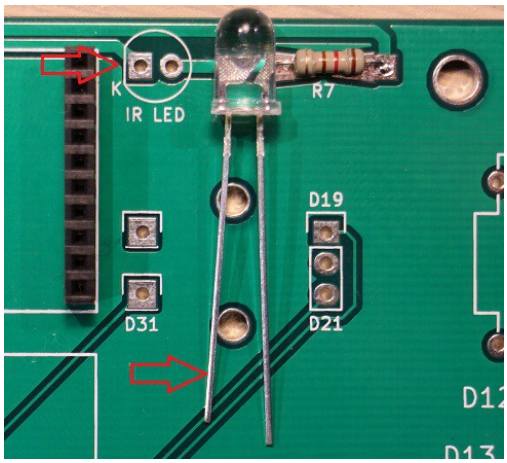
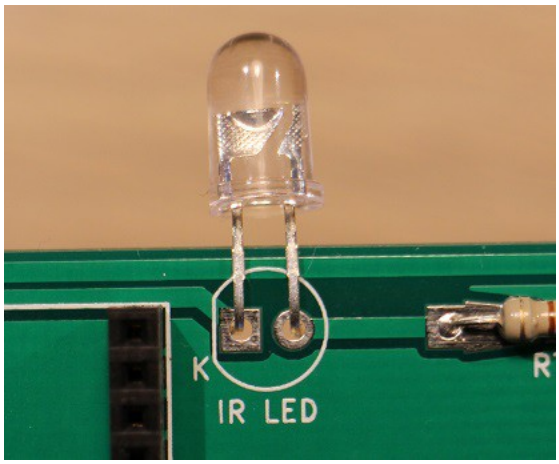
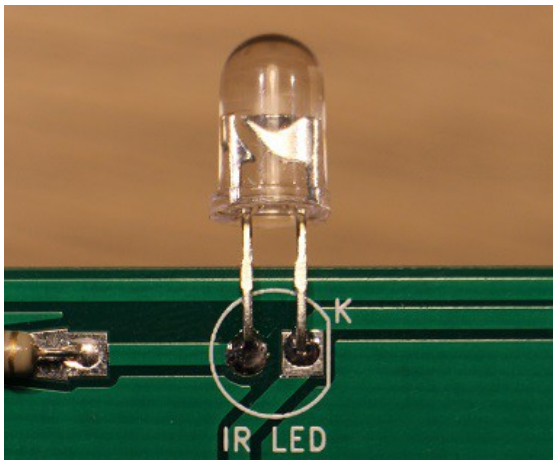
The transistor is used to drive the IR LEDs as the MCU output can't supply sufficient current.

	
Carefully spread the legs of the transistor to line them up with the holes on the PCB. Ensure the curved edge of the transistor is aligned with the curved edge of the symbol on the silk screen.	Gently push the transistor toward the PCB but don't press it hard to the board as you risk damaging the transistor legs. A gap of 3 or 4 mm is perfectly acceptable.

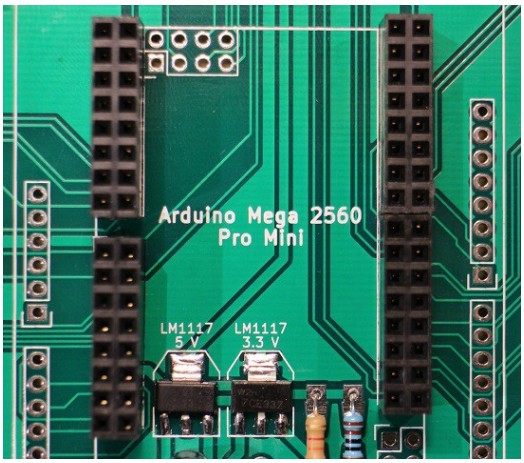
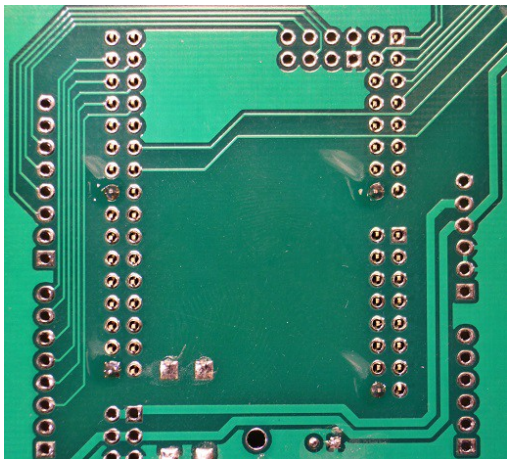


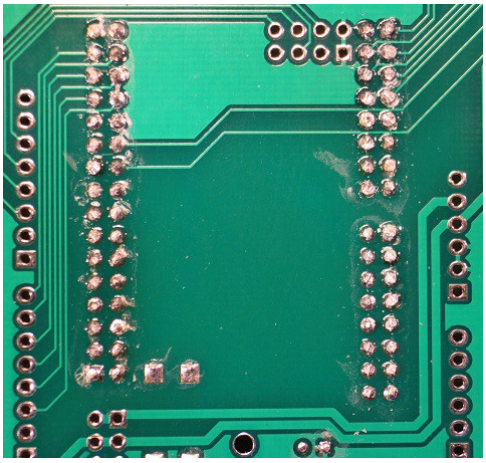
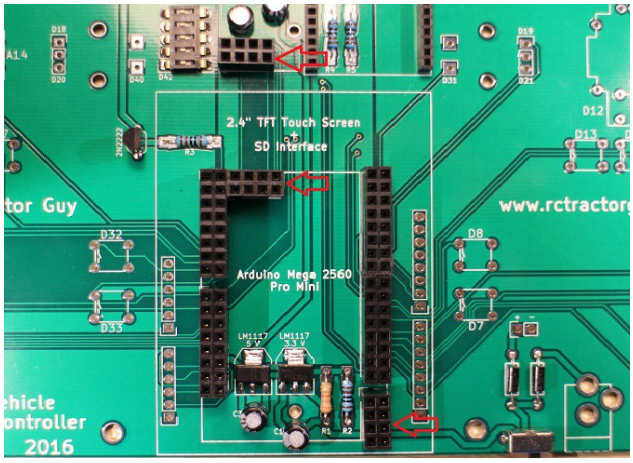
Solder one leg and check the transistor alignment. When you are satisfied with your positioning, solder the remaining legs and trim away the excess.

9. IR LEDs

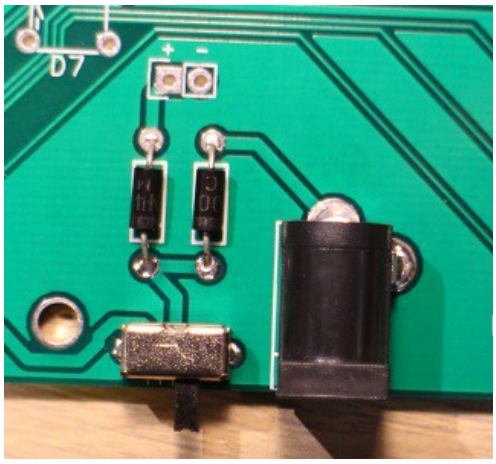
Light emitting diodes are polarity sensitive. The negative leg is denoted both by a shortened leg and also be a flat edge on the base of the LED. Ensure that the flat edge lines up with the edge indicated on the silkscreen.	
	
Bend the legs of the LED at 90° leaving a small gap between the LED and the PCB edge so that you can adjust the transmit direction later.	Solder both IR LEDs in place taking care to check polarity before soldering.

10. MCU Header

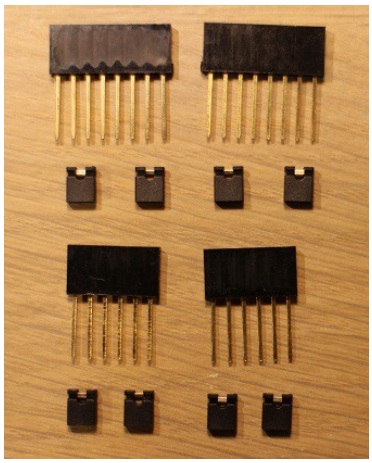
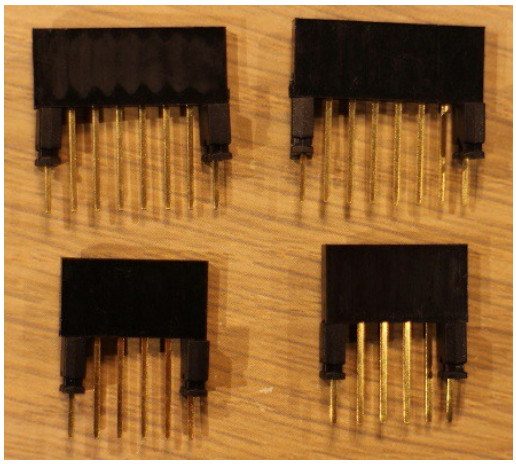
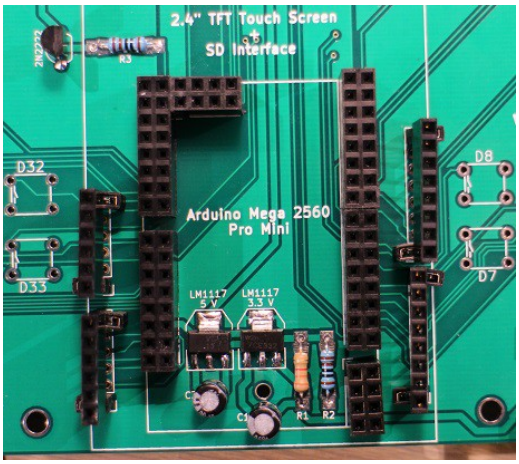
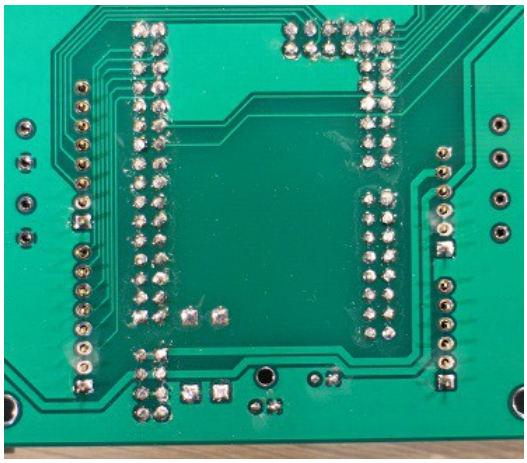
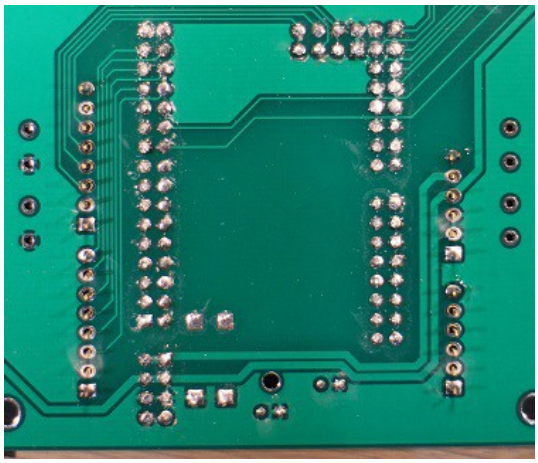
	
The MCU header consists of two 2x8 2.54 mm header sections and one 2x16 2.54 mm female header section as shown above.	Solder a corner pin on each section and check the alignment. The straighter these sections are the easier it will be to plug in the MCU.

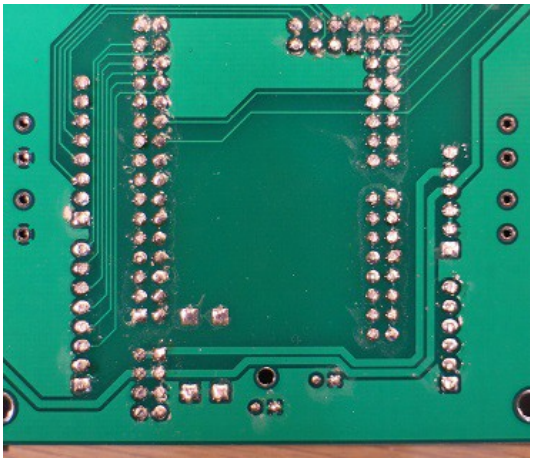
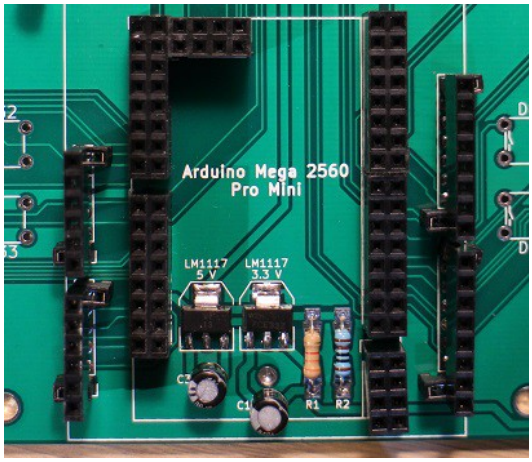
	
When happy with the placement, solder in the remaining pins starting with the furthest pin from the first one you soldered.	Two 2x4 header sections complete the MCU socket and a third is used for the NRF24L01 socket as shown above. Take similar care with placement of these header sections.

11. <u>DC Jack</u>

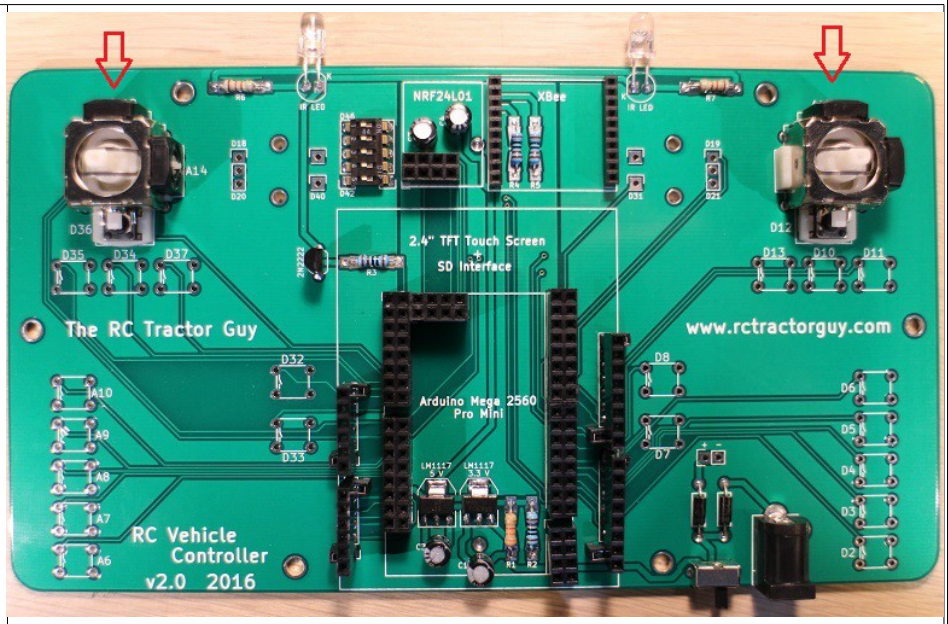
Solder the DC jack in the bottom right corner of the PCB. This is useful for extended use of the controller, using a power supply instead of the batteries. Input voltages of around 9V would be ideal to prevent excessive wasted power and heat. The regulators should be able to cope with voltages up to 15V but I don't recommend you push it that far.	

12. <u>Shield Header</u>

It turns out that finding header which is taller than normal is difficult and expensive when you do find it. To get around this problem I decided to use a combination of stackable header and header jumper connections.	
	
Push a jumper connection onto a pin at each end of the stackable header with the bottom end up so it is firmly against the bottom of the female header plastic.	One jumper should stick out to the left and the other should stick out to the right as shown above.
	
To get the header to be stable you will need to push it tight to the PCB as you solder the first pin.	Check the alignment and while applying pressure again solder the pin on the opposite side of the header.

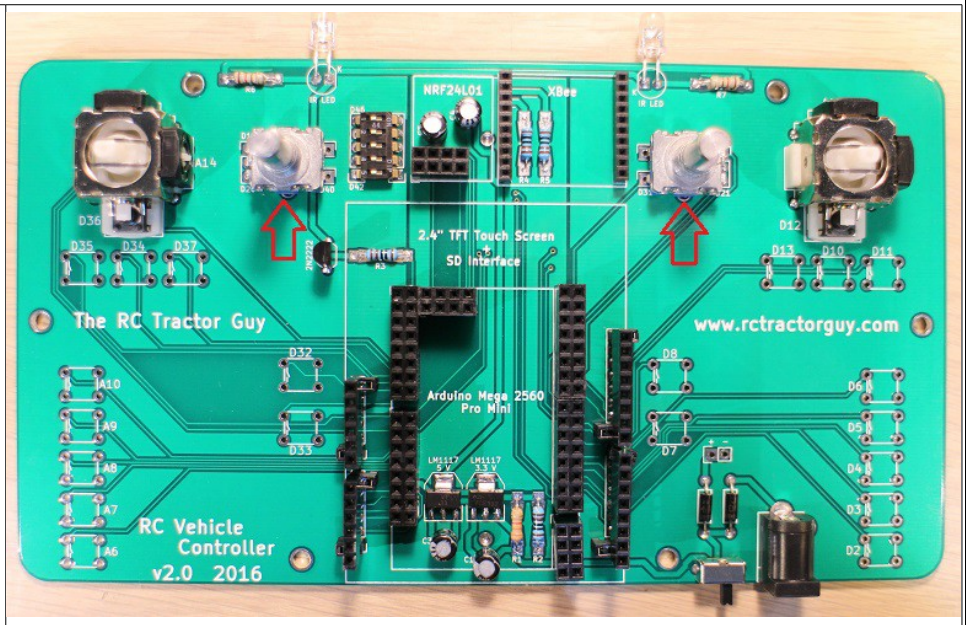
	
Check that the header doesn't seem loose and solder the remaining pins in place.	It should look like this in the end.

13. <u>Thumb Stick Gimbals</u>

The pins of the thumb stick gimbals may need some adjustment to align with the holes in the PCB. Once the are located on the PCB solder one of the larger structural pins and check the alignment. Take care that the gimbals are flat on the PCB before you solder too many pins as making adjustments afterwards would be difficult.	

14. Rotary Encoders

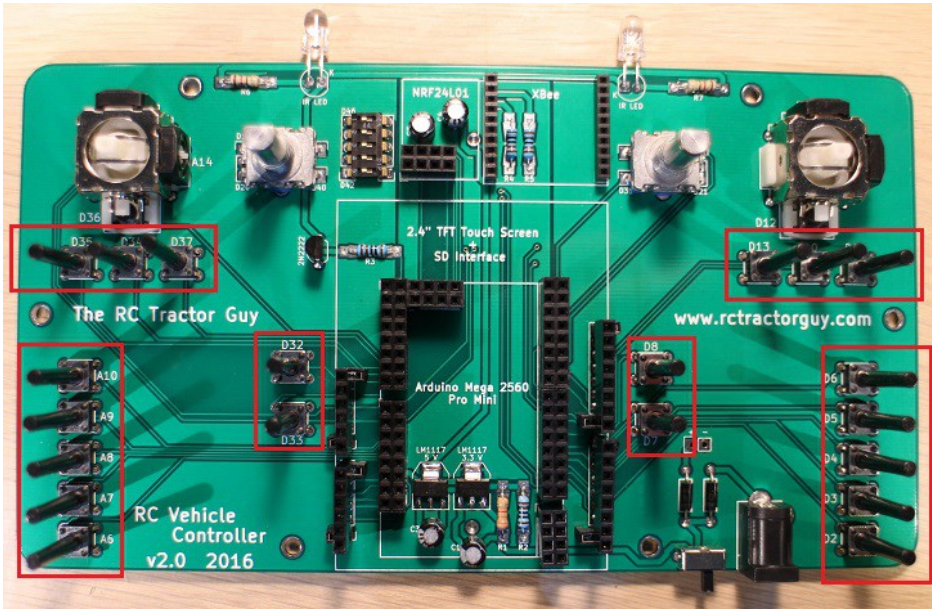
Similar care should be taken when mounting the encoders but it is easier due to the lower number of pins.



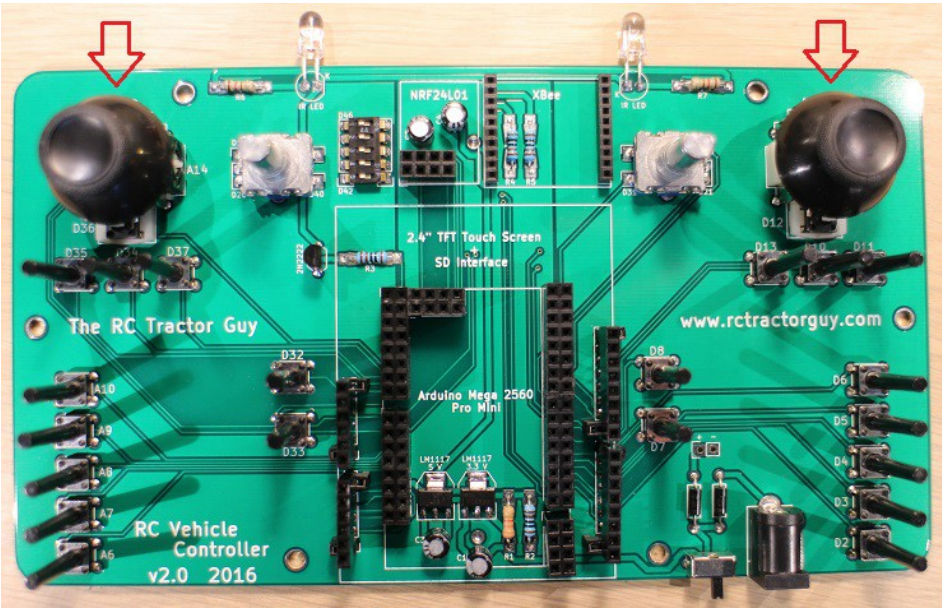
15. Push Buttons

The push buttons have curved legs which makes mounting them to the PCB easier.

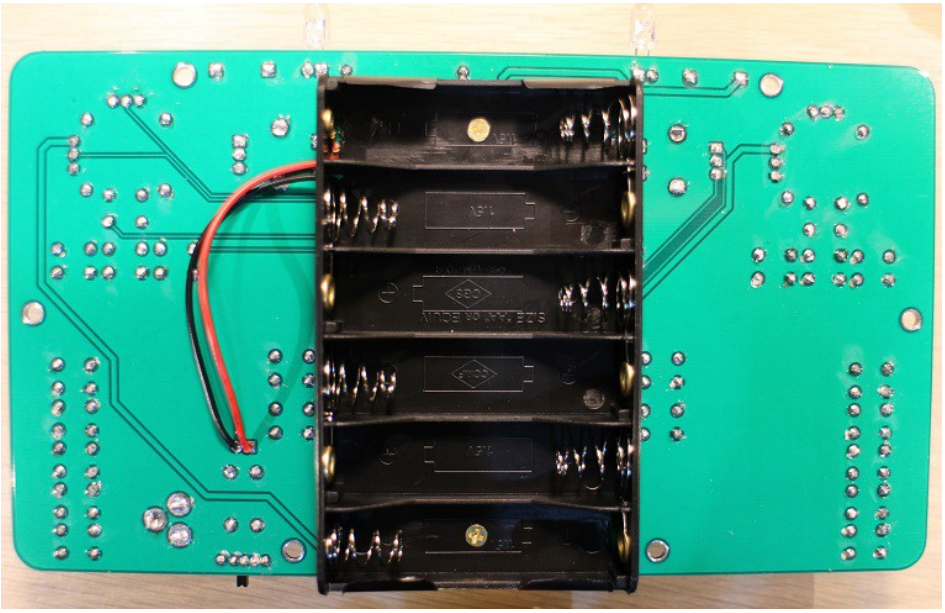
Solder one pin on each and check that they are flat to the PCB before soldering the remaining pins.



16. Thumb Stick Caps

The thumb stick caps simply push onto the thumb stick gimbals.	

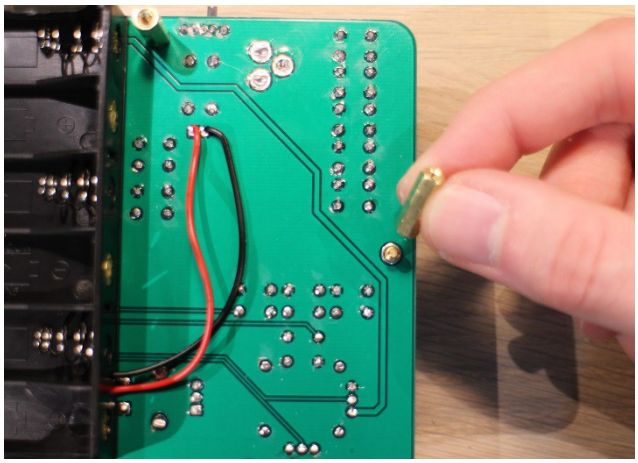
17. Battery Holder

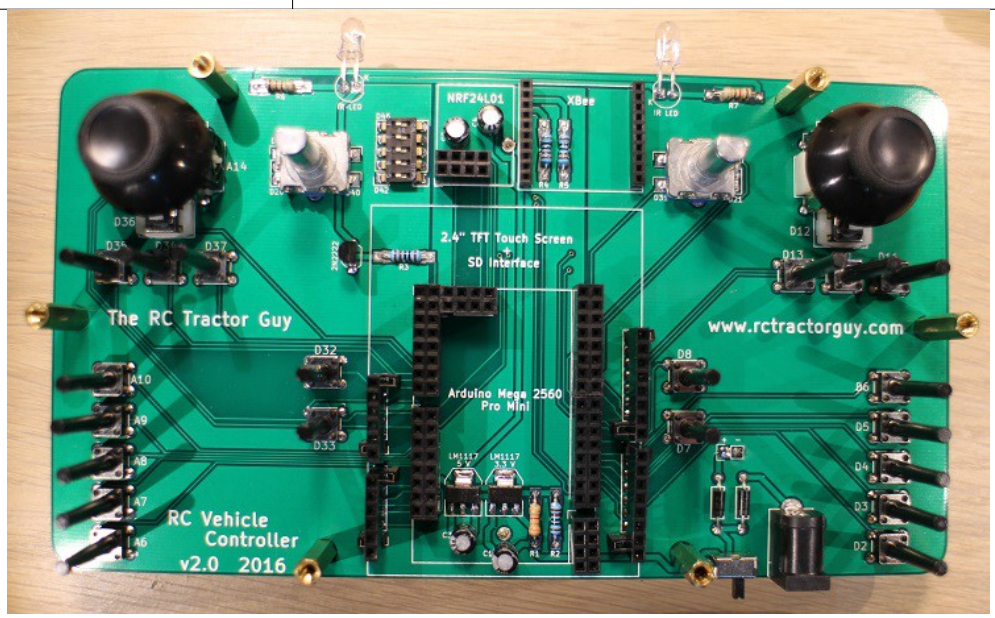
The battery holder is positioned in the center of the PCB as shown. Align it with the mounting holes in the PCB mount it with the brass screws provided. Connect the battery wires as shown. Positive and negative are marked on the front of the PCB.	

18. Front and Rear Panels

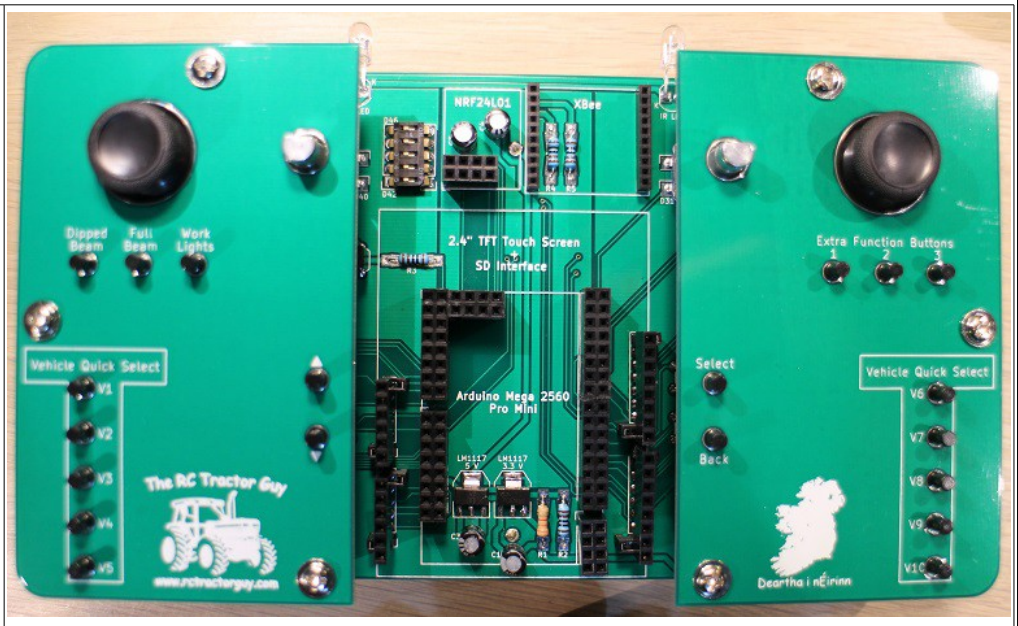
Mount the brass stand offs by pushing the male stand off through the front the mounting holes on the front of the PCB and screwing the female stand off on the back.

Do this for all six stand off pairs which can be seen below.





Screw the front panels in place.



Screw the rear panels in place.	

19. <u>Encoder Knobs</u>

The final step is to attach the encoder knobs and the end result should look like this.