DANGER

OSMC

Xtreme Motor Control

Supporting documentation for the OSMC project

Written in part by:
Sonny Lloyd – slloyd@engineer.com
Matt McFadden, FAQ
Don Jennings, (working with Lexan) - dasjennings@msn.com
Robert L. Doerr (assembly) – **http://www.robotworkshop.com**

and from many helpful additions from the
crew at the site listed below

Edited by Chris Baron

Development archive & ongoing support can be found at:

http://groups.yahoo.com/group/osmc

Open source promotes software reliability and quality by supporting independent peer review and rapid evolution of source code.

December 24, 2001, revision 4.2

Table of Contents

OSMC Project License

With the exceptions noted below, all materials of the OSMC project are covered under the GNU Public License. The copyright for the project material is owned jointly and non-exclusively by members of the OSMC discussion forum hosted on Yahoo Groups™. The full license can be found in the README.txt file that is included in the OSMC package. It can also be found at: **http://www.opensource.org/licenses/gpl-license.html**

Please note that the GNU General Public License covers the entire OSMC project excluding the following (which has its own different license):

- AVRx Kernel
- avrx_tasking.s
- MOB & uMOB software

For licensing information on these topics please email Larry Barello at **larry@barello.net**

Please note that joining the OSMC discussion forum requires you to join Yahoo! Groups™ and sign-in using your own Yahoo! user ID and password. Yahoo! has its own Privacy Policy & Terms of Service regarding this issue which can be found from links on the group's home page: **http://groups.yahoo.com/**

OSMC Introduction

Open **S**ource **M**otor **C**ontrol is an ongoing design project who's goal is to produce high-performance low-cost electronic speed controls for a Direct Current permanent magnet motors.

The project began in late 1998 founded by a group of robotic combat enthusiasts who were dissatisfied with the commercial offerings available at the time. This group wanted a motor speed control unit designed specifically with the needs of robotic combat in mind. After languishing for several years the project was revived in early 2001 by Dennis Millard and Chris Baron, two founders of the original OSMC group. The project first found web space provided by Dennis Millard, http://www.dmillard.com/osmc/index.html.. His insightful words read:

> For the average hobbyist who is trying to design and build a robotic platform, there is a serious need for a motor controller. Many commercial units available are expensive, and many R/C hobby units are inadequate. A group on the RobotWars Delphi Forum decided to apply the open source software development model to the design process of a high quality speed controller that could be built for a reasonable cost, and would be freely available for anyone to tinker with, improve upon, and do whatever they want with it…
>
> -Dennis Millard

The OSMC motor control system as designed is split into two parts. The key part is of course the high-power H-bridge board. This board is a "dumb" controller accepting pulse-width modulation signals at the inputs of the bridge driver circuitry. The intelligence to convert the speed commands into these PWM signals is provided by an external source. This might be a dedicated microcontroller, a PC, or some other signal source. In order to provide a complete speed control solution a second board in the OSMC project was developed – the Modular OSMC Brain or MOB. This is a highly flexible microcontroller platform intended to provide PWM signals to two OSMC boards from a variety of input sources. A block diagram of the OSMC system can be seen below:

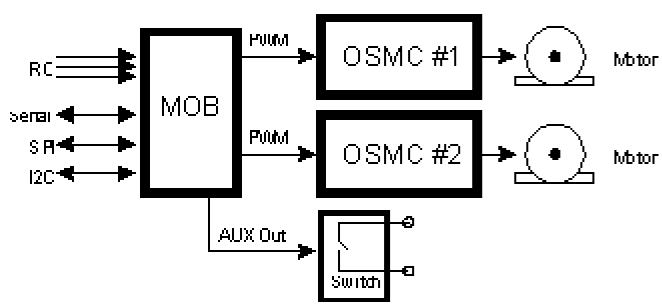


Figure 1. OSMC Project Block Diagram

So far the results of the project have been better than expected as more and more boards are built and used and people report their experiences with the system. The latest information for all material related to the OSMC project is available from its official home at:
http://groups.yahoo.com/group/osmc

There you can review the development process, read archives of the groups discussions, download all schematics, software & PCB layouts, get answers to any questions that you may have (please refer to the FAQ section first) and find other relevant links and pictures; all free of course! The group is open to all and welcomes new members to contribute or just “lurk” to learn and follow the development of this exciting project.

Enjoy :)

OSMC Board Description

This section provides a moderately detailed description of the OSMC board and its major components. Readers should refer to the schematic diagram for full details on the OSMC circuitry and to the PCB diagrams for parts placement information.

The heart of the OSMC board is the HIP4081A bridge driver chip made by Intersil (formerly Harris). This chip is a monolithic full H-bridge driver chip that includes both high- and low-side FET drivers and all needed voltage boost circuitry. The HIP4081A can accept main battery voltage from 12 to 80V and generates all needed signals and voltages to drive an H-bridge. Figure 2 shows a block diagram of a 4081A chip application.

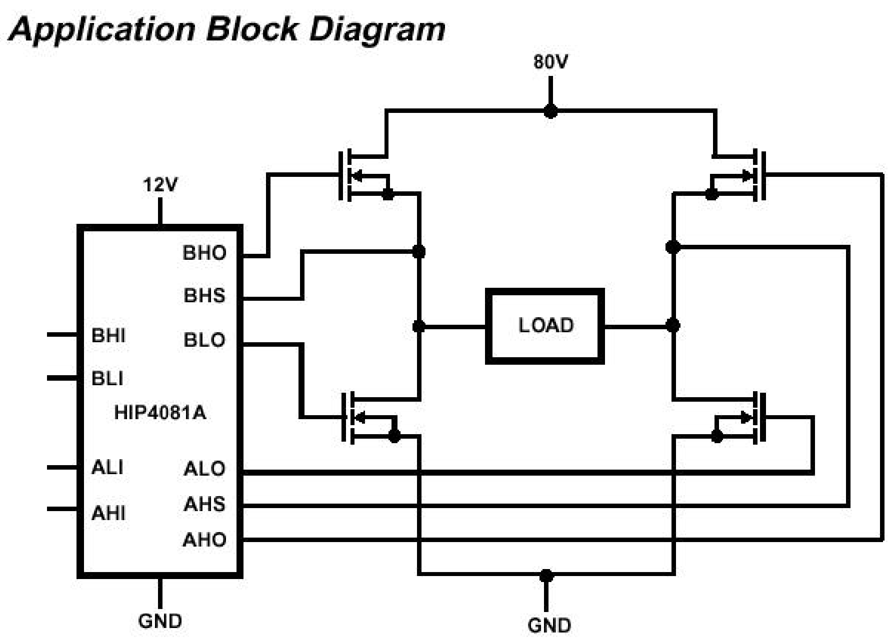


Figure 2: HIP4081A Block Diagram

As you can see the HIP4081A has four inputs that correspond to the outputs used to switch each leg of the H-bridge. An additional disable output is not shown on this diagram. The digital signal source must provide the PWM signals into the 4081A inputs in order to drive the bridge. The input lines of the 4081A are “modified TTL” in that a high level signal is any voltage between 3 and 12V. This allows a wide variety of signal sources to be used to drive the chip.

Because of the nature of the N-channel MOSFET used in high-power H-bridge circuits, the bridge driver must provide approximately 10V above the positive voltage source into the gate of the high-side FETs to turn them on. The HIP4081A can provide up to 90V drive voltage on the high-side output AHO and BHO. It does this through a capacitor switched charge-pump subsystem. Using only an external diode and capacitor, the 4081A generates the required voltages to drive the high-side FETs.

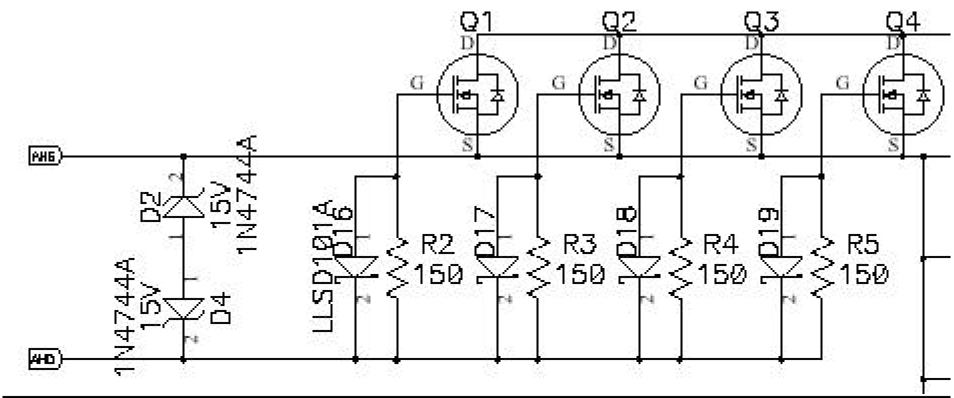


Figure 3. OSMC Gate Drive

The gates of MOSFETs such as those used here are very sensitive to high and low voltages. A few volts too high or low even for an instant can destroy the FET. To protect the gates of the FETs zener diodes are added to the circuit. Two diodes are used in order to clip both high and low voltages. Since the gate of the MOSFET acts like a capacitor, voltage spikes may be generated while rapidly charging or discharging the gate capacitance when switching the transistor. These are due to the dI/dt effect of rapidly change current. The zeners clip these transient voltages and protect the gates.

The 4081A can source/sink up to 1A through the gate drive outputs. For large FETs such as those used in the OSMC, where several are used on each leg, the capacitance of the gates is such that a very large current can be drawn from the gate driver of the 4081A and may overheat and destroy the chip. To limit the gate drive current to something reasonable, small resistors are added between the 4081A and the gates of the FETs. These resistors both balance the gate current into each FET and limit the total current drawn out of the 4081A. While these resistors help balance the turn-on/off of the FETs in each leg, they can contribute to another problem with H-bridge circuits.

Another major problem with H-bridge circuits is called “shoot-through” this dreaded condition occurs when both high- and low-side FETs are turned on at the same side of the bridge. As you can see from the Figure 2 this presents a short-circuit condition to the battery through the FETs and can rapidly destroy them. If the shoot-through is only very short duration the FETs may survive but excessive heating will occur. To prevent shoot-through in the OSMC several things are done. First, the 4081A has shoot-through protection circuits built-in. It is not possible to command the bridge into shoot-through because logic internal to the 4081A turns the high-side driver off whenever the low-side output on the same side of the bridge is commanded on regardless of the state of the high-side input. Even with this protection shoot-through can still occur. This is due to the fact that it takes a non-zero amount of time to charge and discharge the gate capacitance. MOSFETs tend to turn on much faster than they turn off. So you can see that if the high-side is commanded off simultaneously with the low-side being commanded on, there is a possibility that the high-side FETs wouldn't be fully off when the low-side FETs turned on – result shoot-through. The OSMC ensures this won't happen in a couple of ways. First, there is a delay pin on the 4081A that delays the turn-on of the transistors for some amount of time after being commanded on. This is to balance the slower turn-off of the FETs. In extreme cases such as with the OSMC this delay is not enough so the OSMC circuit includes schottky diodes connected in parallel with the gate resistors. The diodes are oriented to conduct when the gate is being discharged i.e. turned off. This greatly speeds up the turn-off time of the FETs and eliminates shoot-through as a concern for the OSMC.

Voltage spikes coming from inductive loads such as motors and high-frequency noise from brushes and commutators are also problems in DC motor drives. The OSMC handles these with devices called Transient Voltage Suppressors (TVS). These devices can be considered "super Zener diodes". They are optimized for handling high-current voltage spikes safely. They are connected in the OSMC to clip spikes across the battery leads and to protect the FETs from voltages exceeding their Drain to Source breakdown limit. Additional protection from high-frequency spikes coming from the motor brushes is provided in the form of an RC-snubber network across the motor leads formed by a low value resistor and a small high-frequency polyester capacitor. Finally gross filtering of the supply voltage is done by large electrolytic capacitors.

The rest of the circuitry on the OSMC is dedicated to providing a stable 12V source to the 4081A. This is done by an LM2574-HV12 voltage regulator. This chip is a step-down switching regulator which provides much higher efficiency when dropping high battery voltages down to 12V than a linear regulator. This regulator also supplies 12V off-board through the interface connector to the digital logic driver circuit. This eliminates the need for a secondary power supply for the microcontroller or other logic source.

This concludes the description of the OSMC H-bridge circuit. A lot of time and effort was spent on making this board as robust and reliable as possible. Some of the protection components may seem excessive or unnecessary but experience has shown them to be needed for reliable operation under the extreme conditions of robot combat.

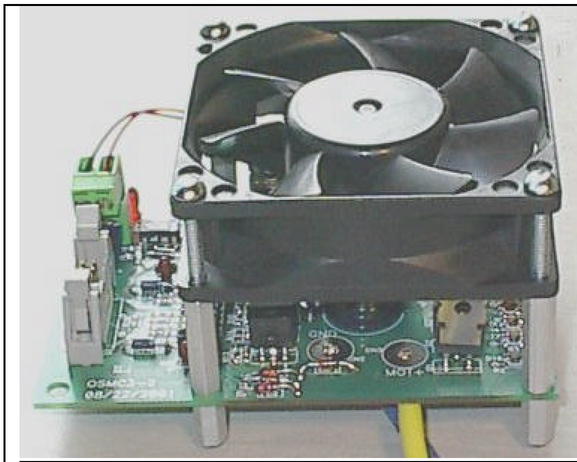
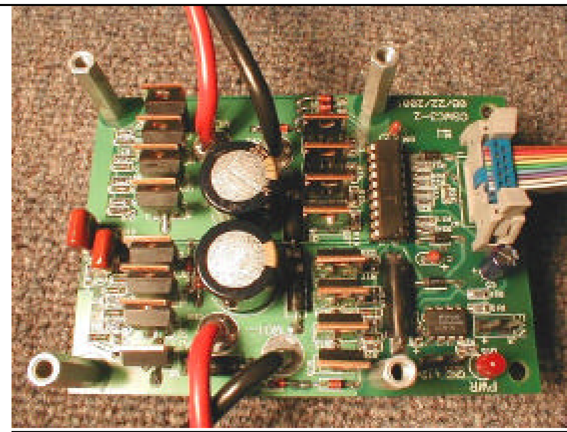
CIRCUIT DESCRIPTION

The OSMC project is a replication of a device more commonly known as an "H-Bridge". Generally, it controls the direction of current through the DC motor thus forcing the rotor to turn clock-wise (or counter clock-wise). It should be powered with 12V to 50V. For maximum current draw specification, please see the FAQ section below. This section will describe the H-Bridge's components, why they are used and any comments. For easy reference, it is convenient to have the circuit schematic (most current version) printed or opened in Adobe Acrobat while reading through this section.

Part Number	Purpose	Comments
C1 & C8	Filter caps used to minimise the voltage ripple from the power source.	Paralleling these capacitors allows for double the capacitance while keeping the physical height of the capacitors close to that of the FETs. Battery voltage **must** be kept below the rated 63V or the caps will explode. The battery supply maximum voltage must be < 50V
D1, D6, D7	The TVS diode is used to suppress voltage spikes. These are like super zeners optimised to clip high current pulses that exceed the voltage limit of the device	TVS stands for Transient Voltage Suppresser. The bi-directional ones handle pulses of either polarity. They can handle much more power on a repeated basis than a Zener and are used to protect the Fets
Q1 through Q16	Mosfets receive the "on" pulse at the Gate "G" which then allows current to flow from the Drain "D" to Source "S" of the mosfet.	The OSMC has 4 MOSFETs per leg. With the fan, each MOSFET can handle about 40A continuously. This gives a total continuous current capacity of around 160A. Fewer FETs may be used to save cost where the maximum required current is less.
D2,3,4,5, 8,9,10,11	The zeners are used to clamp the gate drive voltage (from the 4081A) to approx. 15V.	These diodes protect the gate from transients caused by the rapid charge and discharge of the gate capacitance.
R2 through R20	These resistors, placed in series with the MOSFET's gate are used to slow down the turn-on rate of the MOSFET	If the Mosfets turns on before the opposite(high- or low-side) leg of Mosfets on one side of the bridge has a chance to turn off, then "shoot through" is experienced. This is bad.
D16-31	Schottky diodes provide a path for the MOSFETs to rapidly dissipate their energy, i.e. turn Off.	The Schottky diodes work in conjunction with the series gate resistors (R2 through R20) since they are both used to ensure that the MOSFETs of opposite leg are not on simultaneously. They held the gate discharge (turn off) very rapidly while the resistors slow the turn-on

D12 & D14	Source Return Diodes,	These diodes recover charge for the voltage boost circuit when the upper FETs are turned on. This allows the boost circuit to start at nearly the positive battery voltage and work less hard to boost the drive voltage 10V higher.
R1/C2 & R10/C3	Motor RC-snubber networks	These components form an RC filter across the motor leads. They absorb high-frequency noise and spikes to keep the battery supply voltage clean.
D15	Power indicating LED	A red 5mm LED
Molex 70227 Header	Connects to the MOB via data cable and receives the data.	Use a shrouded polarized connector with locking tabs for maximum reliability under heavy vibration and shock loads.
HIP4081 A	Full-bridge MOSFET driver	Our favorite FET driver. Also used in some of the Vantec controllers. Refer to HIP4081A Datasheet, found in FILES section
LM2574H VN-12	12V Voltage Regulator	Switching step-down regulator. Highly efficient at dropping high battery supply voltages to a stable 12V for the 4081A. Refer to Datasheet, found at www.nsc.com
L1, D13, C5	2574 support components	These components are required for operation of the 2574 switching regulator.

CIRCUIT ASSEMBLY (Kit version only)

	
Finished OSMC with cooling fan	What a great looking rug! Oh… and board too

- If the OSMC is already assembled, please proceed to **Mounting the OSMC Boards**.

When the design for this circuit was first finished a board order was made to a company to have some professional PCB layouts done. Since these boards are rated for some serious amperage, it is <u>not</u> recommended that you attempt to etch the board yourself. There are many web based companies that specialize in soldering boards that you can take advantage of for a reasonable price.

This kit should be built using standard construction methods. The following items are required to build the kit: diagonal cutter, needle nose pliers, reverse action tweezers, soldering iron (pencil type), 60/40 Rosin core solder, liquid flux, solder wick, and some patience. During the Assembly process, keep in mind that the Component side of the board has the silkscreen. All the components will be mounted on the Component side of the board. For the following steps, the 10-pin header for the MOB (or uMOB) board should be on the left. After installing each component at the specified location solder it in place before proceeding to the next.

Before getting started with the circuit assembly you should take the time to make sure that you are grounded. The FETs are static sensitive so if you have carpet, do not walk across it without touching a ground when you get to your worktable. A common place to find a ground near your worktable is the center screw of an outlet cover. By scraping off the paint and touching it with both hands you reduce the chance that your body is carrying a static charge, which could save your component's life!

<u>NOTE:</u> When installing each of the surface mount components put a small amount of solder on one pad. Next, warm the solder on the pad and position the part with a pair of tweezers. To make sure the part is seated you can gently press down on it while warming the solder pad with the soldering iron. If the part

shifts or does not line up properly then you can warm it up again and reposition the part with the tweezers. Once the part is in place solder the other side. If needed re-solder the first pad again using some fresh solder. Dab a little flux on the pads, take out the parts and place them on the pads with tweezers. Then hold them steady with tweezers while you solder one side quickly to tack it down. Be sure to clean your soldering gun by dabbing it onto a wet sponge after EACH solder attempt. Turn the board so you don't have to bend your wrist all around to get the tip of the iron against the end of the part. Tack them all down then flip the board around and to the other side with more solder to get a nice looking joint. It doesn't take much time or heat on those little guys. Then go back and finish soldering the sides you tacked down first.

- ❑ Install the 680 ohm resistor (SMD) at location R29. Polarity does not matter.
- ❑ Install the 90.9K ohm resistor (SMD) at location R12. Polarity does not matter.
- ❑ Install six 10K ohm resistors (SMD) at locations R11, and R21 through R25. Polarity does not matter.
- ❑ Install the 14K ohm resistor (SMD) at location R28. Polarity does not matter.
- ❑ Install two 249K ohm resistors (SMD) at locations R26 and R27. Polarity does not matter.
- ❑ Install two 33 ohm resistor (SMD) at locations R1 and R10. Polarity does not matter.
- ❑ Install sixteen 150 ohm resistors (SMD) at locations R2 through R9 and R13 through R20. Polarity does not matter.

NOTE: The biggest "gotcha" is probably the surface-mount Schottky diodes. These have a green stripe on the cathode (negative) side. If you look at the silkscreen carefully, you will see an extra bar on one side that is the cathode mark. Make sure the green stripe is on the same side as the silkscreen bar. D23, D26, and D27 do not have the cathode in the same direction as the other diodes (they are rotated 180 degrees). The silk screen on the board is not too clear on this; so try to find the same part on the paper silk screen included with the boards and use that as your guide.

- ❑ Install sixteen LLSD101ACT Fast Schottky diodes (SMD) at locations D16 through D31. Make sure the banded (green) end goes toward the bar on the silkscreen.
- ❑ Install the .47uf capacitor (SMD) at location C7. Polarity does not matter.

NOTE: After installing each through hole component on the board solder each lead and trim them flush with the back of the board.

- ❑ Install three UF1002 diodes at locations D12, D13, and D14. Make sure the banded end goes to the right as the silkscreen on the board shows.
- ❑ Install eight 1N4744A 15v zener diodes at locations D2 through D5, and D8 through D11. Make sure the banded end goes the same way as the silkscreen.

- ❑ Install three 1.5KE51 TVS diodes at locations D1, D6, and D7. Polarity does not matter.
- ❑ Install the RED LED at location D15. Make sure the flat (side with short lead) goes down.
- ❑ Install the 330uH inductor at location L1. Polarity does not matter.

NOTE: On DIP components pin 1 is usually designated by a square hole on the board. When installing U1, U2 it works best if each of the two opposing corners is soldered first. Next, gently press on the center of the component while warming the solder on those corner pins with the soldering iron to make sure the component is properly seated. Finally solder the rest of the leads on the part. As an option you can install an 8-pin and 20-pin machined pin socket to make servicing of the following two chips a breeze. A machined pin socket will allow you to use a small cable tie to hold in the chip but can easily remove the chip if needed.

- ❑ Install the LM2574 12v regulator at location U1. The notch should face right.
- ❑ Install the HIP4081A at location U2. The notch should face up.

NOTE: The electrolytic capacitors have their negative terminals marked. Similarly the PCB has a **+** symbol on the positive terminal. Match them up and make sure you get it right, or they could explode! The tantalum caps, on the other hand, have a mark on the positive side. Make sure this positive terminal goes into the PCB hole marked **+**.

- ❑ Install two 1.0uf tantalum capacitors at locations C4 and C6. Make sure the mark on the capacitor matches up with the + mark toward the left.
- ❑ Install the 220uf capacitor at location C5. Make sure the stripe goes toward the left.
- ❑ Install two .1uf capacitors at locations C2 and C3. Polarity does not matter.
- ❑ Install two 680uf capacitors at locations C1 and C8. Make sure the stripe goes toward the left.
- ❑ Install the 10-pin (2X5) connector at location CN5 (J5). Pin 1 will be on the lower right.

Before installing the MOSFETs a quick test is in order. You can check your work by connecting up a battery and making sure that the LED turns ON and that you have 12V on the fan pads. Be sure to disconnect the power to your board before continuing.

NOTE: When installing each MOSFET make sure you solder both sides of the PCB. This will ensure an excellent connection. Take careful consideration to be sure your MOSFETs are correctly orientated. It is usually easier to form the leads before inserting them in the board. If you are installing the optional heat sinks then you may want to bolt the groups of four together first and then install the four all at once. Be sure to use a small amount of heat sink grease between the heat sink and each MOSFET.

- ❑ Install eight IRF1405 MOSFETs at location Q1 through Q8. Make sure the tab is facing down. By soldering them one at a time it will allow you to access the leads on the top for soldering as well.

- ❑ Install eight IRF1405 MOSFETs at locations Q16 through Q9. Make sure the tab is facing up. By soldering them one at a time in reverse order you will have access to the leads on the top of the board for soldering.

- ❑ *OPTIONAL:* Fill in the via's with solder. These are the left over holes that connect top traces to traces on the other side of the board. Over time and from expansion/contraction of the board material (Not to mention flexing of the board) small cracks can develop in the via's. In order to prevent this from happening it is best to fill them in with solder. It might be necessary to heat both sides of the via in order to get a clean connection.

- ❑ *OPTIONAL:* De-flux and clean the board. This step is highly recommended. When properly cleaned the board's appearance is better and it is easier to spot cold solder joints and solder bridges. Depending upon the type of flux used, this can be done economically using common rubbing alcohol and an old toothbrush.

When you are finished soldering all of the components a quick test is in order. For a safety precaution place a 5A fuse inline with the **+** terminal of the battery. Check the output of the 12v regulator on the two pads to the right of the led. The red LED power indicator should light up also.

This concludes the Assembly procedures for the OSMC. Congratulations! Before proceeding look over the board and verify the correct location and orientation of all parts. Also check to make sure there are no solder bridges or cold solder joints. Some of the traces are very close together, and it is easy to accidentally create a solder bridge across a trace or two.

Mounting the OSMC boards

If your application for this electronic speed controller is going to be in an environment of high vibration or impact (such as combat robotics) it is suggested that the board and enclosure be mounted using shock absorbent techniques. There are a number of ways to do this, usually involving some sort of rubber or latex type of material. For the circuit boards there are some items that looked like rubber stand-offs and #6 bolts. Some possible substitutes are stacked rubber washers, or medical hose/tubing. The nuts are tightened enough so the boards stay put, but still are able to move a little when needed. This will greatly help to take up most of the energy of the shock or impact. Locktite is needed to keep the nuts on the bolts held firmly. The mounting plate can then be bolted to the frame via large rubber grommet type things. The grommets and the rubber stand-offs can usually be found at your local hardware store.

Building the OSMC Enclosure (optional)

The OSMC Enclosure is a great way to make the final touches for your speed controller. It provides your board(s) with protection from impact, dust, water etc. Besides, it'll look great!! There are many ways in which an enclosure can be made. The process described here involves a material called Lexan. It is highly recommended because of its sleek look and feel as well as its durability and robustness.

Use the Dremel Tool for cutting lexan. They make a special cut off blade for composites that work really well. You can also use a regular cut off wheel but this is not quite as fast. Run the tool at its fastest RPM so that you actually melt the composite rather than cutting it. This leaves a build up on the edge as you cut but it readily breaks off for a clean cut. If you use a circular saw it can chip and break the lexan. You can use a friction blade on the circular saw but the Dremel Tool works well for all but the really large cuts.

For bonding you can buy adhesives just for composites. Lexan bonding material is called MEK. Plastics and sign shops work with lexan all the time and should have it. Many home improvement stores may carry it as well.

There are several ways of forming lexan. If you want to use clear lexan it is much more work. Lexan attracts water. So if you heat it enough to bend it, the water expands and you get a milky bend. The solution is to heat it in an oven overnight at just above the boiling point of water. Then increase heat to about 250 and form as needed. This will drive the water out and leave a clear bend. Another way to bend lexan is to use a metal brake. This will leave a very clean and nice bend. But do not bend beyond 45 degrees per bend and use a "loose" bend. Otherwise the lexan is weakened significantly. The third method is to use a heat gun. This is the method I prefer. The trouble is that the bends are not very pretty. I can't see the advantage in using clear lexan in that this allows a competitor to see your "guts" in detail. So I use opaque lexan. This covers imperfections well and then you can paint it so the end result looks good. For the actual bending use several pieces of heavy aluminium plate and bars of different sizes. Steel or even wood could also be used but the wood gets scorched. However wood does not conduct the heat away as fast so it helps in forming speed. For straight bends clamp the lexan between the pieces and moving the gun back and forth for several minutes until it bends easily. Using a straight edge such as a piece of 2X4 or other material, bend it to whatever angle you want and then using a wet cloth, cool it off so that it stays where you want it. For odd shapes use a mold. You can heat the material as you bend it over the mold. By using heavy leather gloves you can easily work the lexan with you hands as needed. Using a 1000-watt gun and 3/16 lexan, 18 inches is about the maximum length of a bend without real long heat times. Of course by using wood for clamping and two guns, you could do much longer bends. With IR heaters and such, I think several feet is doable. I also suggest using scrap pieces first. I wasted probably $50 worth of lexan because I didn't practice. Also, carefully measure the bends and "take up" so you can get the right dimensions. I ended up with several nice looking but wrong size pieces because I didn't account for this.

REPLACEMENT PARTS LIST

Most of the parts are listed below the Digi-key part number http://www.digikey.com and second number below showing the same part from Mouser Electronics http://www.mouser.com as another source. In some cases there will only be the generic industry number. The HIP4081A can be ordered from: http://www.alliedelec.com/

CAUTION: Read the FAQ on part replacements before making any changes

Quantity	Part Number	Description	Location
2	P4656-ND	.1uf 100v - Poly	C2, C3
2	P2073-ND	1.0uf 50v tantalum	C4, C6
1	P5139-ND	220uf 16v electrolytic – Radial lead	C5
1	PCF1130CT-ND	.47uf 50v – 1206 SMD	C7
2	P11276-ND	680uf 63v electrolytic – Radial lead	C1, C8
1	P1.0KECT-ND or 263-1K	680 ohm resistor – 1206 SMD	R29
6	P10.0KFCT-ND	10K ohm resistor – 1206 SMD	R11, R21-R25
16	P150ECT-ND	150 ohm resistor – 1206 SMD	R2-R9, R13-R20
2	P33ECT-ND	33 ohm resistor – 1206 SMD	R1, R10
1	P90.9KFCT-ND	90.9K ohm resistor – 1206 SMD	R12
1	P14.0KFCT-ND	14K ohm resistor – 1206 SMD	R28
2	P249KFCT-ND	249K ohm resistor – 1206 SMD	R26, R27
3	UF1002DICT-ND or 625-UF4002	UF1002 1A 100V Ultra fast diode – DO-41	D12, D13, D14
8	1N4744ADICT-ND	15v Zener (one watt)	D2-D5, D8-D11
3	1.5KE51CADICT-ND	TVS Diode	D1, D6, D7
16	LLSD101ACT	Fast Schottky diode– SMD	D16-D31
1	67-1105-ND	RED LED	D15
1	903-3104 (Allied #)	HIP4081A – MOSFET driver – 20 pin DIP	U2
1	LM2574HVN-12-ND	12v regulator – 8 pin DIP	U1
1	DN7431-ND	330uH 740ma - Inductor	L1
1	MHS10K-ND or MHB10K-ND	2x5 header – protected	CN5
1		FAN 80mm (3.15") 12v or 24v depending upon source	J1
1	OSMC3-2	Custom Printed Circuit Board (through hole plated) 4oz See: http://www.robot-power.com **OR** Contact David Moeller DAVID.MOELLER@RHINESTAHL.COM	
4	2219K-ND	8-32 x 1" stand-off	
6	2217K-ND	8-32 x .5" stand-off	
Each of the OSMC boards requires 16 matching FET's. Choose from the list below.			
16	IRF1405-ND	IRF1405 - HEXFET N-Channel, 55V 133A TO-220AB	Q1-Q16
or			
16	IRF1404-ND	IRF1404 - PWR MOS N-Channel, 40V 162A TO-220AB	Q1-Q16
or			
16	903-0048 (Allied #)	HRF3205 - MOSFET, N-Channel, 55V 100A TO-220AB	Q1-Q16
(Optional) Heat sinks for the FET's. Install as four groups of four.			
4		6-32 x ¼" nut	
4		6-32 x 1 ¼" screw	
12		6-32 x 3/16" aluminium spacer	
16	532-577002B00	(Mouser) AAVID TO-220 Heat sink	Q1-Q16

FAQ

❖ What is the OSMC?

The Open Source Motor Controller (OSMC) is an attempt to make an inexpensive and reliable permanent magnet direct current (PMDC) motor controller following open source principles. The project is oriented toward robotic combat but is easily applicable to many other areas. The control system that the group has come up with is composed of two boards, the OSMC (Open Source Motor Controller) and the MOB (Modular OSMC Brain). The OSMC drives the actual DC motor and has no smarts on it at all. The MOB takes signals from a receiver and drives the OSMC board with the appropriate signals. There is also a scaled down version of the MOB available called the uMOB. It can be found in the FILES section of the Yahoo groups web page.

❖ Where can I get more information?

The group mailing list is hosted by Yahoo groups. Go there for all your OSMC needs! http://groups.yahoo.com/group/osmc/

The original project page is at http://www.dmillard.com/osmc

❖ How many Amps Can the OSMC withstand?

The current limit is dependent on many different things. The most important being the MOSFETs used. The IRF1405 is rated at 169A continuous but the package maximum for the TO220 is ~75A. So when you figure on 75A x 4 fets per leg you get a maximum continuous current of 300A. But this is misleading. The fets can handle this current but where the problem comes is in heat dissipation. If the fets get too hot it will toast the fets. If you mount heat sinks on each MOSFET (as shown in a picture above) and also mount a fan, as suggested in the assembly, then you are making steps in the right direction to increasing the maximum continuous current draw. Smoke tests have been done using 24volt batteries and 0.0511Ω load. The amperage applied for 30 seconds (before the batteries were totally drained) was found to be 407A ! The fets were hot.

❖ What is the maximum rated voltage?

You can go up to a maximum battery voltage of 50V. Note: Batteries have a significantly higher voltage “fresh off the charger” so you won’t be able to use 48V worth of batteries with the OSMC. It is recommended that you limit your “rated” battery voltage to 36V or so. The minimum voltage is 12V.

❖ Do I have to drive the OSMC with a MOB or uMOB?

No, you do not have to use the MOB or uMOB, but you will need something to drive the OSMC. The MOB or uMOB are highly recommended because they were designed specifically as a part of the OSMC project. They have proven themselves to be very capable and flexible. All software source code, tools, upgrades and customisations are FREE and downloadable from the group web page.

- ❖ What does the OSMC interface look like, and how does it work?

See the section on Circuit Description found above.

- ❖ What is the cost of making the boards?

The yahoo group site (http://groups.yahoo.com/osmc) has a complete Bill of Materials (BOM) for the OSMC and MOB boards. Go to the web site and click on the FILES section and you will find the BOM for both boards in their respective folders. If you buy everything from Digi-Key, approximate prices are as of October 01, 2001:

- ➢ OSMC PCB : $10 USD
- ➢ OSMC components : $90 USD
- ➢ MOB PCB : $10 USD
- ➢ MOB components : $35 USD

So, for a typical differential drive platform*, you are looking at about $250 USD. Given its capabilities and robustness, you won't find a deal like that anywhere else!

NOTE: A typical differential drive platform means two speed controllers, one for each motor, and 1 MOB or uMOB.

- ❖ How thick is the copper on the OSMC circuit board?

The OSMC3-2 boards are double sided with 4 ounce copper traces.

- ❖ What are the dimensions of the OSMC?

The OSMC board is 3.15" X 4.5" (all units are in inches)

- ❖ What equivalent parts can I swap out, if I can not find or order the parts listed in the BOM ?

This question has come up on many occasions. Here you will find some suggested replacement parts. However, the builder should be warned that the OSMC was built with certain tolerances and part characteristics in mind and that these suggestions may or may not result in favourable results.

- ➢ Mosfet IRF1405

This item is one of the most critical items on the OSMC board. It has direct influence on the ability of your circuit to perform. The most important factor is the chip's continuous current rating. Some quick calculations can be made to determine what your minimum current capability must be:

Mosfet current rating = (motor stall current) ***/*** (# Mosfets per leg of H-Bridge) ***** 1.25

Since the paralleled mosfets may not split the current evenly, your final decision for Mosfet current rating should be at least 25% more, hence the multiplication by 1.25

Suggested part replacement(s):

IRF1404 – cheaper, lower current & voltage rating. Good for low to mid amperage (rated for 162A) applications. Recall that you can actually sink even more current with these Mosfets paralleling 4 per leg, for a total of 16 IRF1404 MOSFETs.

HRF3205 – cheaper, lower current rating. Good for low to mid amperage applications.

- Capacitors C1 & C8

These two capacitors can be replaced with a single electrolytic, polarised, capacitor. The key specs for this replacement cap is it must have a higher voltage rating then the battery source. The capacitor should also be approximately 1380uF charge capacity. Of course a capacitor this big would not fit on the board easily.

- Resistors R2 through R20

Since these resistors control the "on time" of the MOSFETs, they should only be changed if you change MOSFET IRF1405 to some other MOSFETs. What this new resistance *should* be depends on the new MOSFETs characteristics. My best guess is that the resistance should be in the low Ohm range (10 – 20), up to no more then 150 ohm.

- I want to operating at a higher voltage then 50V, what do I need to change?

A lot of changes are needed to achieve this. You might as well have started with an entirely different design.

- I don't want to use Surface mount (SMD) parts, what can I do?

Unfortunately the board was designed to use surface mound devices because they create a physically smaller printed circuit board. You could try creating your own PCB layout using the OSMC schematic, one that allows for through-hole components.

❖ Is the OSMC project offered in a “do it yourself” KIT form, or pre-assembled?

Yes! David Moeller, one of the main designers of the OSMC circuit is now, as of November 16, 2001 selling both bare boards and fully assembled OSMC boards. He is also offering to sell parts kits for the OSMC3-2 Boards including the fan. The cost for the parts kit is $95.00 This will also help with the cost of the fully assembled/tested boards, because I can get the components in qty. I have also quoted some other soldering houses for soldering the OSMC boards. The result is I can lower my OSMC fully assembled price. The new price will not include shipping. That will be extra. So for a fully assembled / tested OSMC the price will be $200.00 Plus shipping. The shipping will have to be insured so I'm not sure I can ship via U.S. Mail. For the first run delivery will be about 3 weeks after I receive the boards. After that I will keep a few in stock.

P.S. I am currently taking orders for MOB boards also.

Please email me David with any questions in this regard.

DAVID.MOELLER@RHINESTAHL.COM

OR

Visit Chris Baron & Dennis Millard’s web site at: http://www.robot-power.com/

❖ Does the HIP4081 shut down if the battery voltage droops?

It shuts down all outputs at 8-9V. However, it is very risky to count on this. If you don't install the 12V regulator chip and parts you can supply 12V to the 4081A. It needs at least 500mA capacity. This would indeed keep the 4081A and MOB alive if the main motor battery drooped. I don't know about ground loops or things like that in such a situation. The board isn't designed with this in mind.

CONCLUSION

I suppose if you've made it this far, then there isn't too much left to say. A hearty "CONGRATULATIONS!" is in order. If you have any questions or comments please post them at:

http://groups.yahoo.com/group/osmc

We would also love to hear about your success stories, or any unfortunate failures. Tell us what you liked about it, what you didn't like about it. How can the circuit be improved? How could this document be improved? Or even if you want to hang out and chat with a group of lively people talking tech.

Take care

slloyd