Opti-Drive LED Voltage Regulator/Pulse Width Modulator

Assembly instructions for deluxe kit and "short" kit.

For use in 12 VDC negative ground vehicles with combined Stop/Turn Bulbs. See back pages for further uses.
Off-Road or Show use only. This module is intended for use on vehicles with retro-fitted LED taillight systems. It outputs a fully regulated 12VDC signal on both Left and Right channels with only 12.8 volts required on the input.

- Input voltage 5 volts up to 20 volts DC. Only 12.8 volts required for a fully regulated 12 VDC output (9-volt output version available upon request).
- Output 12VDC on two channels.
- Maximum load of 2A per channel. Regulator is thermal and over-load protected. Additional capacity can be added externally as a user option with the purchase of additional parts.
- Marker lights brightness function. Duty-cycle is user adjustable from 5%- 98% via potentiometer.

If you are an experienced builder, then you can easily assemble the entire board by following the part outlines on the silkscreen layer and looking at the pictures for guidance.

Opti-Drive V2.4 Release notes:

A minor change on these boards was the addition of two small pads next to the left and right outout connections. The new pads are marked "s-tap" and are only used if you wish to tap into the signal used to drive the Sharp regulators. By tapping into these pads you can drive additional Sharp regulators (not included) that you build on separate circuit boards. This is only required if you have an unusually high number of LEDs in your tail light design. See notes in "Options" diagrams towards the end of this doc for more ideas.

Opti-Drive V2.3 Release notes:

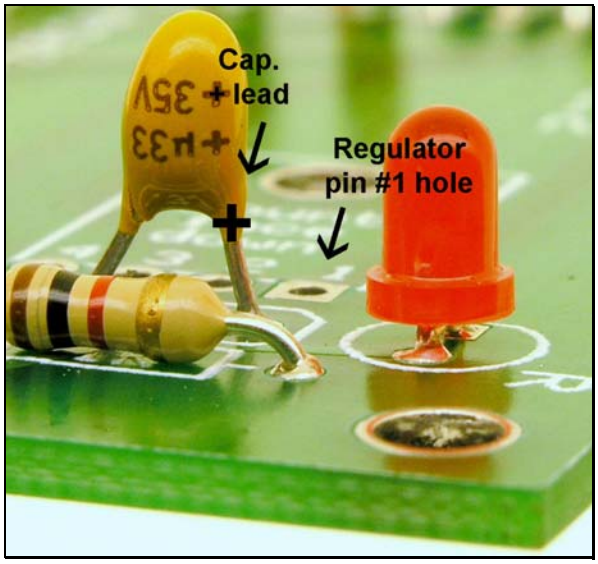
The silkscreen layer omission noted below was corrected on V2.3 so users can skip to the Assembly Notes. A short kit parts list can be found in the final pages of this document

Opti-Drive V2.2 Release notes:

Please note this one item.

There is a minor detail that is not clearly marked on the board. Due to a capacitor change the polarity of the (2) two 0.33uF capacitors (next to the regulators) is not indicated on the silkscreen layer. Be careful to place the "+" lead of the 0.33uF capacitor so that it is closest to the square Pin #1 pad on the regulator. See pictures for more details.

If you are assembling this project using the step-by-step directions, you will be reminded again about the capacitor polarity when it is time to install those parts. The other two capacitors you will find in the kit are non-polarized and may be installed in either direction.



Be sure to place the 0.33uF capacitors with the polarity as shown.

Assembly Notes:

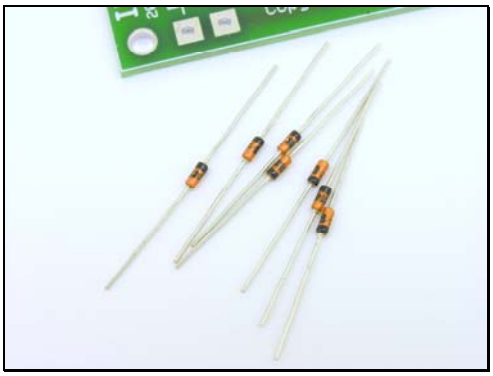
- Always install the smallest parts first, working your way up the largest parts last.
- Solder from the backside of the board only (non-printed side). The holes are through-hole plated so the solder will wick through the holes as you assemble the components.
- Trim excess wire leads as you install the components.
- Refer to the completed board photos on the back page for extra help.

Step-by-Step Instructions

Step 1

Install the (7) seven 1N4148 diodes

Diodes are marked for polarity. You must install them correctly. Install the diodes with the band marked ends properly oriented on the board. Note that where diodes are grouped together on the board they alway point in the same direction. This makes it easier to spot one accidently installed backwards.

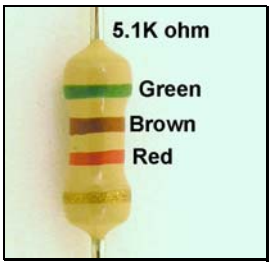


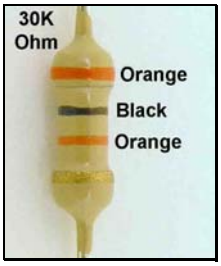
Step 2

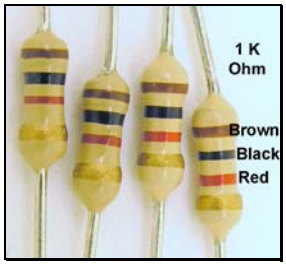
Identify the (7) seven resistors. Some have color bands that look very similar at a glance. Be sure you have them identified correctly before installation. Be most careful with choosing the 1K and 10K as the orange and red bands can look the same. Once you have them identified, install each into the circuitboard. Resistors have no polarity. You may install them in either direction.

They are color marked as follows:

Qty. 1 5.1K Ohm = Green, brown, red, gold.
Qty. 1 10K Ohm = Brown, black, orange, gold
Qty. 1 30K Ohm = Orange, black, orange, gold
Qty. 4 1K Ohm = Brown, black, red, gold

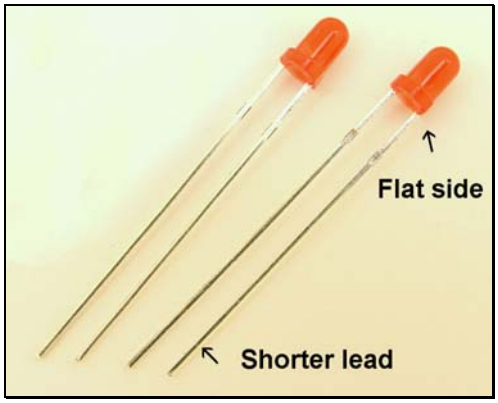






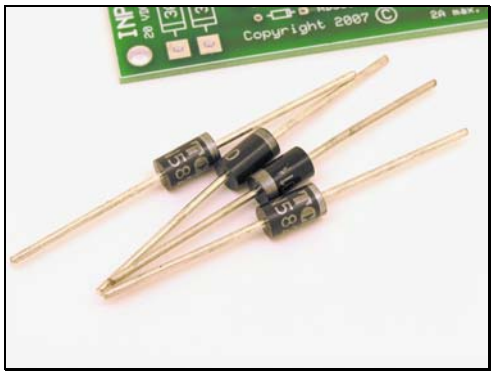
Step 3

Install the two red LEDs. These are polarized devices. Be sure to install them with the flat side (- cathode) of the red plastic base matching the device outline on the circuitboard. The - cathode lead is also easy to identify since it is always the shortest of the two leads.



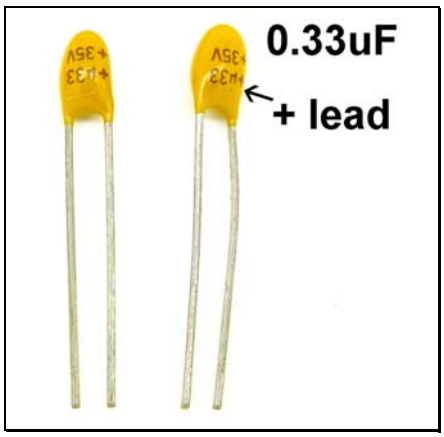
Step 4

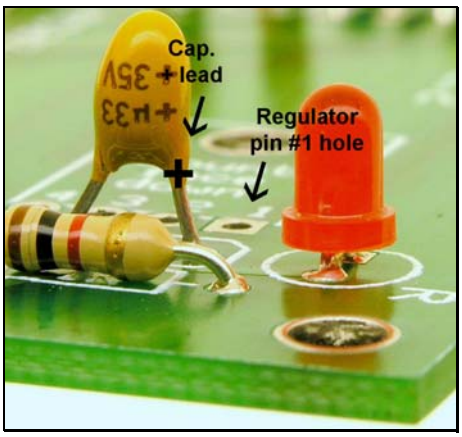
Carefully bend the leads and install the (4) four large 3A barrel diodes. Pay attention to the polarity. Match the band on the diode to the part outline marked on the circuitboard.



Step 5

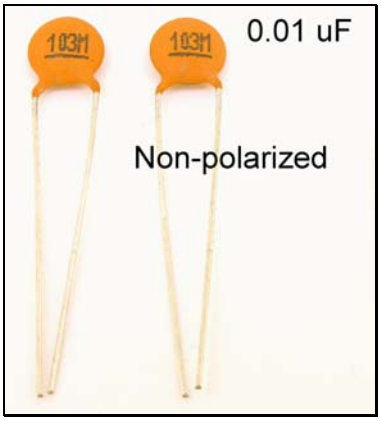
Find the (2) two 0.33uF capacitors. These are marked with "+u33" printed on them. Install them into the circuitboard with the "+" lead closest to the pin #1 hole (square pad) of the regulators. The polarity is not indicated in the parts outline printed on the board so please see the photo below for details.





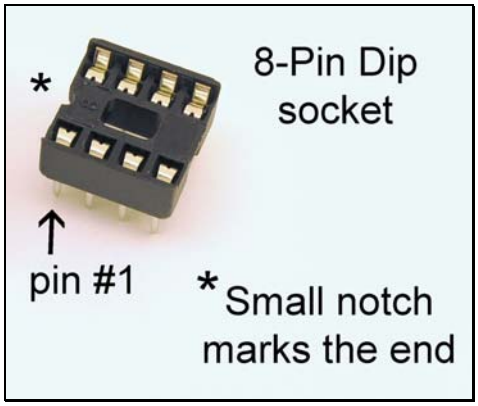
Step 6

Find the (2) two 0.01uF capacitors. They are marked with “103M” printed on the part. Install them into the circuitboard. These have no polarity and you may install them either way.



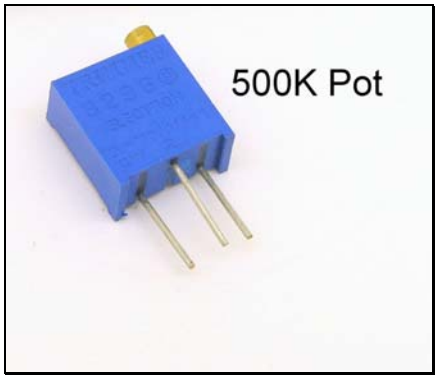
Step 7

Install the 8-pin socket. Note the polarity of the socket when you install it into the circuitboard. The small notch in the plastic identifies the pin #1 end.



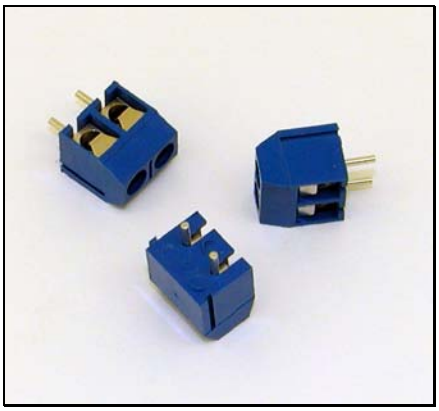
Step 8

Install the potentiometer. It can only be installed in one orientation. Note that when it comes time to adjust the device that it is a 15-turn precision potentiometer and it may take several revolutions of the screw before you see any change in brightness. The pot cannot be damaged by turning it too far right or left so don’t worry about turning it too far either way. You will hear faint clicks as it reaches the end point in both directions.



Step 9

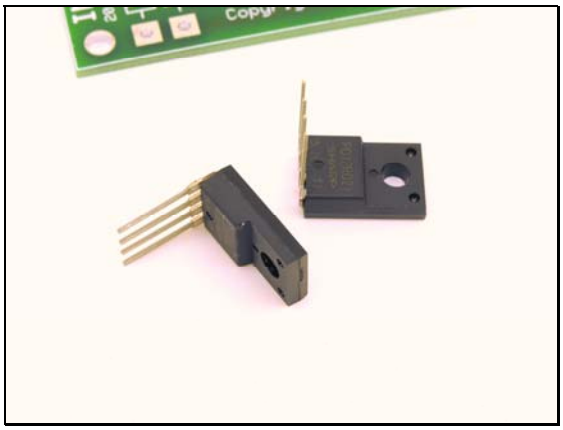
Install the (3) three 2-position screw-terminals. Two of them will slide together to create the 4-position input terminal. Be sure to mount them with the wire clamping side to the edge of the board.

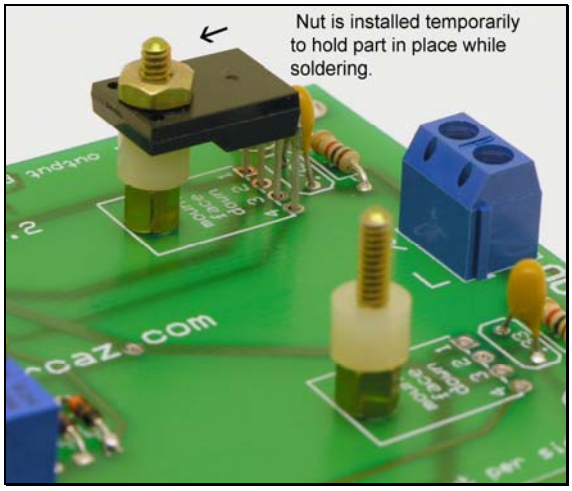


Step 10

Install the two Philips screws from the bottom of the board into the heatsink standoffs. Place the nylon spacers over the standoffs and then position the regulators onto the circuitboard. The regulators install with the face (printed side) down. Temporarily install the two 4-40 nuts to hold the regulators in place. When the alignment looks good, solder the regulators in place. Do NOT over-tighten the hardware.

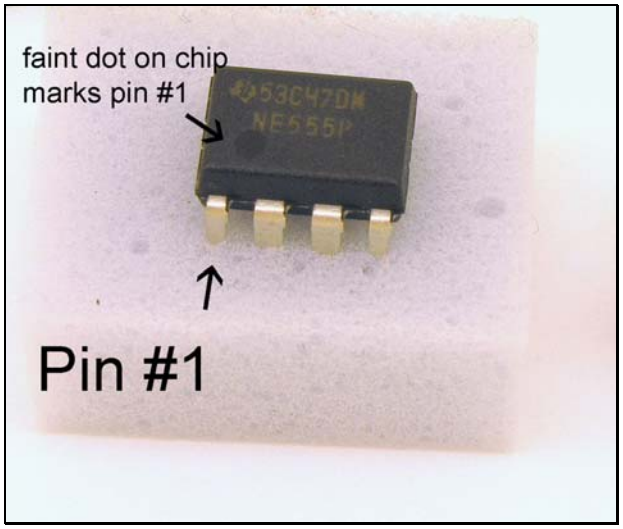
Do NOT install the heatsinks or heatsink grease until after you have tested the board.





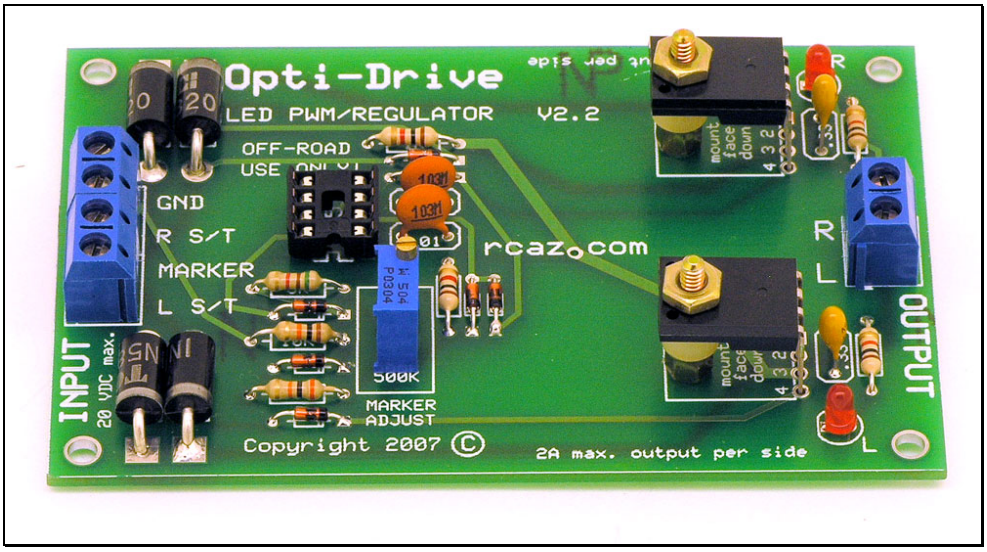
Step 11

Install the 555 timer into the 8-pin DIP socket. Be careful to align the chip into the socket correctly. If static is a problem in your assembly area, touch something grounded on the work bench before installing the chip. This will help prevent damage to the static sensitive 555 timer.

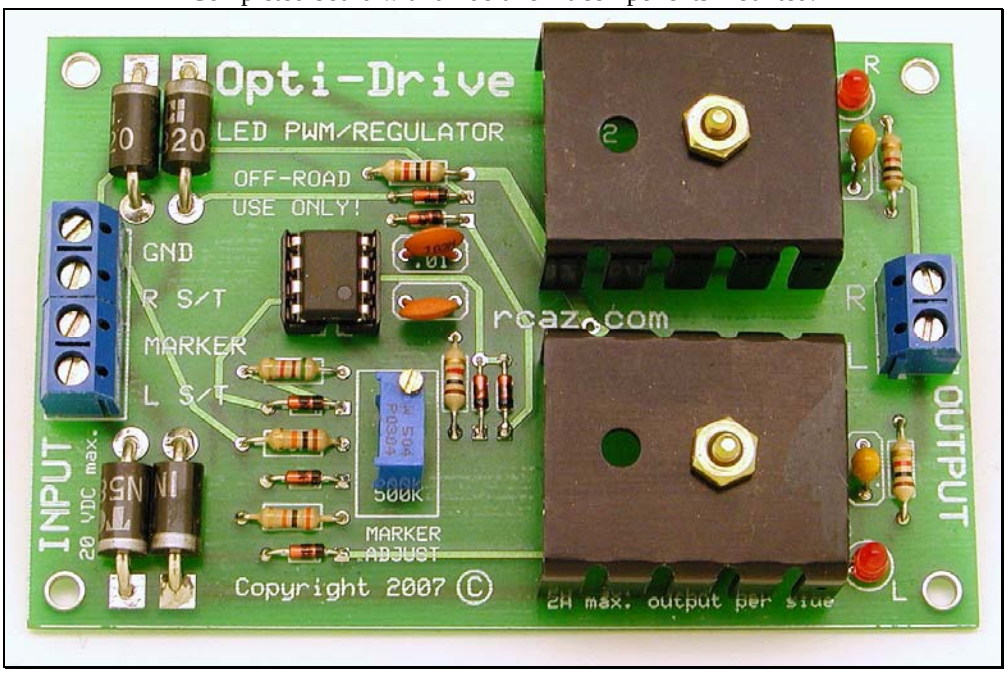


You should now have only two heat sinks and some heat sink grease remaining in your parts bag. Install these after the board testing is completed but before you place a large load on the output. Only a tiny amount of heat sink grease is required. Cut the plastic bag corner and squeeze out a little grease. Smear a thin layer on the regulator before attaching the heat sink. The heat sink installs face down over the regulators. The heat sink is electrically isolated from the circuit board so you do not have to worry about it touching metal parts later when you install it in your application. Do not over tighten the 4-40 nuts or you will twist off the stand-off threads.

Double-check the circuit board under some bright lights and/or a magnifying glass for any accidental connections. Be sure all connections are soldered.
Here is a view of a nearly completed board so you can double-check your work. The 555 timer and heat sinks are not shown in this picture for clarity reasons.



Completed board with all deluxe kit components mounted.

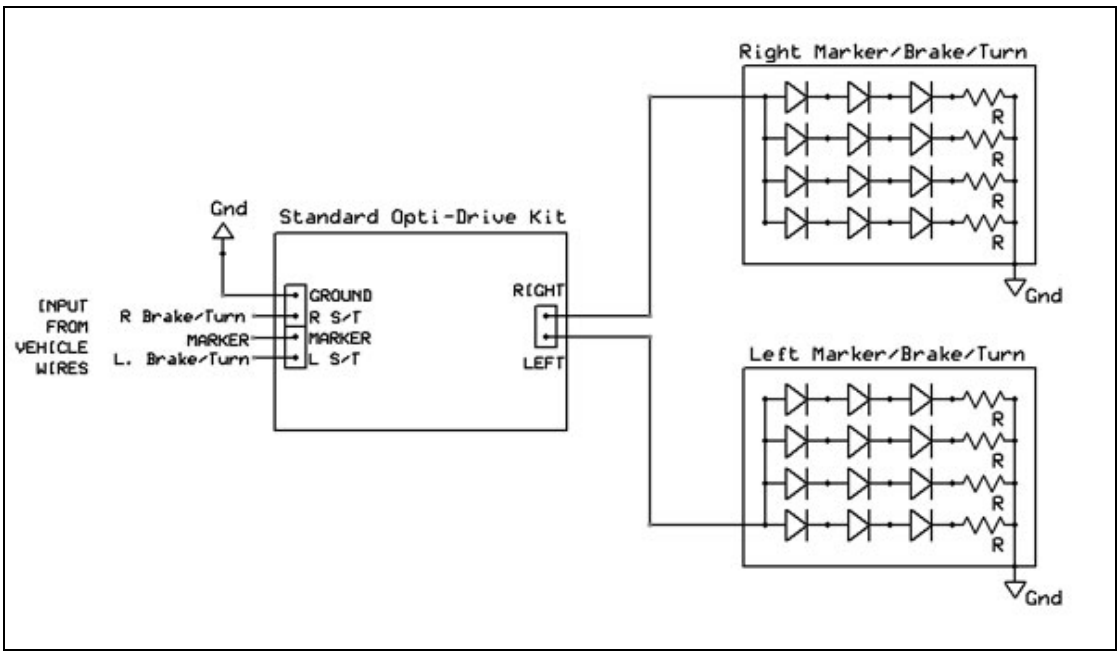


You can safely simulate the operation of this assembly with a 9-volt battery and matching battery clip from Radio Shack, etc.

- Hook up the negative wire from the 9-volt battery to the top screw terminal "GND" on the board.
- Hook up the positive battery wire to the "Marker". Both L and R LEDs should now be lit.
- Test the duty-cycle adjustment (brightness) by turning the potentiometer screw CW and CCW.
- Adjust the potentiometer to the lowest setting
- Touch a short jumper wire or paper clip from the "Marker" terminal to the "R S/T" terminal. The right LED should go to full brightness simulating a right turn signal input from the car. Now test the "L S/T" terminal for the same results.
- Finally, unhook the positive battery wire from the "Marker" terminal and touch it to the " R S/T" and "L S/T" terminals to simulate a brake input from the car.

Wiring Diagrams

Input wires are tapped into your old light bulb wires. The LED arrays can simply be grounded to the vehicle chassis. There is no need to return a ground wire to the Opti-Drive from the LEDs.

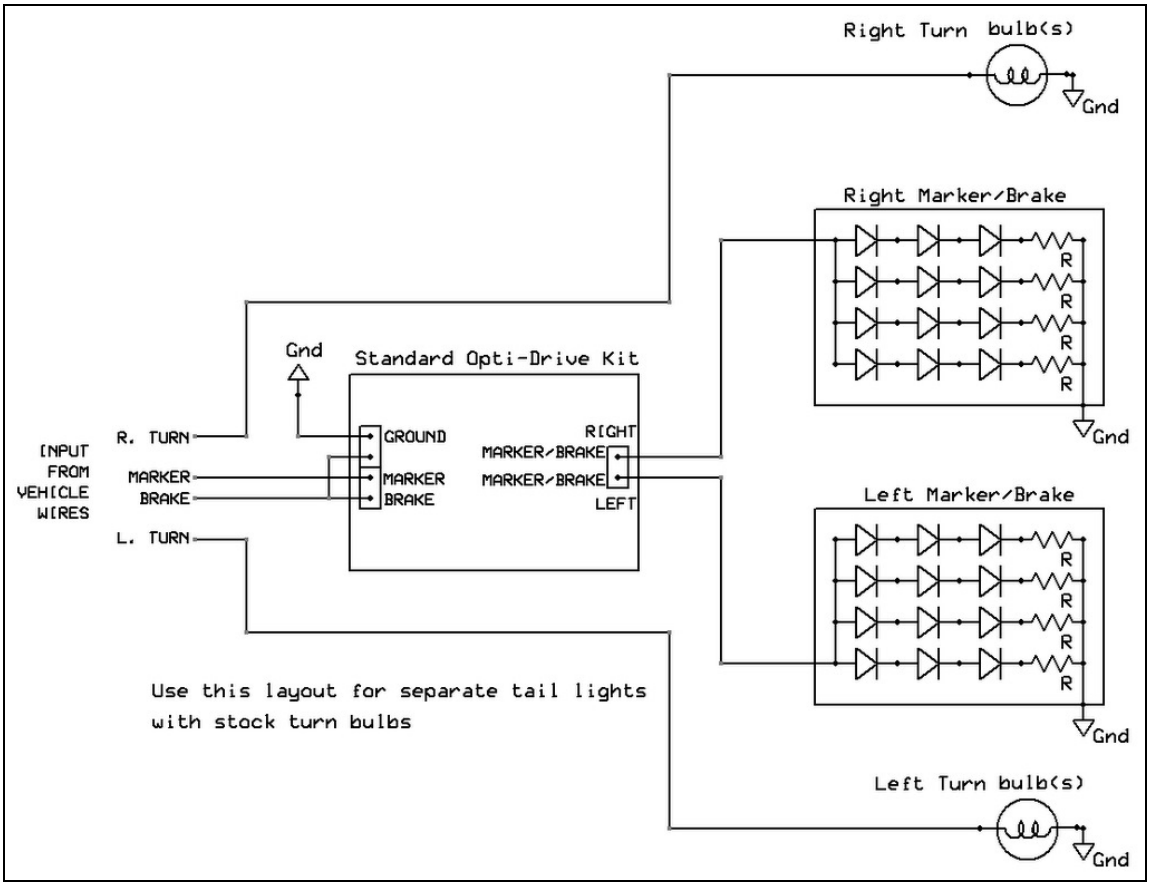


OPTIONS FOR VEHICLES WITH DEDICATED REAR TURN DSIGNALS

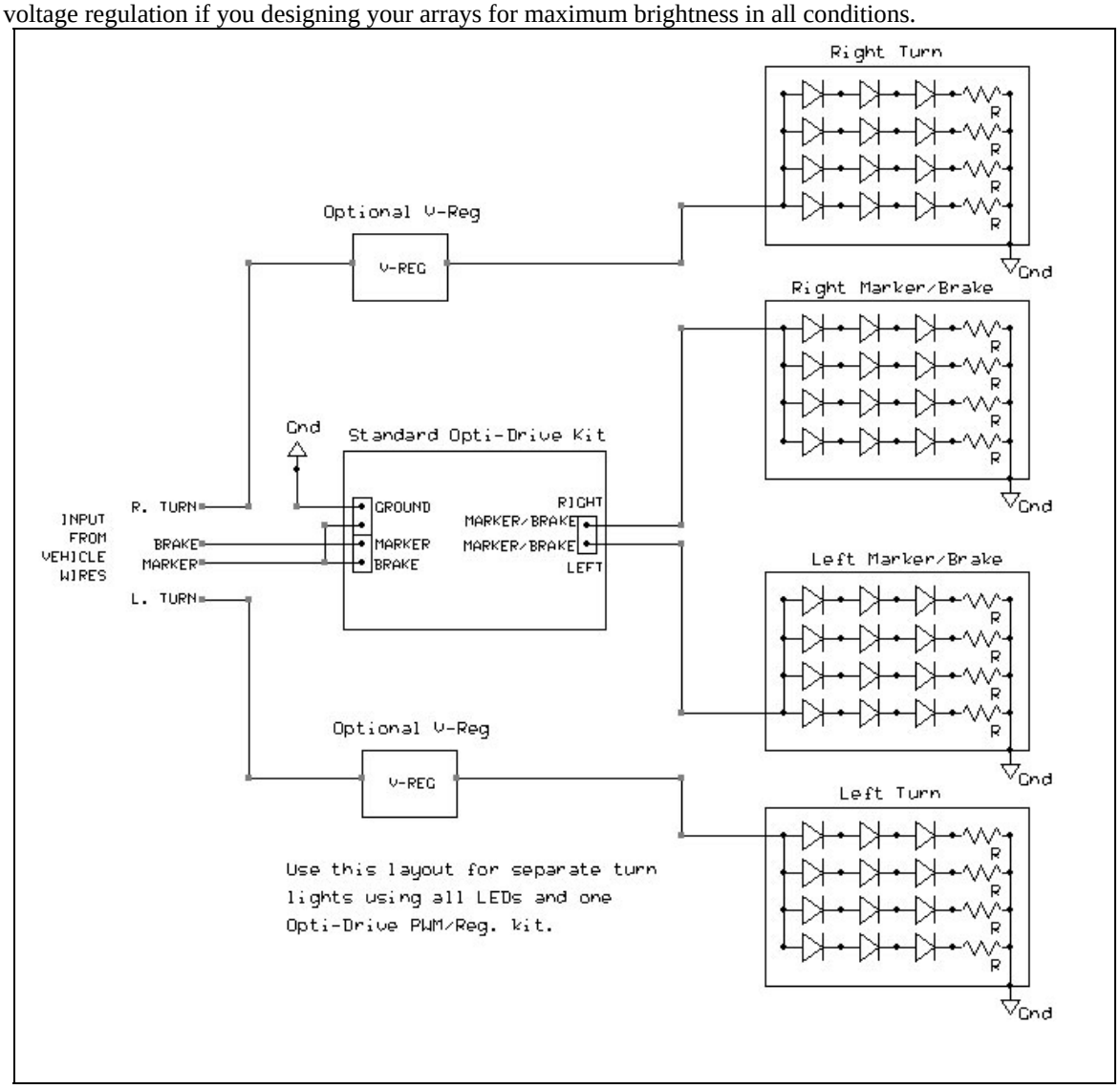
This PWM/Regulator is intended for use in vehicles with combined Stop/Turn Light bulbs.
It can be used in vehicles with separate bulbs for the turn signals. Contact me for design tips or see info below.

Here are three solutions for using this module on vehicles with separate dedicated turn signals.

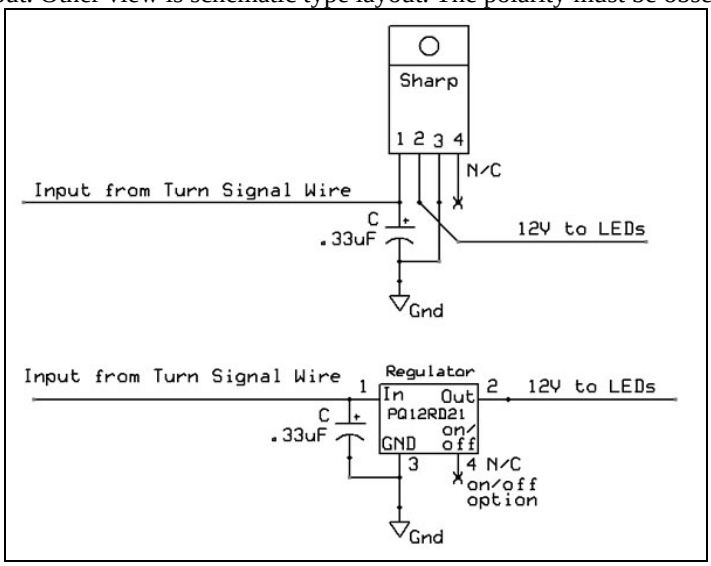
Option 1 Use the OEM turn signals as-is and convert only the Marker and Brake lamps to LEDs. This avoids the hassle of hyper-blinking problems or bulb-out indicators when converting turn signals to LEDs.



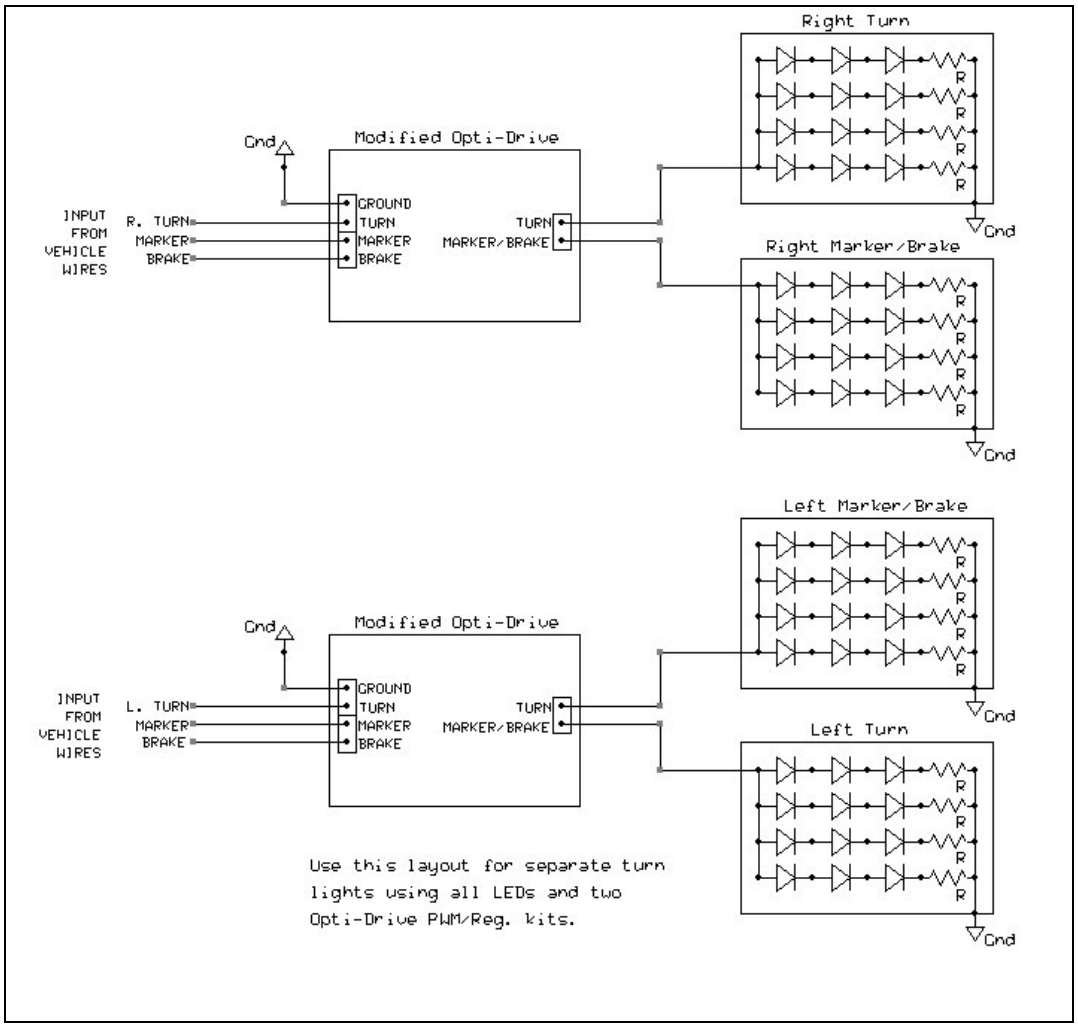
Option 2 Turn signal LEDs bypass the PWM. Since the turn signal LEDs need only operate at full brightness there is no need to go through a PWM. You may use a regulator on the turn signal arrays for best performance but that can be deleted if you desire. I always recommend voltage regulation if you designing your arrays for maximum brightness in all conditions.



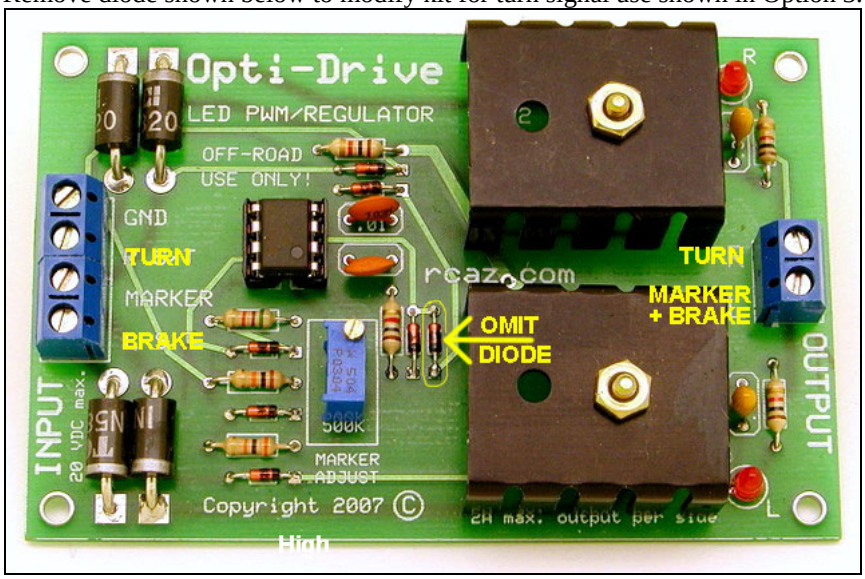
Stand-alone regulator diagram for optional voltage regulator shown in above Option 2.
One view is a pictorial layout. Other view is schematic type layout. The polarity must be observed on the input capacitor!



Option 3 This option is the cleanest and work well for those who want the best option for powering all the arrays. By removing a single diode you can convert the Opti-Drive to function at only full brightness for turn signals on one channel while still functioning at dual brightness levels on the second channel. See below for diode to remove. This mod is easily changed back to normal if you change vehicle later and want to keep the Opti-Drive for another application.



Remove diode shown below to modify kit for turn signal use shown in Option 3.



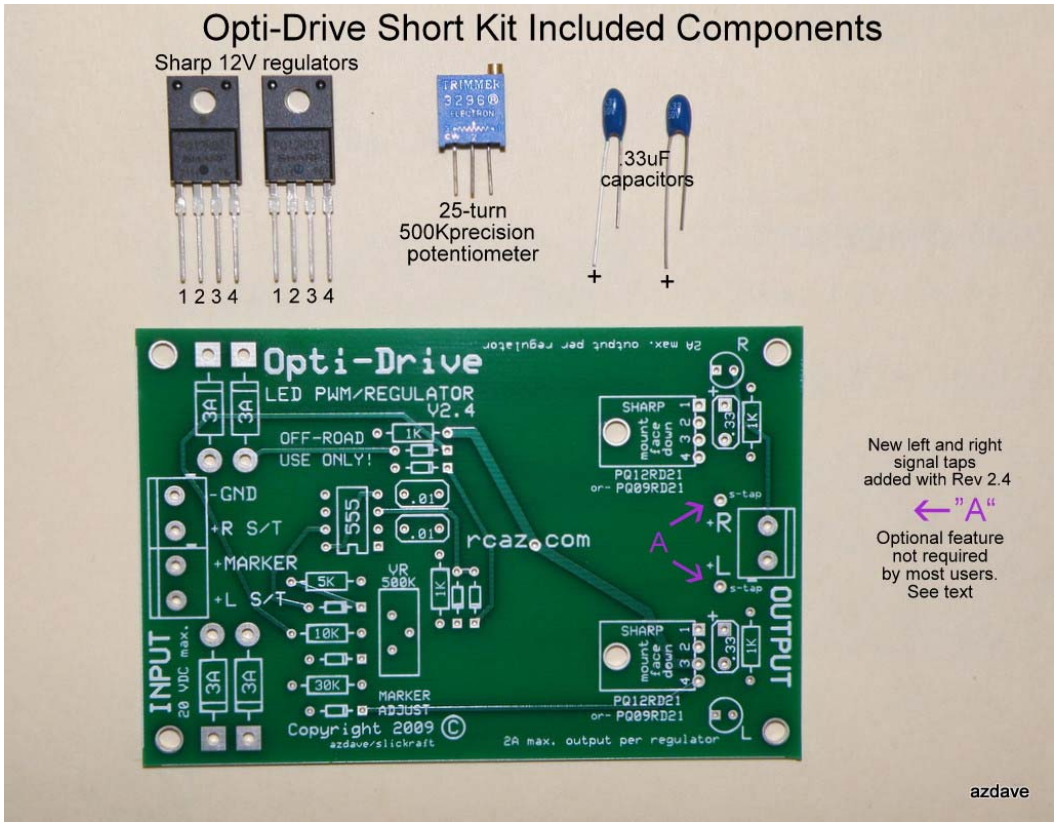
Short Kit Builder's Notes

I made the short kits available for those who wanted to only purchase the board and the few hard-to-find parts and then complete the remainder of the project with parts they already had or can find at local electronics outlets. I compiled a list of suggested parts from Radio Shack but many other substitutions are possible. The short kit is built the same as the deluxe kit except that you may omit some parts like the screw terminals or maybe you will find a different heat sink arrangement.

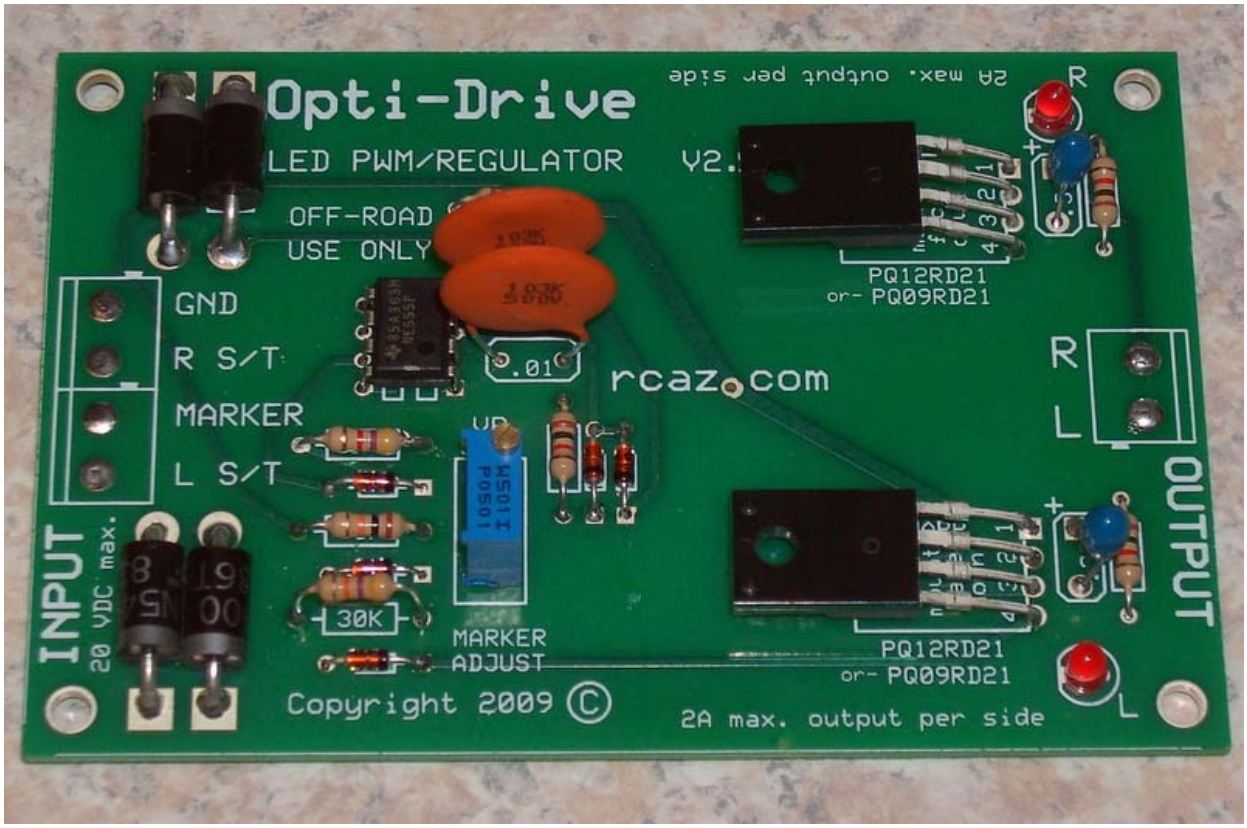
Suggested Components List for Short Kit

The red text marks item included in the short kit.

QTY	COMPONENT	SOURCE	COMMENTS
1	circuit board	azdave	Opti-Drive PWM/REG V2.3
7	switching diode 1N4148	Radio Shack 276-1122	Many small switching diodes will work
1	5K 1/4W resistor	Radio Shack 271-1330 (4.7K okay)	1/8W okay (1/2W may not fit the holes)
1	10 K 1/4W resistor	Radio Shack 271-1335	1/8W okay (1/2W may not fit the holes)
1	30 K 1/4W resistor	Radio Shack 271-1342 (47K but okay)	1/8W okay (1/2W may not fit the holes)
4	1 K 1/4W resistor	Radio Shack 271-1321	1/8W okay (1/2W may not fit the holes)
2	pilot light LEDs	Radio Shack 276-0026	Any small LED should work
4	3A Rectifier diodes	Radio Shack 276-1141	Any 3A diode with 50V PIV or more is okay
2	.33uF Tantalum capacitor	azdave or Digikey, Mouser, etc.	polarized so install + side correctly
2	0.01uF ceramic capacitor	Radio Shack 272-0131	A little big but it fits and works fine
1	8-pin DIP socket	Radio Shack 276-1995	Not required for low budget build
1	555 timer	Radio Shack 276-1723	Any 555 timer should work
1	500K potentiometer	azdave or Digikey, Mouser	15-turn precision pot (must be small to fit)
2	Sharp V-reg. PQ12RD21	azdave, Digikey, Mouser	9-volt version available also
2	heat sinks, grease & hardware	Radio Shack or as required	Many common TO-220 heat sinks are okay
3	Screw terminals	Radio Shack 276-1388 (pack of 4)	Not needed for low budget build



This is an Opti-Drive V2.3 built with the minimum components required using the short kit parts plus components found at Radio Shack. You should add some TO-220 heat sinks to the voltage regulators but they are not shown here since they would hide some components in the photos.



Adding extra capacity to the Opti-Drive