

# Greenpower Speed Controller GpSpeed

Chipping Sodbury School – Rotary Racer

V1.1 – 2014-07-20 – T.Barnaby, L.Barnaby

Updated: 2016-06-02

## 1 Introduction

This document gives a description of the GpSpeed Greenpower motor speed controller developed for the Greenpower cars on instructions on how to build it. We have been using a, team built, motor speed controller in the Rotary Racer cars for a number of years. 3 different designs have been built and used. This is a new design that tries to bring all of our ideas together to produce a relatively simple, pupil build-able, speed controller which is flexible enough for Greenpower teams to use and allow for different uses as well as provide extendibility.



The idea is to provide a platform and possibly a kit allowing teams to get pupils involved in electronics

rather than provide a fully featured built end system. The basic speed controller can be extended with software and/or other hardware modules to provide a highly featured electronics control system for a Greenpower car. As well a speed control it offers battery voltage, current and other parameter monitoring.

A separate car computer/data-logging module can be attached to provide higher level control, data logging and telemetry etc.

This manual provides information on how to build, test and use the basic speed controller. There are a set of detailed build instructions and a resistor colour chart at the end of this document. More detailed information is available in the GpSpeed development manual.

Note we do not provide any warranty that these systems will work or guarantee any support. The design and any kits are supplied on an “as is” basis.

## 2 Overview

The basic operation is simple. The unit takes the 24 Volts DC power from the set of batteries and connects this to the motor via an electronic switch. The unit switches the motor on and off at about 20,000 times per second. It varies the amount of time the motor is on to the time it is off (the duty cycle of the pulse width modified (PWM) signal) to vary the amount of power that goes to the motor. The main power MOSFET transistors do this work driven by the microprocessor than generates the PWM motor on/off control signal based on the throttle position. See the Internet for information on PWM motor control.

Most of the rest of the circuitry is to provide a stable power supply for the control electronics, provide for cooling fan control, provide information on the voltage and current used and take in external information for extended control purposes.

On the website will be some information on how the unit works in more detail.

## 3 Specifications

Voltage	12 to 24V DC
Motor PWM Current	Typical 40 Amps, Peak 160 Amps, transient 640Amps
Motor Current Limit	Hardware 70 Amps approx (can be changed). Software controlled limit normally about 60 Amps but programmable
Soft start	Programmable.
Motor PWM Efficiency	At about 90 - 100% PWM, about 98%
Motor PWM Frequency	20kHz (Programmable)
Fan PWM Current	Max 1Amp 24V, filtered PWM at 20 kHz (Programmable)
External Power supply	12V at 500mA, 5V at 250 mA
Processor	Up to 32 MHz clock
Current sensor	Hall effect -100 to +100 Amps (10 bits ADC)
Voltage measurement	(10 bits ADC)
Speed control input	Analogue 0 – 5V (10 bits ADC) for potentiometer or Hall effect throttle
Spare IO	3 analogue/digital IO inputs/outputs
Display connection	With I2C communications and 2 digital IO lines
Communications	Serial port, I2C port

## 4 Changes

Version 1.1

Minor modifications as a result of feedback from the pupils and teams building the units as well as our own testing and review.

- Modified Picaxe programming circuit to conform to standard Picaxe circuit. Will probably be more compatible with older Picaxe download cables.
- Increased size of some surface mount pads to aid soldering.
- Added extra holes near test pins to allow wire loops instead of pins for easier soldering.
- Added some extra PCB labelling to help with capacitor orientation and modified some silkscreen graphics.
- Moved C21 closer to MOSFET's to reduce current spikes affecting hardware current limit setting.
- Moved R20 to other side of R17. This will improve resolution of secondary current measure system.
- Added some extra vias and tracks to strengthen power tracks and uncovered more PCB to allow thickening.
- Test points have been renamed TP\* and some of the connectors numbers have been changed.

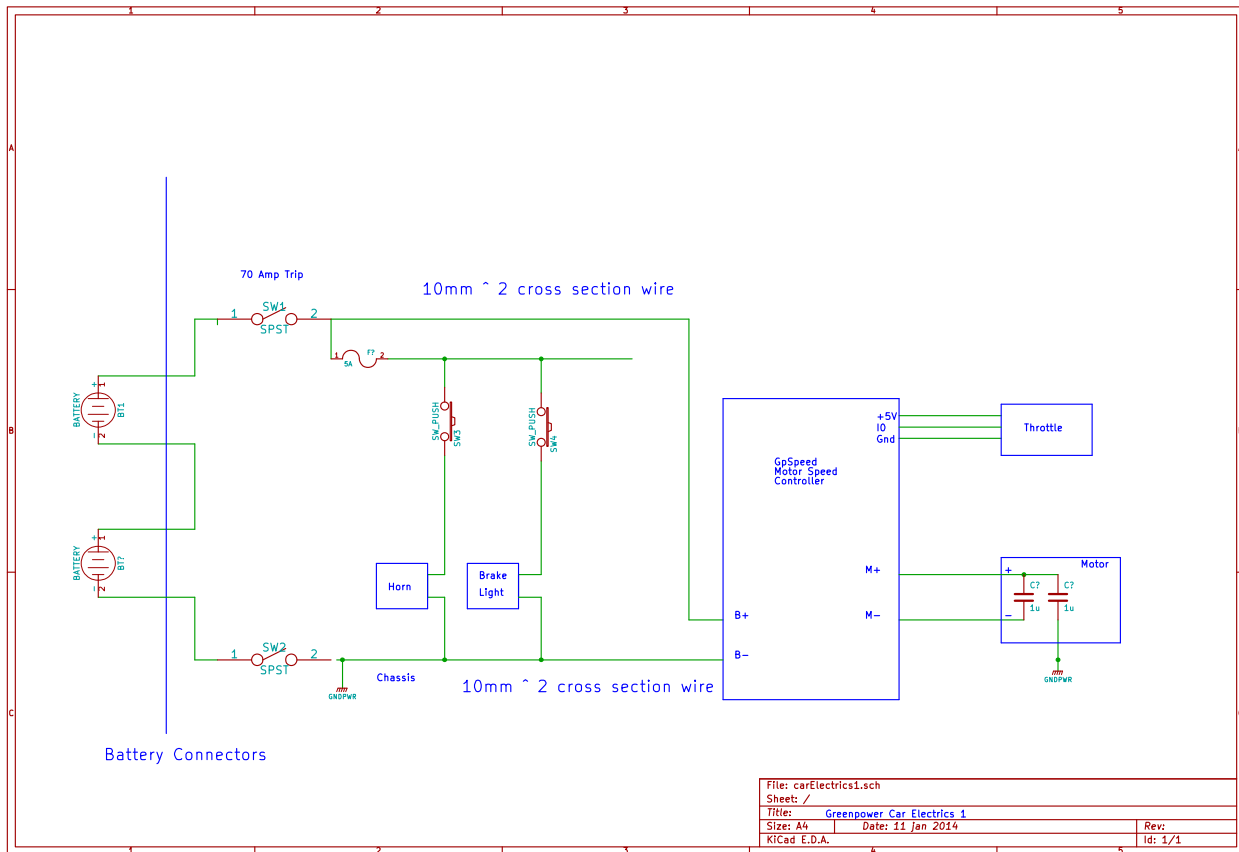
Note some of the pictures/drawings in this manual and on the website are for the 1.0 version and some minor differences will be seen.

## **5 Using the Speed Controller**

### **5.1 Car Connections**

This shows the basic connections needed to use the GpSpeed unit as a basic Greenpower speed controller.. Further info on all the other possibilities is in the extended GpSpeed Development Manual.

*\*\* Board picture with connections needed here \*\**



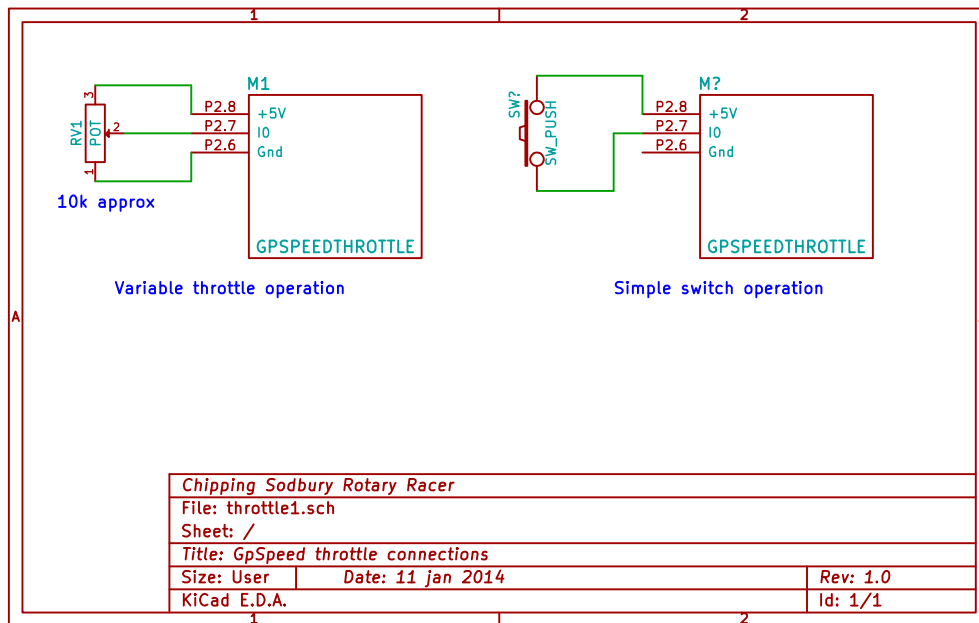
The main connections are:

Name	PCB Pin	Description
B-	P12.3	Battery negative and system ground
B+	P12.2	Battery positive switched and fused
M-	P13.2	Motor negative. This is the PWM switched output
M+	P13.1	Motor positive.
+5V	P2.8	5V supply for throttle potentiometer or switch
IO	P2.7	Throttle input. From 0 to 5V.
Gnd	P2.6	Ground for throttle potentiometer

This circuit diagram shows a complete circuit for a Greenpower Car using the gpSpeed controller. It is designed to be simple to use, no more difficult than using a relay, probably easier due to its use of screw terminals. The main batteries are on the left connected in series to provide 24 Volts. There are two main power switches, one SW2 in the -ve, ground connection accessible outside of the car and SW1 a current trip breaker accessible by the driver. Alternatively a 70 Amp fuse with switch could be installed in place of the breaker. It is ESSENTIAL to have a fuse or current trip breaker as near to the batteries as possible EVEN when testing on the bench. Batteries can generate a huge current that can be dangerous and damaging. It is also important to have a main power cut off switch close to the driver and easily used in case of speed controller or other electrical/electronic problems.

The gpSpeed has a number of extra connections that can be used if desired (and programmed for). See the development manual for more information.

Typical throttle connections are show below:



The GpSpeed heat-sink bracket needs to be bolted to a suitable heat-sink. Bolting it to the cars chassis or some aluminium sheet used as a case will normally suffice as the unit is quite efficient. Alternatively a small aluminium heat-sink can be attached.

## 6 Building The Unit

The unit is designed to be built by pupils in the year 7 to year 12 and above. The kit consists of a PCB and a set of components to be soldered and/or screwed to the board. There are many articles on soldering on the Internet, the following just lists the basics.

Try and get an electronics engineer to help you show the pupils and impart information on what the components are and what they do. But make sure they realise the idea is for the pupils to make the unit and be involved as much as possible.

A temperature controlled, high wattage, soldering iron (soldering station) is best as this will reduce the likely hood of damage to the PCB while soldering. Non temperature controlled ones can heat the board to a temperature that damages the adhesive holding the copper tracks to the PCB. Something like the following are fine:

Farnell:TENMA 21-10115, 2064549, £37

RS: RS AT60D, Soldering Station, 220V ac 60W, 799-8939, £66

Maplin 60W Mains LCD Solder Station, Code: A55KJ, £40

Also see: <http://www.pcb-soldering.co.uk/>

The tools needed are:

- Soldering iron. Ideally a high wattage, electronics temperature controlled type, this will reduce the likely hood of damage to the PCB. You do need a reasonable powered one (approx. 50W) for the track “thickening” at the end.
- Solder. Thin-ish multi-core solder.
- A “solder sucker” is useful when the soldering goes wrong !

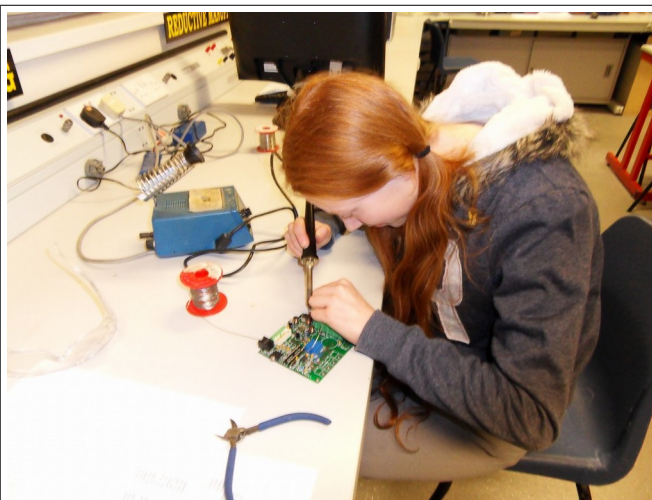
- A length of “solder wick” can also be useful to remove excess solder.
- Small side cutters.
- Small pliers.
- Screwdriver (cross head).
- A “helping hand” and/or PCB support tool can be useful to hold the board while soldering.
- A multi-meter to test the board.
- A 70 Amp fuse or 70 Amp current circuit breaker for testing.

Use the PCB layout diagram and the legends on the PCB in conjunction with the parts list to determine which components go where. There are detailed, step-by-step, instructions at the end of this manual that can be printed and items ticked off as they are done.

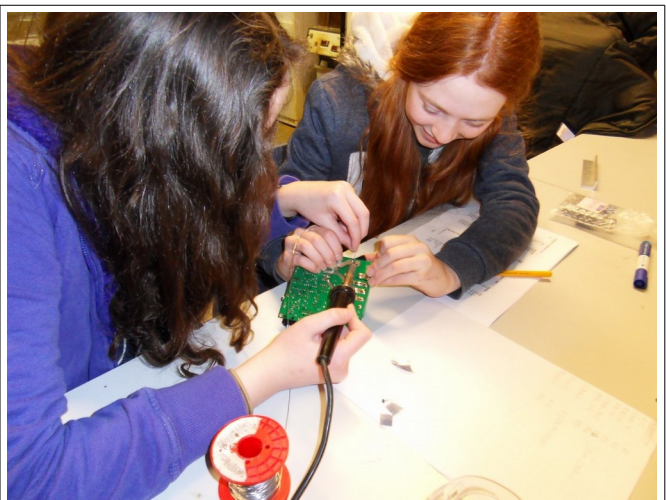
In general start by fitting the resistors, then capacitors, then IC's and connectors followed by the MOSFET's but have a look at the notes for specific information. The components list gives all of the components. Use the resistor colour band chart later in this manual to determine the values of the resistors. You can also check the resistor value by using the multi-meter set to its ohms/resistance setting, this is worthwhile to check that the colour bands have been read correctly. You can build a basic unit by not fitting some of the components. See the information below for components that can be left out in this case.

To fit a component bend the leads as needed using fingers or pliers and push the leads through the holes of the PCB. Once inserted bend the leads slightly to hold the component in place for soldering. Then turn the PCB over and solder the component in place.

Make sure the soldering iron's tip is clean (wipe on damp sponge and apply a small amount of solder to it). Hold the soldering iron's tip between the PCB and components wire and after a few seconds apply a small amount of the solder. Apply just enough solder such that a cone shape is seen, not a blob. Make sure the component and PCB are heated with the soldering iron to a level such that the solder melts onto these rather than put a blob of solder from the soldering iron onto the board. This should take about 5 seconds in all. Too little time and the PCB won't have heated enough for the solder to attach properly (a dry joint), too much time and there is the risk of damaging the component or PCB. This board has a ground plane of copper. When soldering components connected to this it will take more time for the PCB to heat up sufficiently.



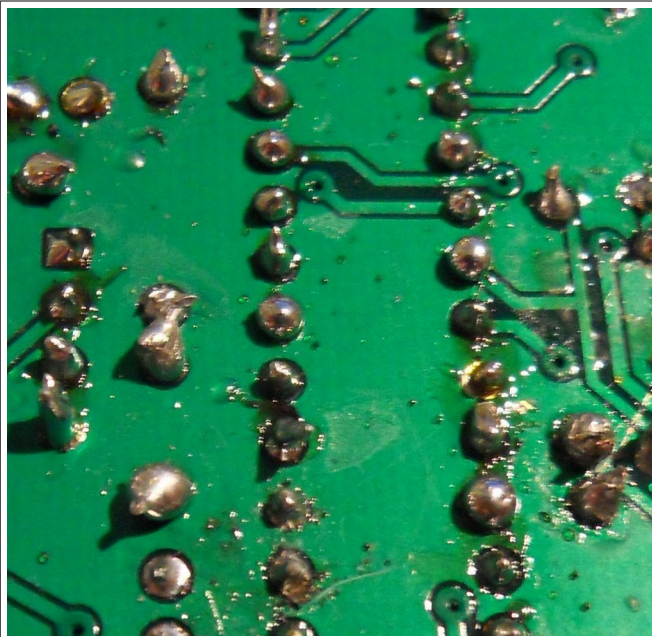
Pupils making up a unit.



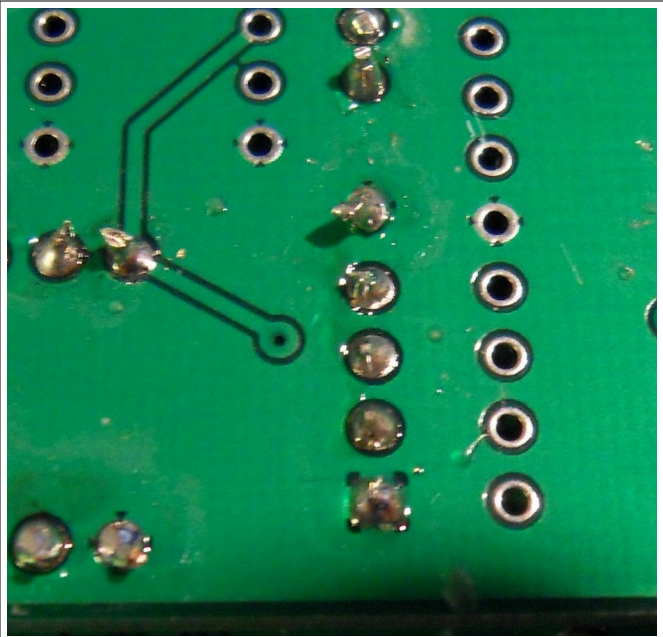
Pupils making up a unit.

The biggest issue with soldering I have seen from the pupils is applying too much solder. You only need the minimum amount to just produce a cone shape. Do some practice soldering on some veroboard or an old PCB first.





These have too much solder :(



These are about right :)

Finally use the side cutters to cut the leads flush with the solder.

For the few surface mount components, such as U3, place these in position and then solder one of the small legs to the board. Check the positioning (can be re-heated to move) and then solder in the remaining pins. Remember to solder the heat transfer pads if there are any keeping the soldering iron on long enough for the solder to flow under the component..

Notes:

1. Check the resistor values with an DVM after selecting them based on the colour bands. It is easily possible to read the colour bands incorrectly!
2. The speed controller can pass a high level of current. In order to do this it is worth “thickening” some of the PCB tracks with a layer of solder and/or a length of copper wire. Some areas of the PCB tracks have been left bare to allow this to be done. These should be added to the board near the end, but some components will need to be placed after this has been done. So leave the big capacitor C15 and the MOSFET gate resistors R21, R22, R23, R24 until the end.
3. Always use a 70 Amp fuse or 70 Amp current circuit breaker for testing. Batteries can generate a huge current that can be dangerous and damaging.
4. Solder in the 28pin IC socket for U1, the microprocessor, not the IC itself. The notch on the socket should match the one on the boards silkscreen print and the microprocessor.
5. Make sure the diodes and larger capacitors are fitted the correct way. The 100uF capacitors: C4, C11, C14, C18 should have their +ve wires fitted to the square PCB pads marked with a “+” symbol on the PCB. C15 should have its +ve pin nearest the main MOSFET transistors. **MAKE SURE THIS IS CORRECT!** The diodes and diode locations are marked with a bar where the cathode (arrow pointing to) should be.
6. The 8x2 header should be fitted with the key slot towards the nearest board edge (pin 1) side.
7. Install the surface mount components U3, U5, Q1 and S1 near the start, just after the resistors as it is easier to get to them to solder without the large capacitors and inductors that are near them.
8. Some of the connectors (P3, P6) use 0.1 inch pin arrays. Simply cut the lengths supplied to length and solder in place.
9. For the test pins: TP\*, you can use wire “loops” made from resistor leg cut-offs going from one hole to the adjacent one (on the same track). The single TP1 test point can use 1pin from the

0.1inch pin arrays or a made up wire loop. These can be left out if wanted. The idea is to provide an easy attachment for an oscilloscope for testing/viewing.

10. The big power connectors P7 and P8 should be slotted together before soldering into the board.
11. Put the main power MOSFET's Q2, Q3, Q4 and Q5 on last and fit them with the bolts to the heat sink bracket so they are held in place. Then are solder them in at the correct height off the board. The MOSFETS need to be electrically insulated from the bracket using thermal pads and plastic bushes. Make sure the brackets holes have been de-burred. You can us a 6mm drill turned by hand to take off the burr and chamfer the hole a touch. Finish by sanding with fine Emory paper/sandpaper and clean with a rag and spirit. Make sure the holes are clean.
12. When most of the components have been installed, some of the power passing PCB tracks need to be thickened up by flowing some solder over them and/or soldering lengths of copper wire on them. See the pictures for these. The wire will need to be bent and cut to fit. Tack solder these in place and then fill in with solder. You will need a reasonable power soldering iron ( $\geq 50W$ ) for this.
13. Fit the MOSFET gate resistors R21, R22, R23, R24 and finally install the large capacitor, C15, adding a layer of solder to the PCB track underneath to thicken the conductive path. Make sure the case of the large capacitor cannot short to the underlying track. You can use a thin plastic spacer here to be sure. There are some 3D printable items available linked from the GpSpeed website.

## 7 Component Values

Generally resistors, capacitors and inductors have their value printed on them using digits or colour bands. To reduce the number of digits the last digit of the main set of numbers is a  $\times 10$  multiplier. They also sometimes use a character ( $u = 10^{-6}$ ,  $n = 10^{-9}$ ,  $p = 10^{-12}$ ) to further define a multiplier. So for example the capacitor value 104 means 10 with 4 following 0's = 100,000. Capacitor values are normally given in pF ( $10^{-12}F$  so this value is 100,000pF or 100nF or 0.1uF).

Resistor colour codes are given at the end of this manual for both 4 and 5 colour band schemes.

## 8 Mechanics

The board requires the main Motor control MOSFETS to be bolted to a heat-sink using insulating pads and small insulating bush. We use a short length of 25x25mm angle aluminium 3mm thick to transfer the heat from the MOSFETS to a small heat-sink or car body part. This aluminium will need to be drilled for the MOSFET mounting holes and drilled for suitable heat-sink mount holes. See the mechanical drawing for this on the GpSpeed website. Make sure the MOSFET mount holes are de-burred and cleaned before bolting the MOSFET's to the heat-sink with electrically insulating pads and plastic bushes. You can turn a 6mm drill by hand in the holes to de-burr them and chamfer then slightly followed by Emory paper sanding and cleaning with a rag and some spirit. Make sure the inside of the holes are clean. It is worth using washers on the bush side and nylock nuts on these bolts.

Test that the bracket is isolated from the MOSFET tabs and the bolts using an Ohm meter. There is a DXF drawing of the angle bracket on our support website.



The board has 4 x 3mm holes, one in each corner for mounting. It can be mounted "open frame" or in some form of box as long as a degree of ventilation is provided to the electronics or the box is thermally conductive to get the heat out.

Note that the user wound big inductors can warm up. So some cooling airflow is useful here. The copper wire for these is 1.6mm. It is possible to use 2mm wire which will reduce the heating up of these inductors.

## 9 Basic Unit Components

If you don't want current monitoring you can leave out: U4,C8,C9,C12. You will need to solder a copper wire across the U4 power pins.

If you don't want the auxiliary fan control you can leave out:Q1, D2, L1, C4, R14 and R15.

If you don't want a display connection you can leave out: P1.

P6 can be left out if you don't want to program the processor on the board.

P4 can be left out if you don't want the ability to connect to a separate computer/data logger module.

## 10 Software Install

The PIC microprocessor will come with basic software installed that will allow the unit to behave as a simple variable speed controller with soft start and current limiting. Please see the development manual for how to produce and download new software or install software on a bare PicAxe or PIC chip. There is example software on our website.

Note, if you use the picaxe serial cable as a serial debug port (prints out things and allows user input from keyboard), then the picaxe cannot be programmed once the program has started. To program it in this case: Start the programming and then press the GpSpeed reset button. There is a small window of opportunity when the picaxe starts up where it can be programmed in this case.

## 11 Testing The Unit

The first thing to do is visually inspect the board for solder shorts and bad solder joints. Removal of excess solder can be done using a "solder wick". Especially check the orientation of the polarised capacitors and diodes.

Once this has been done use a multi-meter set to resistance to make sure there are no shorts. Connect one end to the B- (GND) terminal and check that the the resistance is greater than 1k Ohm to the B+,

F-, M- and M+ terminals. Also check the +5V/Vcc line by checking pin 4 of connector P2.

Measure the resistance between the aluminium heat-sink bracket and each of the MOSFET metal tabs and the MOSFET mounting bolts. It should be greater than 10kohms.

Ideally a small test board can be made up to help test and experiment with the unit. Some ideas for this are given below.

You can now connect a suitable potentiometer or all effect throttle to the accelerator inputs and a 12V to 24V power source to the B-, B+ terminals. Ideally start with a current limited bench top power supply (100mA to start then 3 Amps), if not use a 24V or 12V “brick” power supply or batteries connected through a low current fuse (5 Amps). Connect a load, either some bulbs or a Greenpower motor to the M-,M+ terminals. Make sure the PCB has nothing to short its connections. Ideally mount it with PCB pillars above a surface (MDF board would be fine).

When powering up start with the potentiometer at 0 (rotated fully anti-clockwise). The boards LED should flash about once per second. As you turn up the potentiometer the light should brighten or the motor should spin faster.

If you have an oscilloscope you can show the pupils the output PWM (pulse width modulated) waveform by connecting it between test pins TP5 (Gnd) and TP3. It should show a 20 kHz variable duty cycle PWM waveform varying with the potentiometer position.

When doing full current testing ALWAYS use a 70 Amp fuse or 70 Amp current circuit breaker connected as close to the batteries as possible. Batteries can generate a huge current that can be dangerous and damaging. Always get an adult to double check the battery connections to the GpSpeed controller and especially look out for reverse polarity connections.

If connected in reverse polarity the GpSpeed's MOSEFT;s act as a diode across the power rails and looks like a near short. Assuming a quick blow fuse/breaker is inline this can protect the GpSpeed controller. If no fuse is present a very large current can flow which will be dangerous. Suitable 70 Amp fuse: <http://uk.rs-online.com> 431-6725 and 548-9183.

Always use separately coloured wiring Red (or Brown) for Positive and Black (or Blue) for negative. Don't be tempted to reduce costs by using wire of one colour only.

## 12 Test Board

Ideally you can use a 24V DC supply for testing and experimentation, but 12V stuff is a bit more available and cheaper.

A small MDF board with 24V/12V power supply, and 12V 5W car bulbs (two in series for 24V) can be used. This is useful for testing and then experimenting with electronics test equipment such as multi-meters and oscilloscopes. It would also be useful for further development with other electronics modules and software development. A basic idea for this is:

- Use a small piece of 12mm MDF board as a base (300x200mm ?).
- Screw the gpSpeed controller to this base using 3mm bolts self tapped into the MDF through 10mm spacers.
- Mount and connect a 10 kOhm (linear) rotary potentiometer to the board to use as the accelerator.
- Use a “brick” 12V 4.0Amp (48Watt) mains power supply as a power source or a 24V/24Watt unit. Purchase a suitable connector with wires for this or cut of the power supplies 24/12V connector and wire directly to the GpSpeed's B-, B+ terminals. Something like: <http://uk.rs-online.com/web/p/desktop-power-supply/0435914/>. These can be purchased on Ebay for something like: £8.00. Or a 24V LED Driver Power unit could be used <http://uk.rs-online.com/web/p/led-drivers/7393510/> (Ebay £5).
- Use car 12V 5W side light bulbs with wires soldered to them as a Motor load and if wanted a Fan load. (Two bulbs in series for 24V operation). Small 24V bulbs such as these could be used:

<http://uk.rs-online.com/web/p/filament-indicator-lamps/6559794/>

- Suitable 70 Amp fuse: <http://uk.rs-online.com> 431-6725 and 548-9183.

## **13 Copyright and Warranty**

The design is copyrighted to the Chipping Sodbury Rotary Racer Greenpower team. It is provided free for non commercial educational uses.

Note we do not provide any warranty that these systems will work or guarantee any support. The design, information, software and any kits are supplied on an “as is” basis.

## **14 More Information**

There is more information on the CSS's Electronics website at:  
<http://www.greenpower.beamweb.co.uk/groups/electronics>

## 15 Detailed Build Instructions

Please read the build notes in the main manual section first. Additional pictures and videos of how to make it can be found on our website. It is worth printing this section of the manual and ticking of the components as they are fitted. It is also worth printing the Resistor colour chart and PCB component layout to help select components and locate where they fit.

### Step 1- Resistors

Find the resistors with the values by matching the right colour bands. Then match the resistor number with the one on the circuit board and solder the resistor into place. R28 can be a simple wire link (Use a resistor leg cutoff). Some resistors are left till the end.

Value $\Omega$	Resistor number	Colours 4 band	Colours 5 band
0	R28	Bl (Or use piece of wire)	Bl
2.2	R27	R R G G	R R Bl S Br
100	R2	Br Bl Br G	Br Bl Bl Bl Br
470	R5	Y P Br G	Y P Bl Bl Br
1k	R14	Br Bl R G	Br Bl Bl Br Br
1.2k	R26	Br R R G	Br R Bl Br Br
2.2k	R6,R7	R R R G	R R Bl Br Br
3.3k	R18	O O R G	O O Bl Br Br
4.7k	R9,R10,R30	Y P R G	Y P Bl Br Br
8.2k	R11	Gr R R G	Gr R Bl Br Br
10k	R19,R25	Br Bl O G	Br Bl Bl R Br
22k	R1,R3,R15,R17	R R O G	R R Bl R Br
47k	R4,R8	Y P O G	Y P Bl R Br
100k	R12,R13,R16,R20,R29	Br Bl Y G	Br Bl Bl O Br

Bl = black, R = red, G = gold, Br = brown, O = orange, Y = yellow, Gr = green, P = purple, S = silver

Note: Fit R21,22,23 and 24 late in the build, to allow copper wire soldered to board, to pass under the resistors.

### Step 2 – Surface mount IC's and Bits

Be careful soldering the surface mount components. They only need a little solder. Solder a single pin first then if they are not correctly positioned heat this joint and move the component with a small jewellers screwdriver until it is right. The rest of the pins can then be soldered remembering to solder the large heat sink pads if the component has these.

Name	Number
RFD14N05LSM	Q1
SWITCH_PUSH	S1
IR2125	U2
HXS50	U4
DIL-28-300	U1 (Just the 28pin socket)
LM2575	U3
MC78M05CDTRKG	U5
LED	D1 (Short lead to flat side of PCB silk screen's circle)

### Step 3- Capacitors

Value	Number	Markings
-------	--------	----------

1n	C7	102 (or 102k)
10n	C13	103
100n	C6,C8,C9,C12,C16,C19,C20	104
470n	C17,C21,C22	474
1u	C1,C2,C3,C5,C10	105
100u	C4,C11,C14,C18 (+ve wires fitted to the square PCB pads marked with a "+")	100uF

#### Step 4- Diodes

Name	Number
VS-11DQ10	D2,D4 (Note bar indicates cathode end on PCB and diode)
BAT41	D3(Note bar indicates cathode end on PCB and diode)

#### Step 5- Mini Inductors

Name	Number
680u	L1,L2,L3

#### Step 6- Connectors

Name	Number
RJ12	P1
CONN_8	P2 (wire holes to board edge)
CONN_8x2	P4 ( key slot towards the nearest board edge)
Jack_3.5	P5
PINS4	P3 (break of set of pins)
PINS6	P6(break of set of pins)
Power3	P7 (slots together with P8)
Power2	P8 (slots together with P7)
CONN_1	TP* (Just for test/debug/playing with)

PINS4 and PINS6 are both from CONN\_1. Break it to get the amount of pins it says (6 or 4) and solder it in to place. For the TP\* test points use cut-off resistor legs bent in a "U" loop and soldered between the adjacent test points (on the same track). TP1 will need a single pin of CONN\_1 or a wire loop made up from resistor leg. These don't need to be installed they are for testing/playing with the circuit. With Power3 and Power2, before you solder them in, slot them together at the sides.

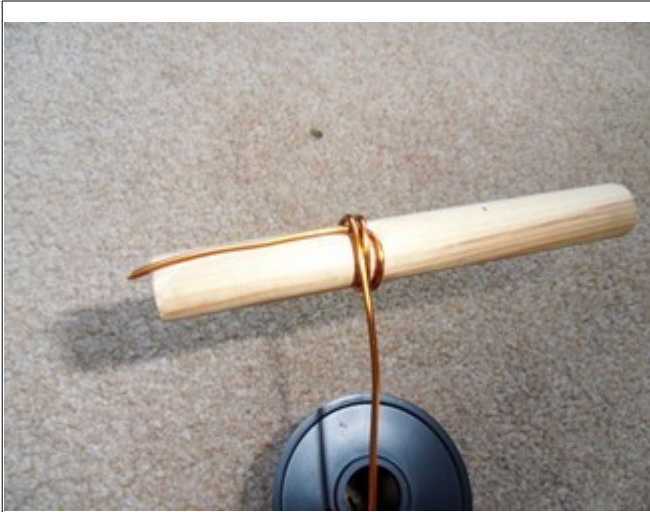
#### Step 7- Self wound Inductors

There are two identical ones of these that go in, L4 and L5. To make these you need to get a wooden rod at the right diameter (about 13mm) and wind 8 turns of the thick enamelled copper wire (1.6mm or 2mm diameter) around it, with both ends going down. It is worth cutting a groove in the wood for the inner vertical wire to PCB to make the coil more round. Make sure it is no higher than about 4 cm so it doesn't go above the highest components on the board.

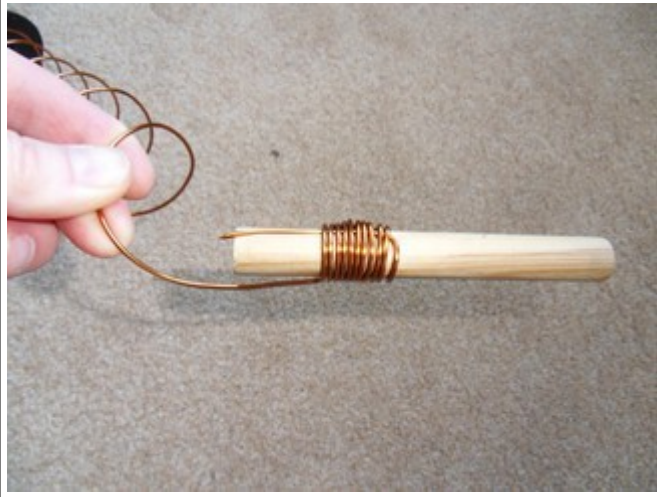
The enamel must be removed from the wire were they are soldered into the board. This can be done by scraping with a Stanley knife's blade and then using some sandpaper to finish off. Tin the wire with solder to make sure you have cleaned off the copper before soldering into place.

There will be a video on our website that explains this more thoroughly.





Winding the inductors

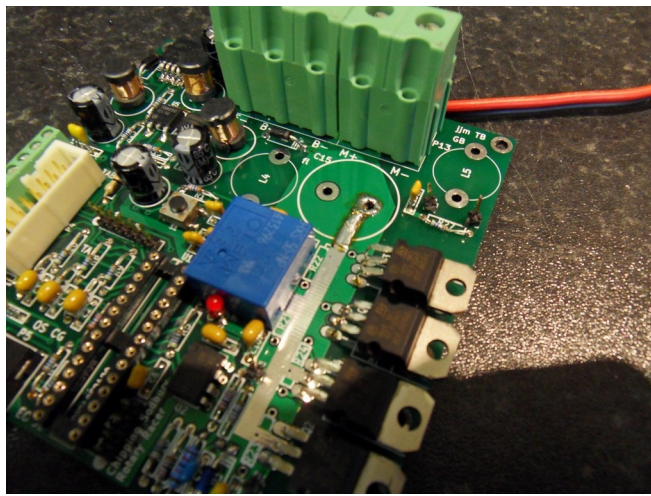


The finished inductor

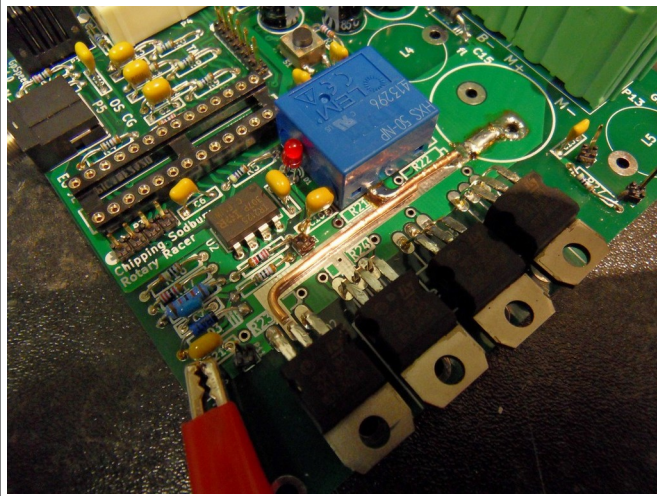
### Step 8- Thicken the Power Tracks

The board carries quite a lot of current. For efficiency it is worth thickening the tracks that carry the main motor current. You can simply add a layer of solder over these tracks. A better way is to use medium thick non insulated copper wire to strengthen these main power carrying tracks. The earth conductor from 32Amp ring main cable is ideal. The copper wire should be bent and cut to shape and then soldered on the board. The pictures at the end will show you which tracks need to be strengthened. A relatively high powered and ideally temperature controlled soldering iron (50W) will be needed for this. Under C15 the copper plane should have a layer of solder but no copper wire added so that the capacitor can sit on the board better. Make sure there are no sharp bits here and it is worth adding an extra layer of insulation/protection between the capacitor and thickened solder track at this point to make sure they cannot short with wear and tear.

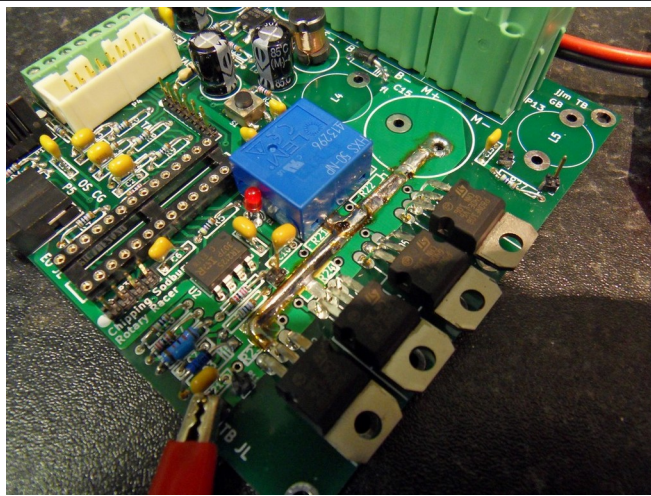




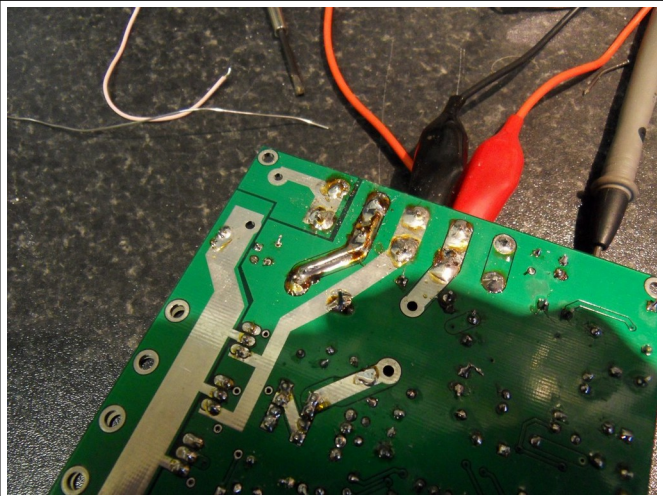
Thicken the track under the large capacitor with solder all the way to and around the pin hole



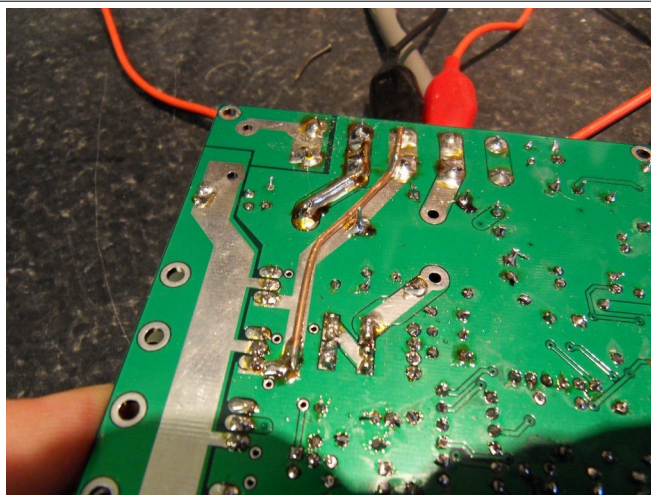
Bend, cut, lay and solder the thickish copper wire onto the PCB track.



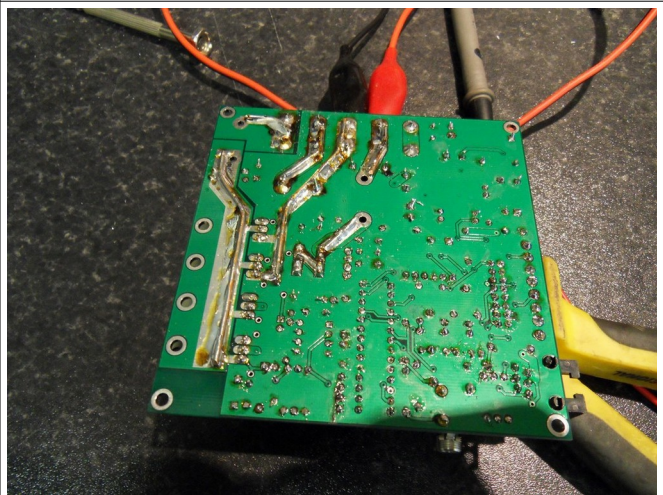
Top side board finished



On the rear of the board the first one.



Thickening the second track.



The completed thickening of tracks.

### Step 9- Bracket and MOSFETs

Make the bracket as per the drawing. Make sure there are no metal burrs. De-burr with a 6mm drill by hand and use Emory cloth followed by a rag with some spirit. Make sure the holes are clean. Place on the PCB and place one of the heat sink pads in position. Bend MOSFET legs and fit. Bolt it in place using the nuts, bolts, washers and bushes. Repeat this for each of the MOSFETs: Q2,Q3,Q4,Q5. Then solder them in and cut off the extra part of their legs. Make sure the solder extends to the thickened

areas of the tracks.

Name	Number
STP80NF10	Q2,Q3,Q4,Q5

### Step 10- End components

Name	Number
47Ω Resistor	R21,R22,R23,R24
2200u Capacitor	C15 (+ve pin nearest the main MOSFET transistors. MAKE SURE THIS IS CORRECT!)
PicAxe 28x2 or PIC18F2420	In the DIL-28-300 socket (U1) pin 1 and the top (notches line up)

Now the board needs testing. Thank you for choosing Rotary Racer enterprise's :)

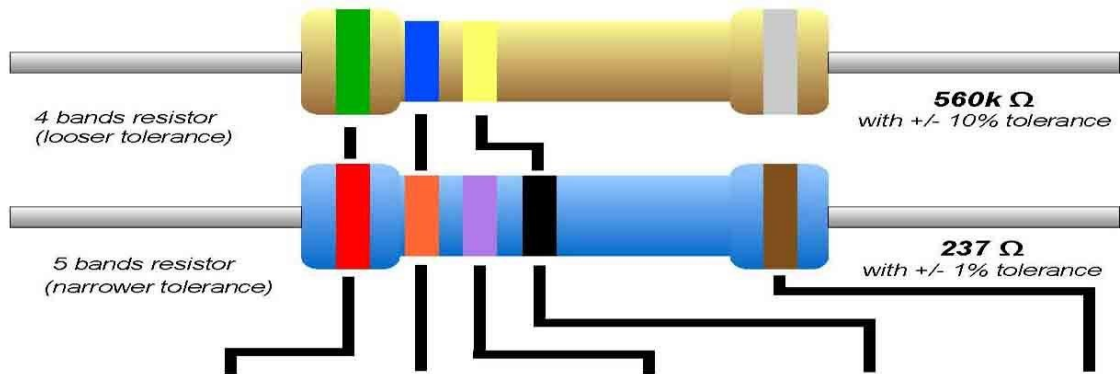
## 16 Notes

1. The board has a hardware current limit feature. When this triggers it switches of the output power until a new PWM pulse arrives. If the power is set to 100% and the hardware current limit triggers the GpSpeed will switch off. Easing off the power and then bringing it back up will restart the output. If this happens the value of resistor R18 can be reduced in value to raise the over current tripping point. Normally the software would limit the current before this point is reached.
2. Always get an adult to check over the unit and test it before using it in a Greenpower Car.
3. When using electronics to control the motor in a Greenpower car. Always have a physical main power switch easily accessible to and known to the driver in case of problems.



# 17 Resistor Colour Chart

## Resistor Color Code



Color	1 <sup>st</sup> Band	2 <sup>nd</sup> Band	3 <sup>rd</sup> Band	Multiplier	Tolerance
Black	0	0	0	x 1 $\Omega$	
Brown	1	1	1	x 10 $\Omega$	+/- 1%
Red	2	2	2	x 100 $\Omega$	+/- 2%
Orange	3	3	3	x 1K $\Omega$	
Yellow	4	4	4	x 10K $\Omega$	
Green	5	5	5	x 100K $\Omega$	+/- 5%
Blue	6	6	6	x 1M $\Omega$	+/- .25%
Violet	7	7	7	x 10M $\Omega$	+/- .1%
Grey	8	8	8		+/- .05%
White	9	9	9		
Gold				x .1 $\Omega$	+/- 5%
Silver				x .01 $\Omega$	+/- 10%

# 18 PCB Component Layout

