EMW SmartCharge-12000 12kW EV Charger

Non-PFC V12 Assembly Instructions

June 28, 2013

These instructions are similar to the PFC-version assembly videos at http://www.youtube.com/vmiftakhov – look for

EMW SmartCharge-12000
Assembly Manual

Message from the EMW Founder:

Dear Fellow EV Builder!

Congratulations on your decision to take on this ambitious project!

Throughout this project, you will learn new skills that will help you immensely in your other projects. You will understand the fundamentals of power conversion – something that's in high demand in today's society fueled by electrical energy. You will feel the enjoyment of getting things done with your own hands. You will be rightly proud to show off your achievement to other fellow EV Converters and will be able to say: "I went the extra mile".

Thank you for supporting the grassroots EV revolution in transportation and making our planet better!

And thank you for supporting our Open Source philosophy by purchasing our products.

Valery Miftakhov
Founder, Electric Motor Werks, Inc.

CAUTION!

This is a High-Voltage, High-Power design.

It is NOT your average weekend electronics project so do NOT treat it like one.

If not approached with caution and extreme attention to detail, this project can kill you, burn your house, and damage your car beyond repair.

By starting this project, you expressly agree that EMW and any of its directors, employees, and partners are not liable for any damage that may result from this project and associated activities.

That said, over 100 builders have successfully completed this project without hurting themselves (at least to our knowledge ;-), so you can, too. Common sense is your friend, as always.

General Specs / Info

This is just a reference for specs of the unit. It helps to understand the parameters involved as you build the unit so we brought this info here

Product description

EMW SmartCharge-12000 is a fully programmable, open-source (for non-commercial use only) 12 intelligent Non-PFC charger. Designed & manufactured by EMW (Electric Motor Werks, a High-Performance Electric Conversion group).

The charger is sold as a kit or a complete unit. Or you can try to source the parts yourself and build it (not recommended - you would probably spend more than if you bought a kit as you will be paying full retail price for all components; also, you have to buy from us to get technical support for your build).

Common questions:

Q: Why buy EMW charger and not someone else's?

A: The inferior products (non-programmable, non-tweakable, closed-source) of similar power output from competitors run for ~$3,500. Our kits run at ~$1,000 and complete units around $2,000.

Q: Why buy parts from EMW and not direct from DigiKey, etc.

A: Few reasons:

- You will save money by buying from us. We just did a single-charger-quantity quote from DigiKey and all other sources (ExpressPCB, some eBay sellers, 4D systems, ExpressPCB, etc.) and the totals for non-PFC and PFC kits came down to $1,042 and $1,191, respectively. Our kits are priced around the same level.
- You will save a LOT of time not having to procure components yourself. Digikey will get you only that far. You will have to locate and order components from at least 6 different places, some of them international. We estimate that you will spend at least 50-100 hours sourcing all the components yourself. Even if you value your time at McDonald's rates, this is $500-$1,000 wasted...

Finally, you get support from EMW for your build only if you buy the kit from us. Unless you are a power electronics / embedded systems design & programming pro, this might come in handy…

Design notes:

The original charger design was created early last year (2011) and was the basis for our kits up until May 2012. Information on that design is still available from EMW on request.

The version described on this page has been completely redesigned from the ground up, with the following main improvements:

- 95% of the components are now placed on PCBs, including all power components. This eliminates all the layout-related build errors - a significant consideration for the high-power electronics. It also cuts down the build time at least by a factor of two
- All PCBs are made with silkscreens to simplify assembly and allow for the professional look-and-feel
- Current Sensor circuit is now 4x more sensitive relative to the previous version
- Voltage Sensor circuits are now using isolated OPAMPS instead of hall sensors and sensing resistors. As a result, resolution is 4x higher, along with better temperature stability
- Layout is designed to be mountable to both air-cooled heatsink and to the liquid cold plate (you can specify the option when ordering)
- Highly modular design of PCBs and connectors allows rapid swaps and removal of boards when repair / service is needed
- Circuitry and firmware for precharge, automatic 110/220V switching, under-voltage protection
- Firmware optimized for stability

Specs

Input voltage:

- 240VAC or 110VAC for the non-PFC version
- 90-260VAC for the PFC version (output power automatically limited to ~1.5kW if connected to 110VAC)
- Both non-PFC and PFC units can be used with a DC source.This feature can be used to charge one EV from another, to use the charger with a separate rectifier (e.g., for 3-phase operation), etc.
    - Non-PFC: input voltage has to be above the CV voltage of the battery being charged

Output voltage

- programmable ~20-250VDC (non-PFC version)
- programmable ~20-350VDC (PFC version; upgradeable up to 425VDC with simple modifications)

Output current:

- Up to 70A continuous (100A option available)

Output power:

- 12kW rated
- Tested to 15kW in our labs
- Up to 35kW possible in DC-DC mode!

Efficiency at full power:

- 95%+ for non-PFC version (offset by higher losses in AC wires, however)
- 92%+ for PFC version

Switching frequency:

- 22kHz (PFC stage)
- 15-20kHz (buck output stage)

Additional features:

- J1772 support with automatic power regulation based on J1772 pilot signal
- BMS stop-charge TTL input
- End-of-Charge TTL output
- Hardware current limiting
- All configuration settings are stored in on-board flash memory and survive power-downs. Most of the parameters are fully adjustable through the color LCD interface:
    - Power control: at start-up
        - Set input and output current levels
    - Timer control: at start-up
        - Set timer shut-off duration
    - Flexible battery settings:
        - Set your battery type (Li, NiXX are currently available)

- Number of batteries
- Battery capacity

Part 0. Before you start

1. **Required tools:**
   i. Soldering setup
      a. A low-power soldering station with a relatively fine element – something like this (what we use at EMW, $20 from Amazon)
      b. A high-power station / gun (100W minimum, 250W or more preferred) – something like this (what we use at EMW, $70 from amazon)
      c. Electrical solder (make sure you never use a plumbing solder as it may have conductive flux!) – ideally in 2 thicknesses: 1.2-1.5mm for high-power connections and 0.3-0.5mm for small parts
      d. If you haven't done much soldering before, check http://www.dummies.com/how-to/content/what-is-soldering-and-how-do-you-use-solder-tools.html
   ii. Screwdrivers
   iii. Wire strippers
   iv. Small snips for wire / lead cutting
   v. Small pliers
   vi. Drill & drillbits
   vii. Threading taps: 10-32 or 10-24, 6-32
      a. See http://www.wikihow.com/Use-a-Tap for some tips
   viii. Multi-meter with Capacitance / Resistance measurements. This thing is awesome and we use it at EMW ($130 at Amazon – pricey but worth it!)
      a. Also download / print the resistor color coding reference: http://en.wikipedia.org/wiki/Electronic_color_code
   ix. Clear Protective Goggles
   x. Metal fab tools (saw, 3.5" drill saw, snips, etc) for enclosure fabrication – NOT needed if you order a fully machined enclosure from us – see our online store at http://www.emotorwerks.com/products/online-store
   xi. Strongly recommended tools
      a. Clamp meter with 100A+ DC current measurement capacity (something like this – $40 on Amazon - make sure you do NOT just buy an AC current meter – in a lot of listings, it's hard to see that the meter does not have DC capabilities and you find out only when you try to measure something…)
      b. Infrared thermometer such as this ($40 on Amazon)
      c. Scope with at least 1MHz bandwidth (10MHz or higher preferred). Our favorite is this 100MHz OWON scope ($440 at Amazon). Probably an overkill

if you just want to assemble one charger – then you can get this 25MHz one for $270. If you plan to use this awesome tool often, get a Li-ion battery for it (best deals are on eBay)

2. **Helpful aids**
    i. Flat piece of thin plastic / carton to hold the parts while you turn over the board to solder so that components don't fall out
    ii. Small soldering vise to hold pcbs while you solder
    iii. Magnifying glass to read small parts' markings

3. **Required additional commodity parts** (not supplied with the kit, available at any hardware / electronics store or (soon) as an add-on option in EMW online store)
    i. Bolts, nuts, washers: #10 and #8, various lengths
    ii. Signal wiring
        a. Ideally, a set of Pololu pre-crimped female-female wires and housings
            i. Housings (http://www.pololu.com/catalog/category/70): 5x 3-pin, 4x 6-pin (with 2 of them used as 5-pin)
            ii. Wires (http://www.pololu.com/catalog/category/71): 3x 12", 10x 6", 3x 24"
        b. Alternatively, 5-6 colors of AWG22 isolated wire (3' of each color). Could
    iii. Power wiring – ideally 3-4 colors of AWG6-8 isolated multi-threaded wire (2' of each color) plus 1' of bare copper AWG10 wire
    iv. Silicone sealant. Something like http://www.ebay.com/itm/White-Electronic-Grade-Silicone-10-2-oz-Cartridge-Good-Dielectric-Properties-/250924206081?pt=LH_DefaultDomain_0&hash=item3a6c3f8801. Make sure that the silicone sealant you use (page 31) is electrical grade. Ordinary silicone releases acetic acid (vinegar smell) and this will slowly but inevitably erode the exposed copper and solder and cause problems, and the trapped acid is conductive and can cause leakage.

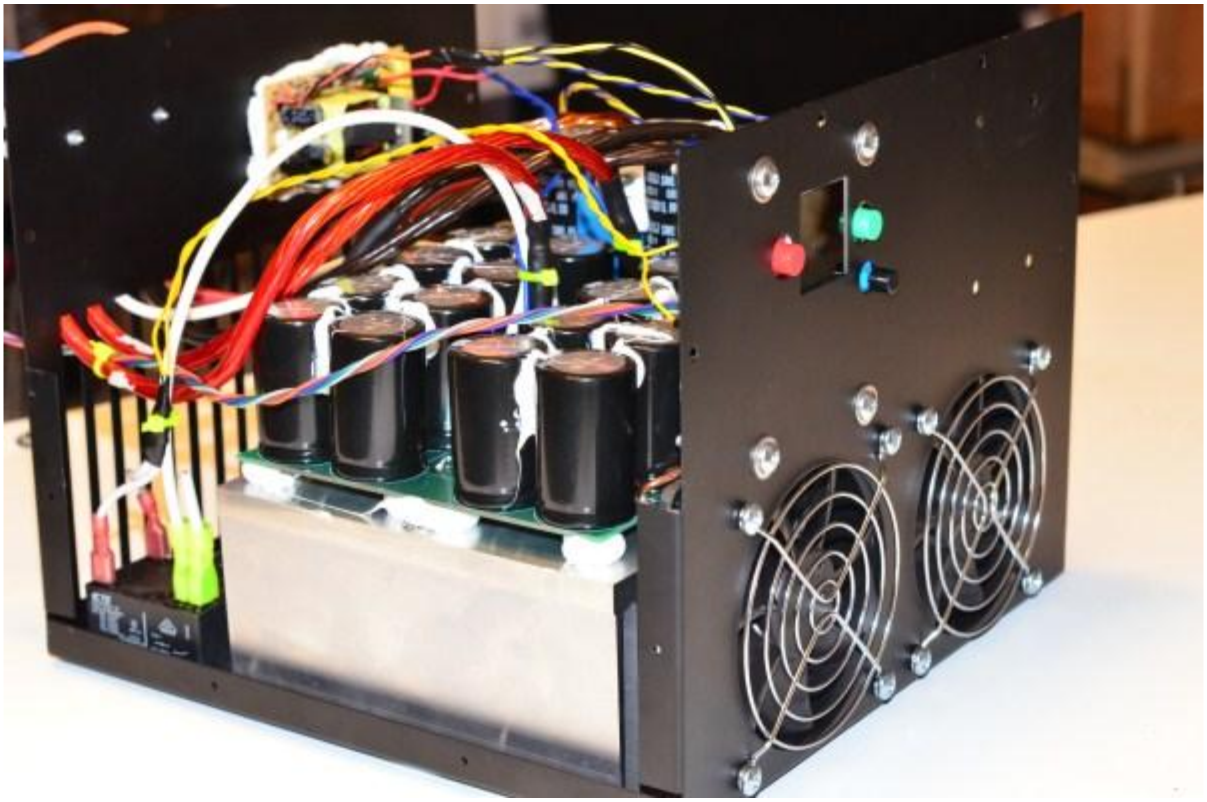
4. **Assembly Tips**
    i. Sequence of assembly is often very important: some parts may not be able to fit after others have been soldered
    ii. Read instructions for the ENTIRE step before proceeding with the first instruction under that step. Ideally, you should scan this entire doc before starting assembly
    iii. If you can't find some part ID on the board / PCB file, do Ctrl-F from within ExpressPCB to find the part
    iv. Place many parts at a time, bending pins on the other side of the board so that the parts stay in place when you turn over the board to solder
    v. It is a good idea to use the standard resistor color code for wire colors on connectors. So brown=1, red=2, ..., white=9, black-10, then start again with brown=11. Or you can use white/brown stripe for 11, etc.

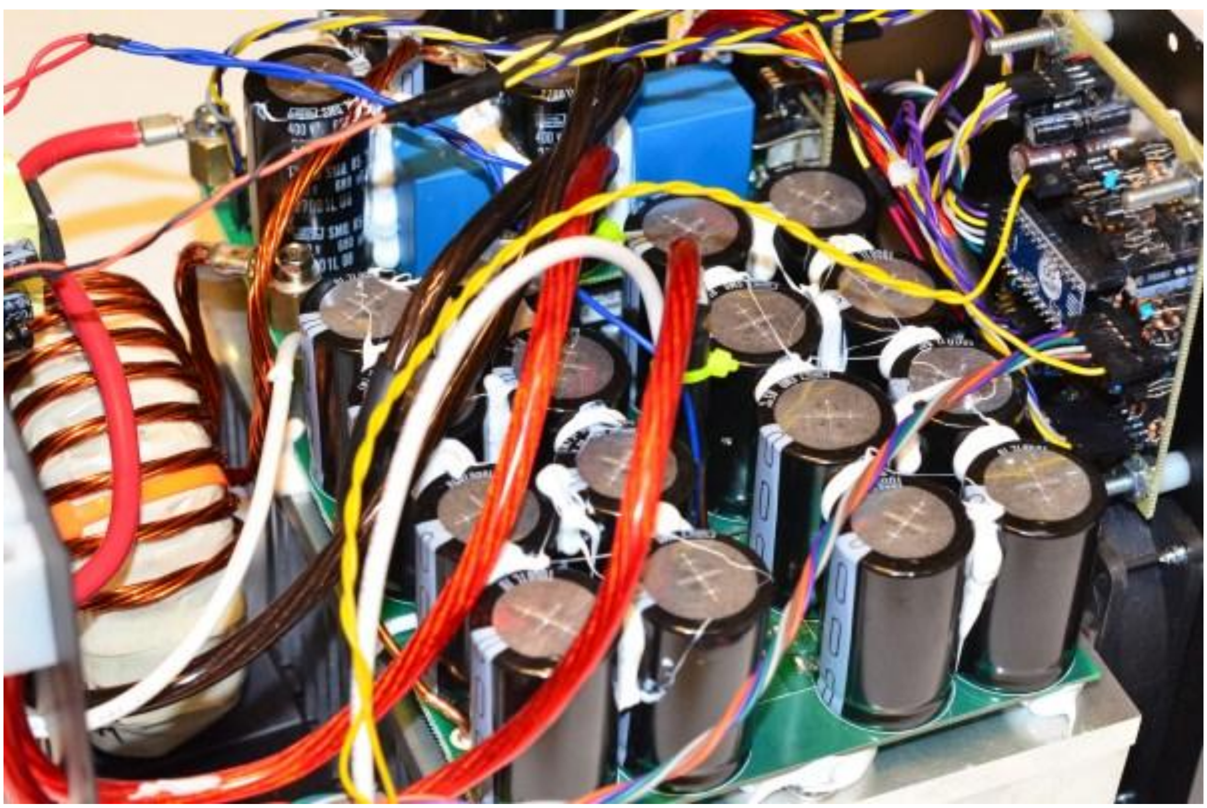
5. Education

i. Wouldn't it be nice to actually understand what you are building?
ii. While mastering power electronics is a multi-year full-time project, you can pick up good amount of fundamentals in a week by reading a good book
iii. I suggest **Switching Power Supplies A - Z, Second Edition** by Sanjaya Maniktala (Apr 18, 2012) - $60 on Amazon

Overall build reference:

Use the image below to get a general idea of relative placement of components. Specific details are described in the corresponding sections of this document.





Part 1. Kit Contents

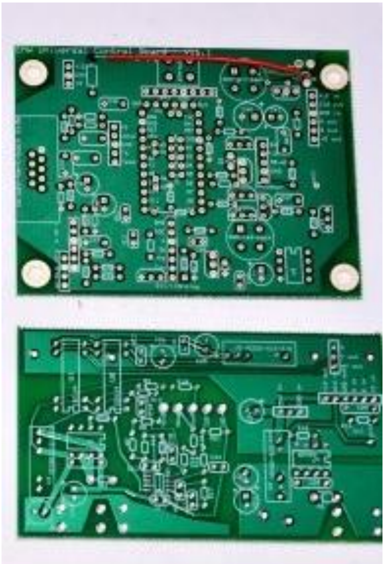
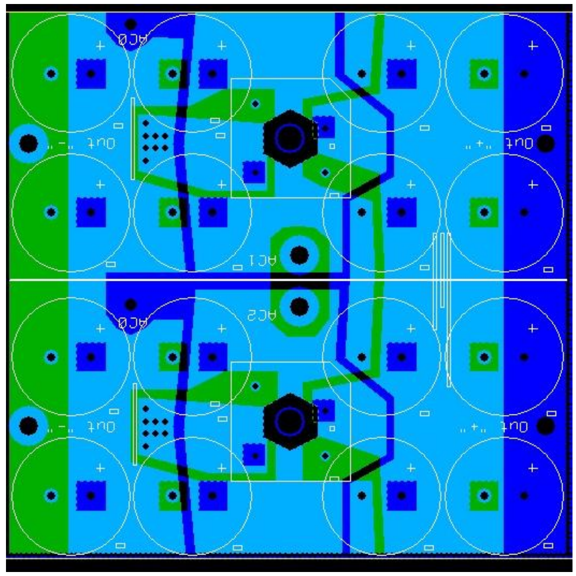
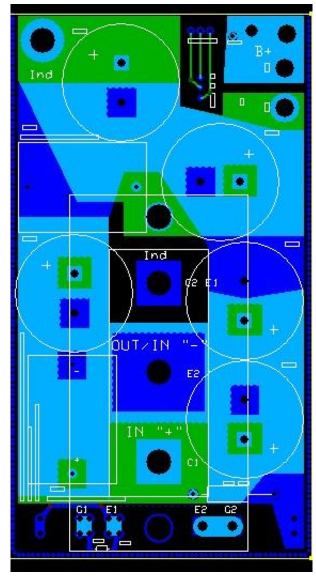
Please refer to the Bill Of Materials file on our site for more details on component lists, part numbers, etc: http://www.emotorwerks.com/VMcharger_V12P/. Tip: enter part number into the DigiKey (http://www.DigiKey.com) search box to get a detailed part info page with full datasheets, photos, etc.

Most of the smaller parts will be placed in a plastic organizer, with types of parts separated (all resistors in one compartment, capacitors in another, etc.). Majority of the parts will have clear manufacturer labeling on them, except resistors. You will have to use a color-code aid to decipher the resistor values (see required tools section above for links to all such aids).

Note: PFC kit contents shown – non-PFC will have only one IGBT and instead of the small bridge boards on the right side you will have a large voltage doubler board. Check PCB images below for reference on how the boards should look like in your version.

1. Heatsink:
    a. Air or Liquid cooled depending on what you ordered
    b. You can order the fully machined version with all the mounting holes drilled and tapped

2. PCBs

a. Control Board (Far Left, Top)
b. Driver Board (Far Left, Bottom)
c. Main Power Board Non-PFC version(Far Right)
d. Voltage Doubler board (Middle)
e. FTDI board (used to program your Arduino, supplied with a USB cable)

3. One EMW High-Power Custom Inductors
    a. If you have selected a high-current option at the order time, you may be supplied a 100A rated output inductor

4. Integrated Circuit (IC) Parts:
    a. Arduino Pro Mini 5V / 16MHz board
    b. IGBT Driver chip (A3120)
    c. Comparator chip (LM211P)
    d. ISO124 Isolating Amplifier
    e. A7520 isolating amplifier
    f. Current Sensor (100A unidirectional Allegro Hall sensor)
    g. 2 or 3 DC-DC converters used to convert 12V to 15V (depending on the type of the driver board you have)

5. Connectors
    a. 2-3 sets of 40 male breakable pins (0.1” pitch)
    b. Assorted female .1 inch headers
    c. 2-pin white female connector + pins; 2-pin male connector. Can be omitted depending

on type of driver board – see below in Driver Board instructions. 0.1" pins can be used instead.

   d. Various terminal types for Power Board and Driver Board
   e. Control & programming / LCD buttons
   f. High-power Input Connector: 4-6 positions (minimum of 3 for Input AC power and 2 for Output DC)

6. Resistors
   a. Most are 1/8 W (1/4W will also work by mounting them vertically)
   b. 3W 10R gate drive resistor
   c. 2-3W 50-100k safety resistors

7. Capacitors
   a. ~10x small electrolytic (labels directly on the caps)
   b. ~20x small ceramic (harder to decipher labeling – generally 3-digit label 'XYZ' means value of XY * 10^Z pF. Example: 104 = 10 * 10^4 = 100000 pF = 0.1uF)
   c. 3x blue film caps used on Power Board
   d. 16x 1500-1800uF 200V Power Capacitors for the voltage doubler Board (generally shipped wrapped together with the corresponding board)
      i. If you live in the country where only 220-240V are available, we may supply a separate bridge board and a single large cap
   e. 5x 560-680uF 400-450V Power Capacitors for the power Board (generally shipped wrapped together with the corresponding board)

8. Color LCD (1.44" diagonal 128x128 resolution) programmed for using a simple serial protocol

9. 2 In-rush limiters: 50 amps max current (use one on each AC input)

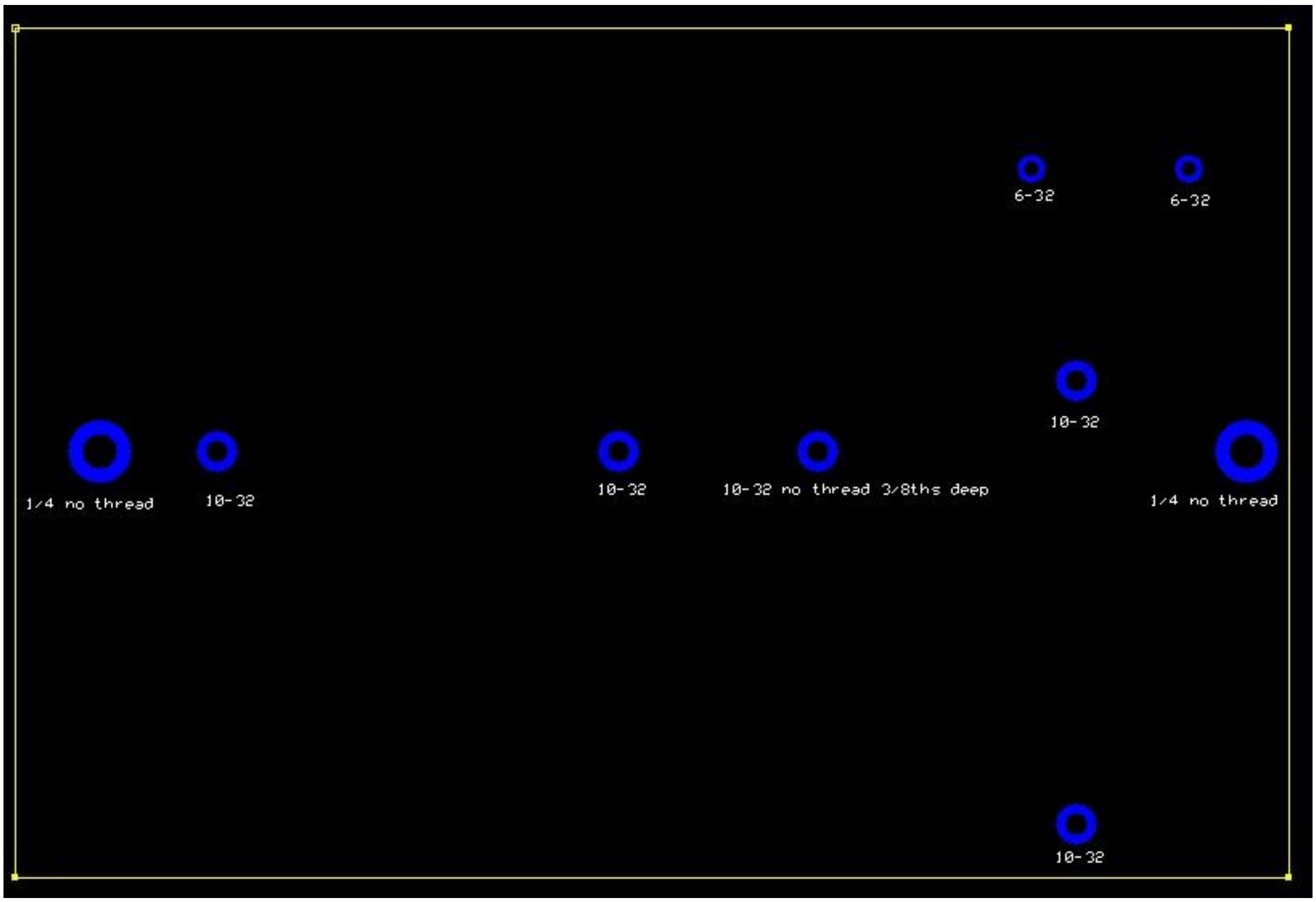
10. Semiconductors:
   a. Small N2222 transistors
   b. 2x small +5V regulator (78L05) and 1x small -5V regulator (79L05). Note that these will look identical to N2222 transistors – make sure you separate them by label
   c. 5V linear regulator
   d. 1x signal diode
   e. 2x Diode Bridges used on the doubler board
   f. 1x IGBTs
      i. 150-200 amps, 600V
      ii. Some IGBTs might have slightly different positions of tabs, which may need to be bent slightly to match the Power Board layout

11. 12V regulated AC supply rated for 2-5 amps (to power the fans and all circuitry in kit)

12. Optional Enclosure and Fans
    a. Option 1: standard enclosure
        i. 10x10x8" steel box
        ii. Needs to be drilled / cut to fit fans, LCD, heatsinks, etc.
    b. Option 2 (Recommended for much easier, faster, and error-free assembly): Fully machined enclosure matching your kit:
        i. No drilling / cutting required. Save yourself 20+ hours of metal work
        ii. Cooling is further optimized by having long slots on the back – something nearly impossible to do by hand.
        iii. Make your charger build look professional. Chances are, you will bang up the standard enclosure pretty badly as you fabricate it. With the fully machined enclosure to start with, your build will look pristine when finished.
        iv. Here is how it looks:

Part 2. Marking up the Heatsink

1. **AIR COOLED:** Prepare the top of the heatsink (NOTE: make sure most of the holes are drilled between the fins) according to the below picture. The associated ExpressPCB file is available from the build info folder on EMW site – request if you can't find it. This file will give you exact dimensions between mounting points.



From left to right, using a 9x6" heatsink you have received from EMW with its long side (9") facing you and the side **with cut lip facing to the right**:

a. Mark up 2 holes for mounting the heatsink itself, in between the two final fins on each side of the heatsink (Center Left side and center right side). Drill with ¼" bit.
b. 2 holes for mounting the diode bridges, tap with 10-32 or 10-24 thread. Tip: best to use an unpopulated voltage doubler board as a pattern for holes.
c. 3/8" deep thermistor hole (no thread, use the same bit as you used for 10-32 holes)
d. 2 holes for IGBT, tap with 10-32 or 10-24 thread. Tip: best to use an unpopulated main non-pfc power board as a pattern for holes.

e. Drill 2 holes for the output diode on the top right side of the sink (6-32), tap with 6-32 thread. Align the diode so that the anode is closer to the power board output pads as it will be connected to the power board. Note that the diode supplied is a dual package in which you have to use both diodes connected in parallel. Place a diode on the top right corner of the sink
   i. Align mounting holes horizontally, with an open hole facing right
   ii. Center of the open hole should be ~20mm from the top and 20mm from the right edge of the sink.
   iii. Mark up the holes
   iv. In this configuration, the anode of the output diode will be positioned directly under the output pad of the PCB, which will simplify the assembly

f. Make a 3-pin (only 2 side pins used) thermistor connector.
   i. Take 2 12" signal wires and solder one end to thermistor
   ii. Heat-shrink-wrap thermistor connections and thermistor itself to prevent shorting to heatsink
   iii. Solder other ends of the wires to the 3/4-pin female 0.1" pitch header (use positions 1 and 3)

   Using silicone sealant, secure the thermistor in the thermistor hole

   Some pics here:

   iv. 1.

   v. 2

vi. 3.

Using heat glue or super-glue, secure the thermistor in the thermistor hole

g. Mount the output diode
   i. Make sure you use thermal paste between diode and sink/plate
   ii. Note that the diode supplied is a dual package in which you have to use both diodes connected in parallel
   iii. Prep the diode connection wires
      i. Cut 10” length of the bare copper wire gauge 10/12. Cut in half so you get 2 5” bare copper wires.
      ii. Anode Wire:
         1. Make a ¾”-long oval loop on one end of a wire, solder the end to

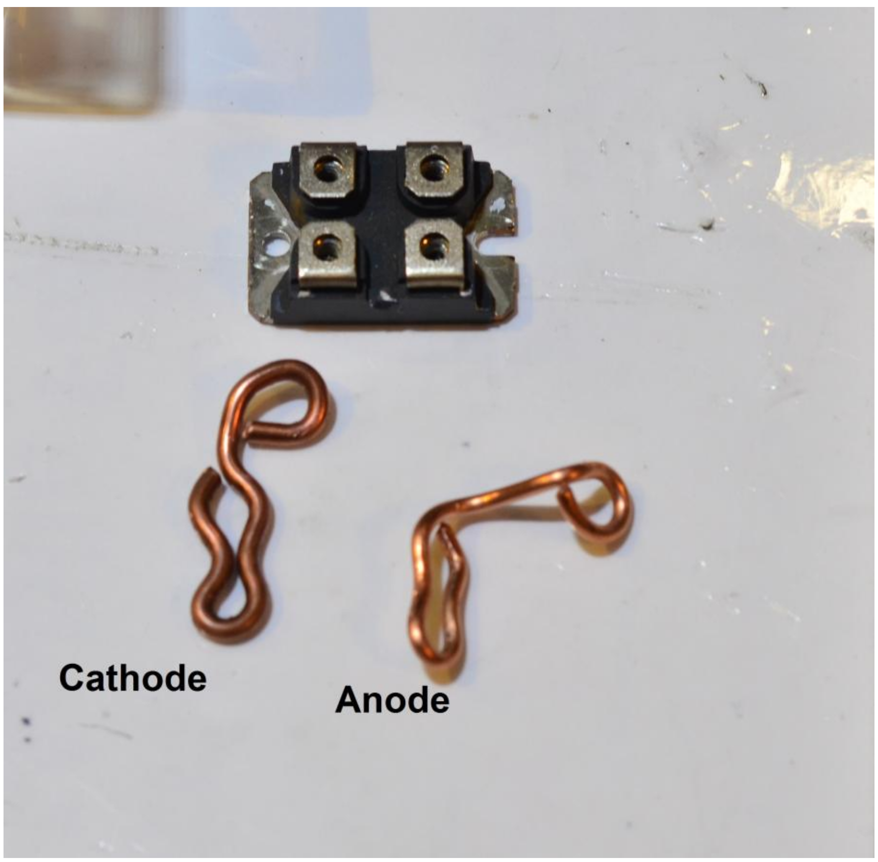
the wire closing the loop.

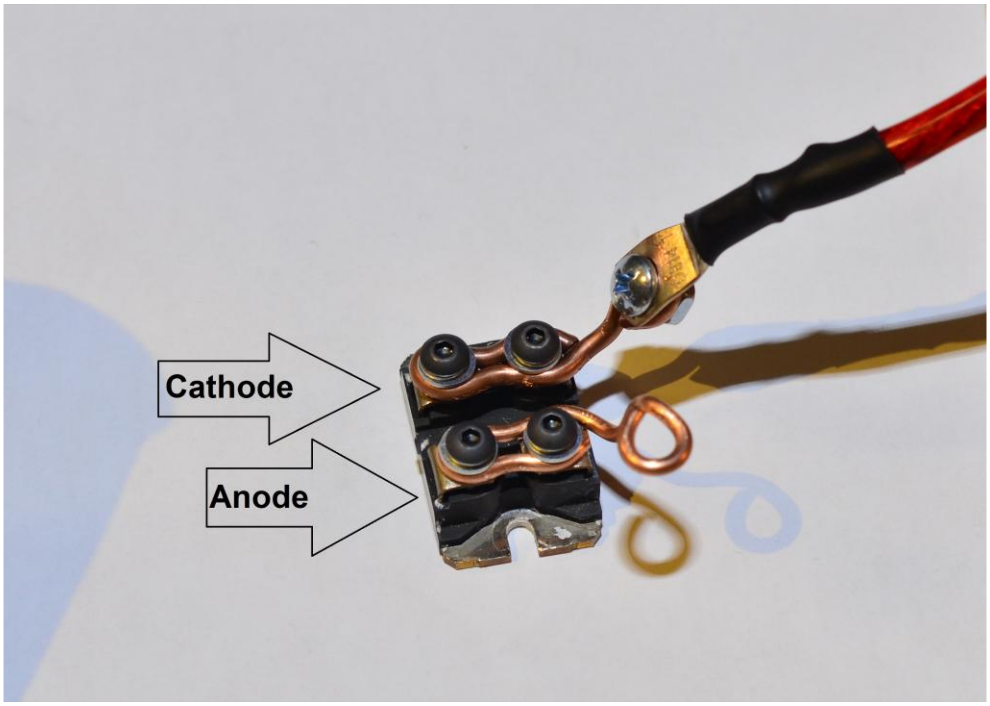
2. This will be mounted to the output diode (using the double diode assembly, with 2 diodes connected in parallel).
3. Screw-mount the loop onto the anode terminal[s].
4. Bend the other end of the wire to make a ½" loop which will be connected to the upper right corner of the power board.

iii. Cathode Wire:

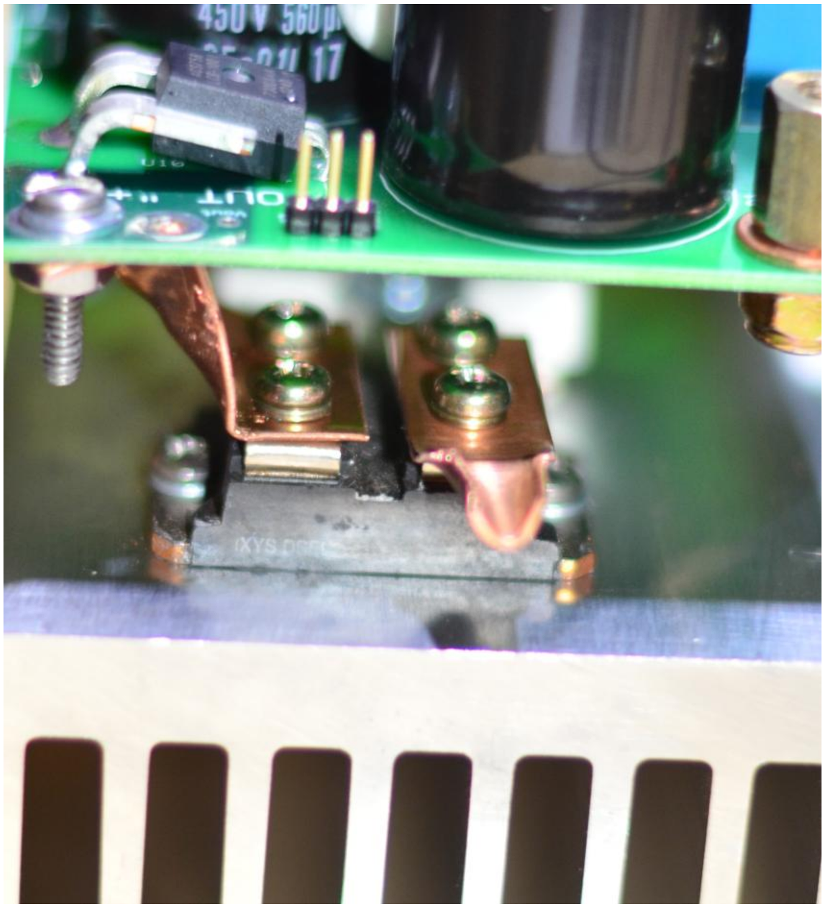
1. Make an identical 3/4" long oval loop on the second wire so that the diodes' cathodes are also connected in parallel.
2. Screw-mount the loop onto the cathode terminal[s].
3. Make a ½" loop at the other end of the wire which will be used for connecting to the charger output wire

iv. See pictures below:

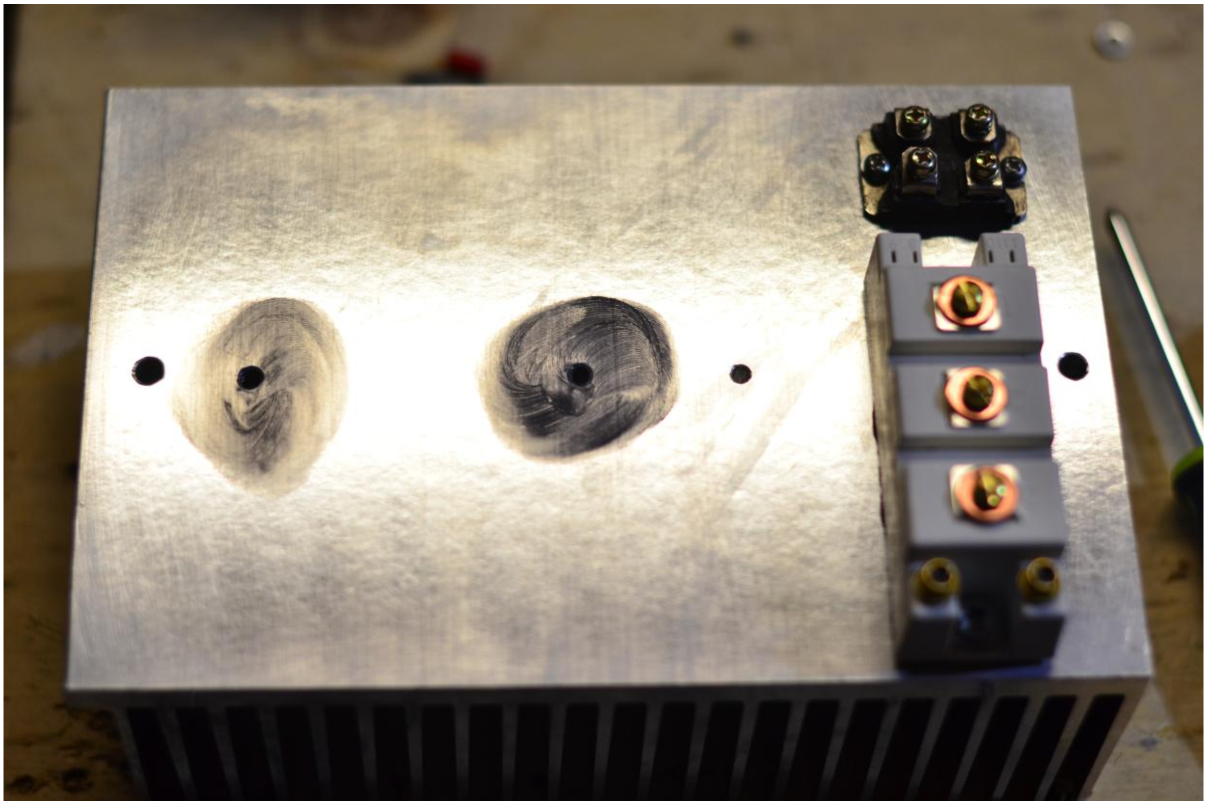




v. Test-fit the assembly under the power board, aligning the Anode loop under the B “+” (can also be marked as OUT “+”) pad in the upper right corner of the power board
   1. Note that on the image below, we used copper sheets instead of the wires. Either method is fine and wires are probably easier to make if you don’t have copper sheets lying around like we do…

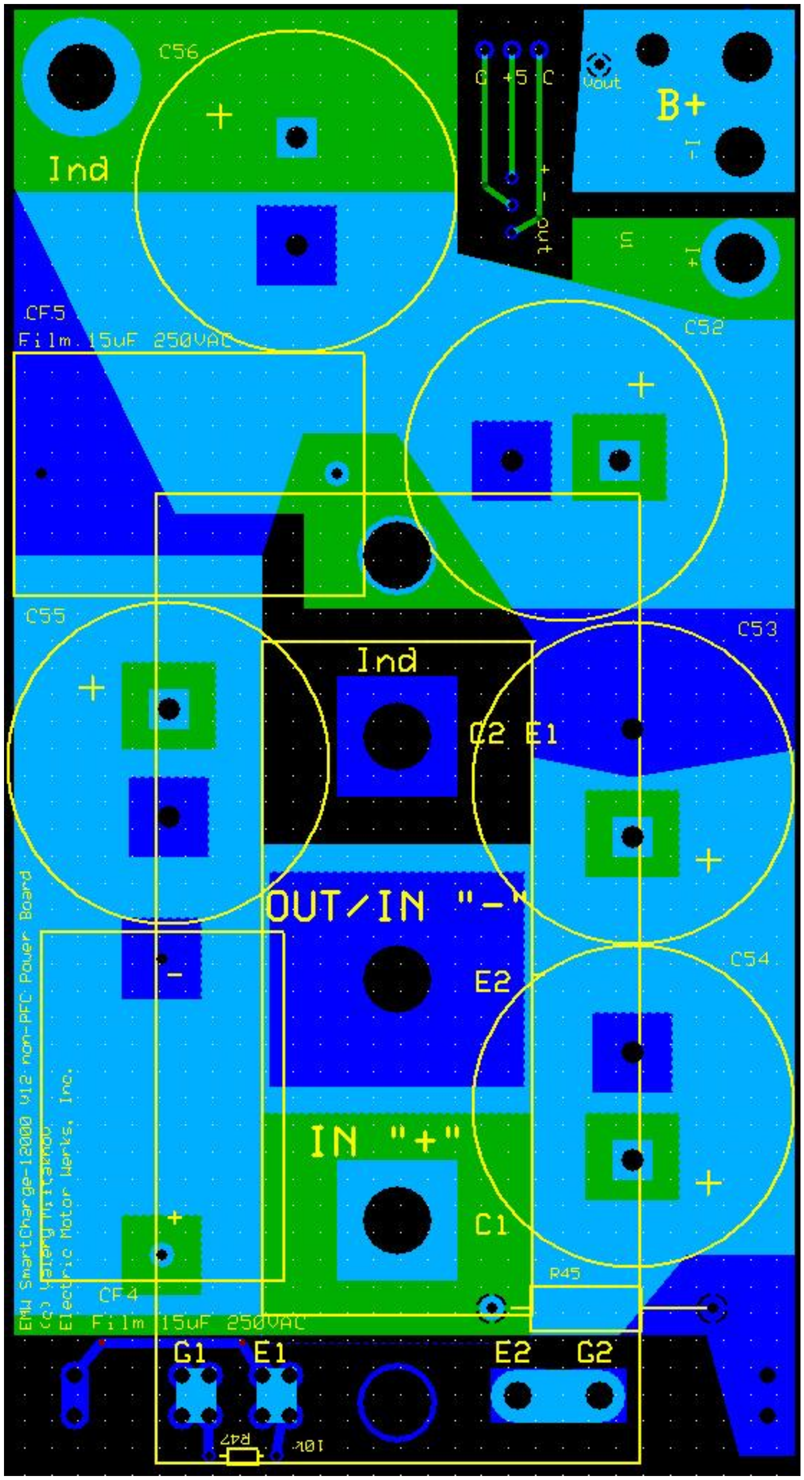


h. Here's how the components will be placed on the sink (do NOT mount anything except the output diode yet!) The picture already includes the placed IGBT but it is recommended to mount IGBT after connecting it with power board. Also, the output diode here is shown without the copper diode connection attachments.

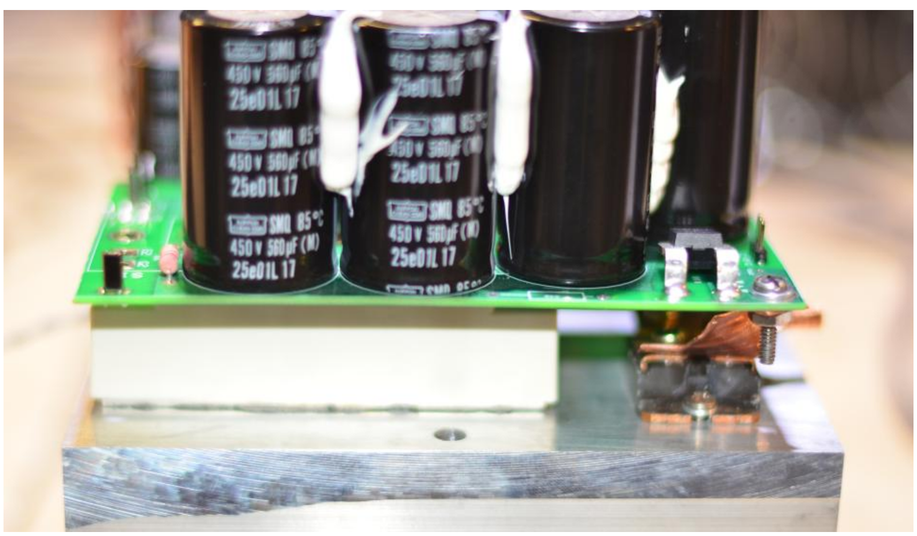


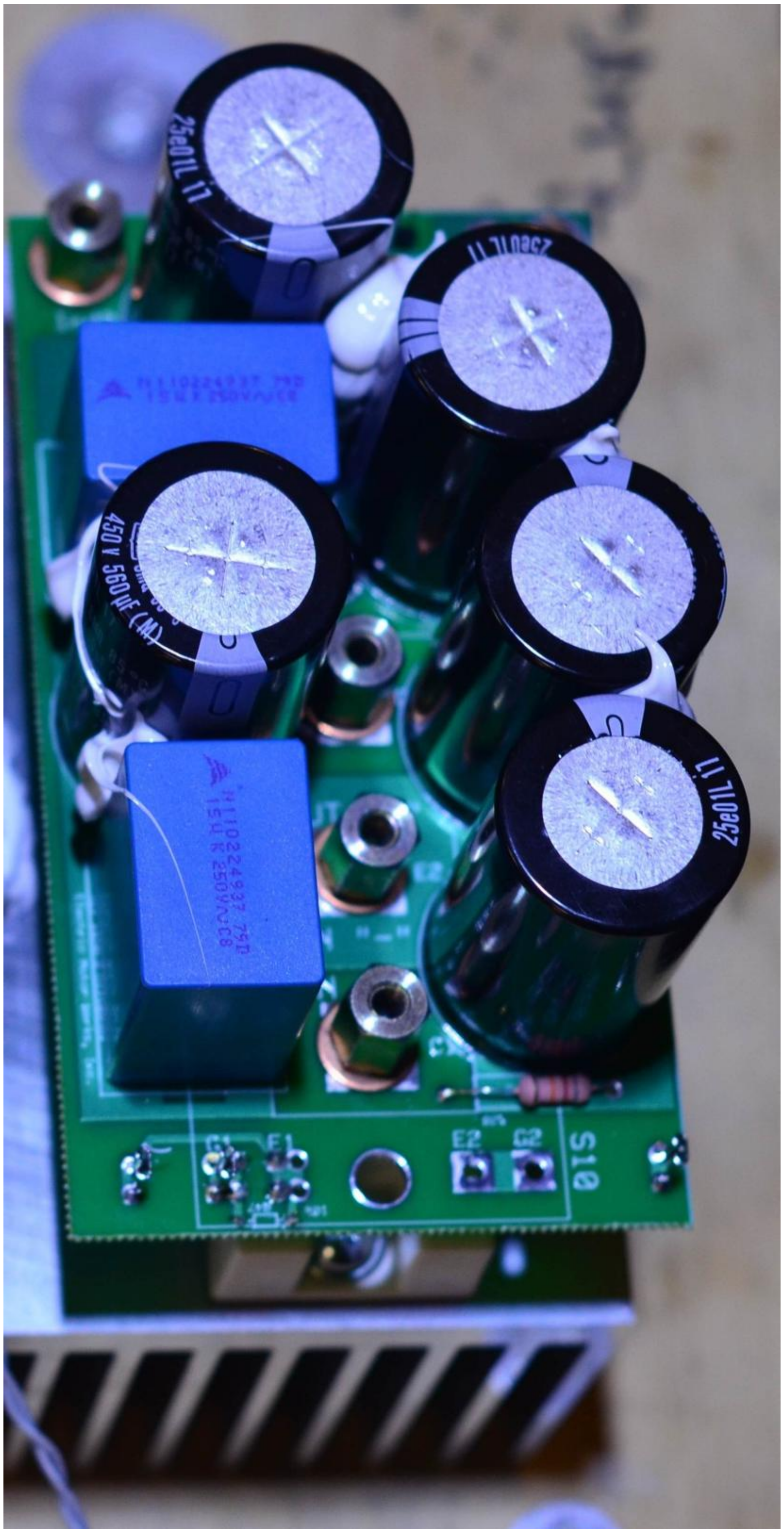
2. **LIQUID COOLED** - contact EMW for instructions. Generally speaking, liquid-cooled non-PFC units are possible but require more complex assembly – with doubler boards and power board being on opposite sides of the cold plate.

Part 3. Main Power Board



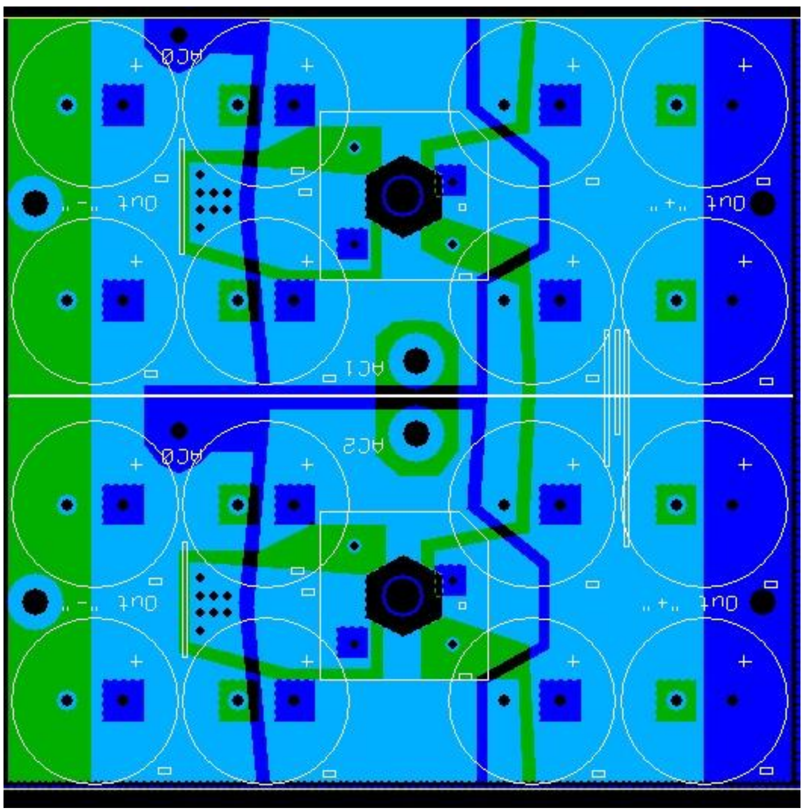
a. Pre-assemble the top of the power board
    i. Solder Resistors: (R47, R45) from the top of the board
    ii. Solder a 100A hall sensor and the corresponding 3-pin connector from the top
    iii. Solder 2 Blue film caps from the top
    iv. Solder 5 Elcaps from the top (observe polarity!!!)
b. Prepare the IGBT
    i. Mount the IGBT on a heatsink (don’t forget to apply thermal paste between the IGBT and the heatsink)
    ii. Solder together the E2 and G2 pins of the IGBT
    iii. Fit the 2 supplied female spade connectors on the E1 and G1 pins of the IGBT
c. Place the pre-assembled power board on top of the IGBT, fitting the pins of the spade connector through their mounting holes from the bottom of the board. You might need to bend the IGBT pins slightly to make a fit. Secure the board to IGBT’s top terminal; solder the spade connector pins.
d. Place the third female spade connector from the top of the board (on the right side close to you). Solder from the bottom
e. Apply silicone between all caps to protect against vibration
f. This is what it should look like after all is done:
    i. Note that the standoffs in the picture are optional – we use them to ease the assembly & testing of our fully built units but they are not required




450 V 560µF (M)
G1
E1
E2
G2
S10

Part 4. Voltage Doubler

This is non-PFC-specific board. It's function is to rectify the AC and also provide possibility for doubling the input 120VAC to get the same rectified DC as one would get from a 240VAC supply. It is a standard diode-capacitor doubler design used in every other OC power suopply



1. Assemble diode bridge/ Voltage Doubler Board
    a. Place bridges from the bottom of the board. Solder from the top side
    b. Fill the through holes between C3 and C4 with solder. This is important as these holes are all that carries the current from one side of the board to the other. If not filled, they may blow up under power
    c. Place all caps and solder
    d. Connect both Out "+" outputs with 8/10-gauge bare copper wire bracket. Do same for both Out "-" outputs.
    e. Wire AC connections
        ii. Cut 12" length of gauge 6/8 wire, connect one end to AC1 output (soldering directly or via a bare copper stud). This will be the shared 110/220VAC input.

- iii. Cut 12” length of gauge 6/8 wire, connect one end to AC2 output. This will be the second input line for 220VAC.
- iv. Cut 12” length of gauge 12 wire, solder one end to AC0 output. This will be the second input line for 110VAC.

f. Wire DC connections between voltage doubler and power board
- v. Cut 8” length of gauge 8 wire (ideally red color). Fit eye crimp connector to one side, solder another to the copper bracket connecting Out “+” outputs.
- vi. Cut 8” length of gauge 8 wire (ideally black color). Fit eye crimp connector to one side, solder another to the copper bracket connecting Out “-” outputs.
- vii. Cut 10” length of gauge 8 wire (ideally black color). Fit eye crimp connectors to both sides.
- viii. Take the two black wires from III and IV steps above and screw-mount to the middle (E2) terminal of the IGBT. The second wire (the one that is NOT going to the voltage doubler) will be your charger negative output.
- ix. Screw-mount the red wire from voltage doubler to the bottom (C1) terminal of the IGBT

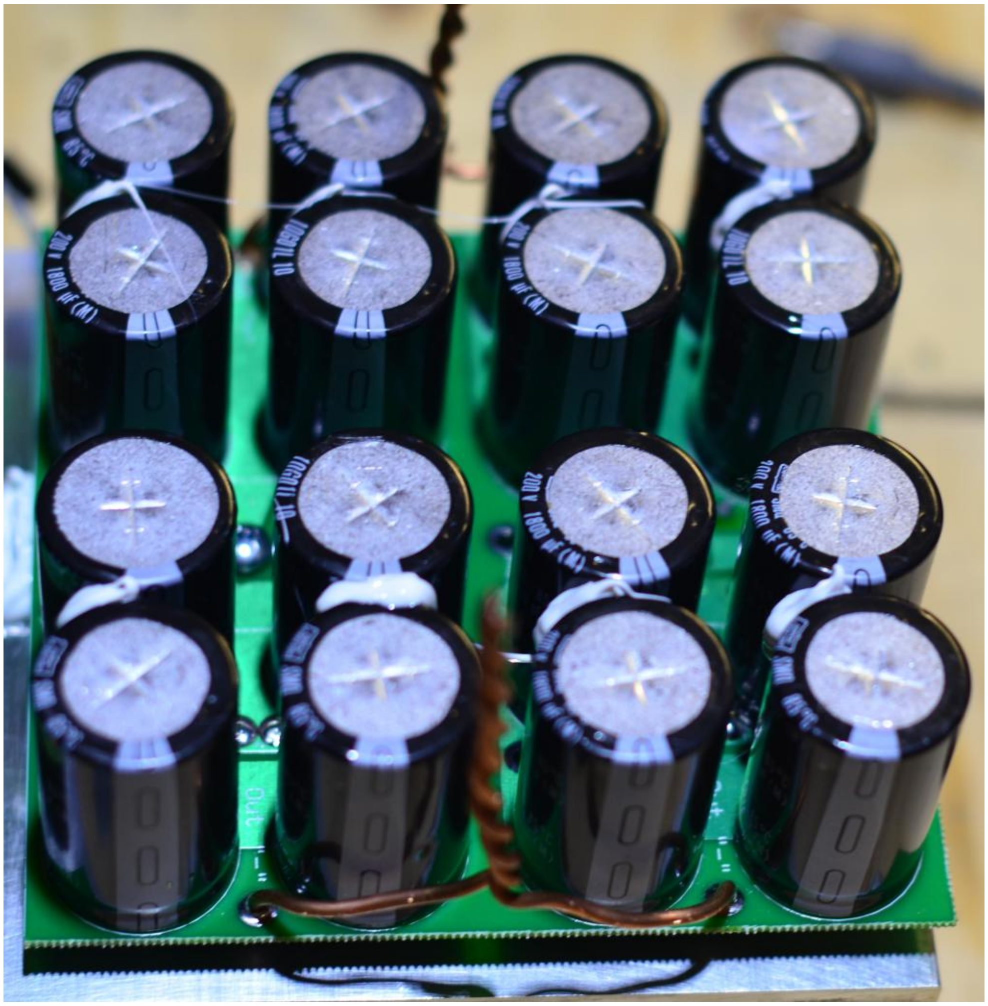
g. Wire the output of the power board to the output relay or diode
- x. OPTION 1 – diode
  1. Solder or screw the other end of the anode copper wire (the one you have mounted to the anode earlier) to one of the OUT “+” pads in the upper right corner of the power board
  2. Cut 10” length of a red wire gauge 8. Fit eye crimp connector to one side. Screw-mount the other side to the cathode copper wire that you mounted to the output diode earlier. This will be the main output of the charger
- xi. Option 2 – relay (not supplied in the kit)
  1. Same steps , only instead of the diode, use relay rated for at least 240VAC, 60A (some 2/3-pole relays rated for 30A can be used if you parallel the poles together)
  2. Ensure connection of the relay to the control board (see below in control board section for instructions)
  3. Connect pre-charge circuit across the relay
     a. Connect a 330R 10W resistor in series with a 1A 600V rectifier diode (anode to resistor)
     b. Connect resistor end of the assembly to the output of the relay, cathode end of the assembly to the input of the relay

h. Apply silicone between all caps to protect against vibration

i. Bolt the bridge board to the heatsink (don’t forget to apply the thermal grease to the bridge surfaces)

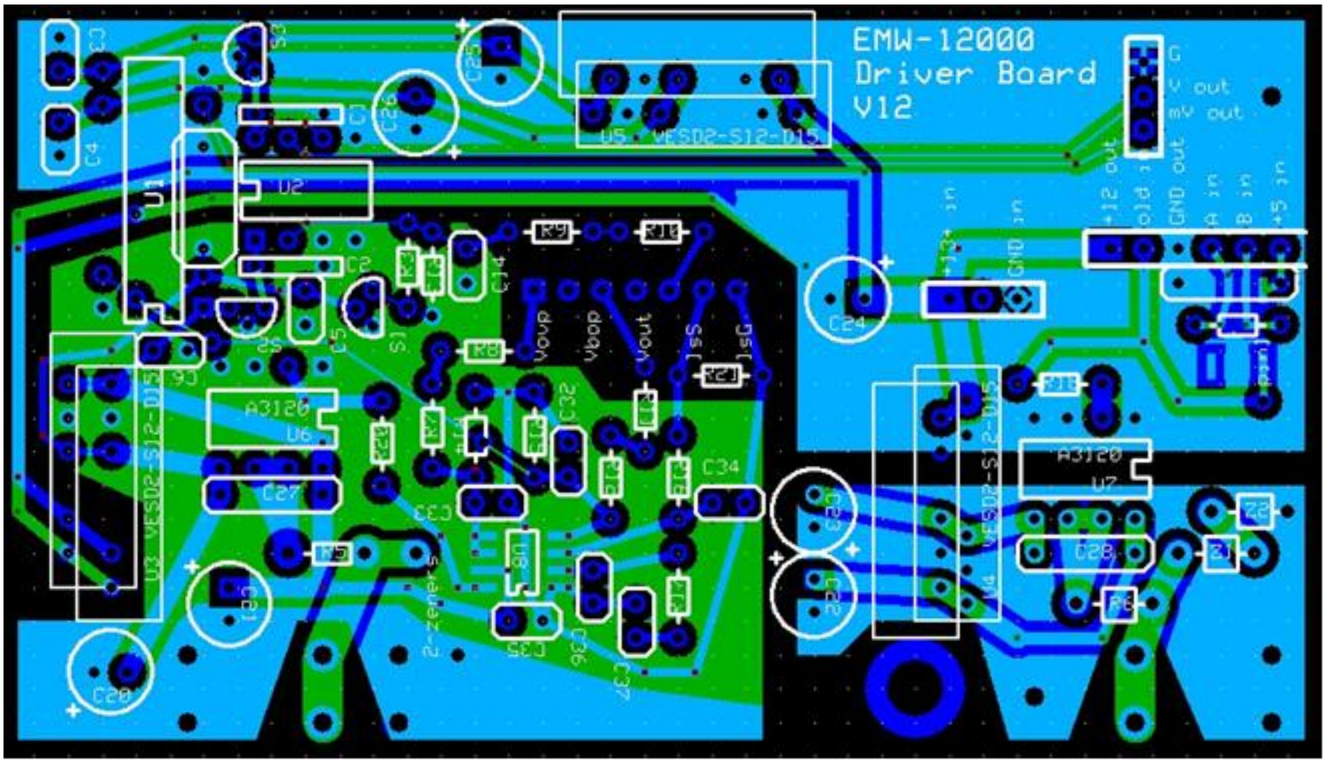
j. Apply silicone under 4 corners of the board (between the board and the heatsink) to protect against shorting during board flexing.

1. Here's how it should look (approximately) after you're done: (picture does not show wiring)



Part 5. Driver Board

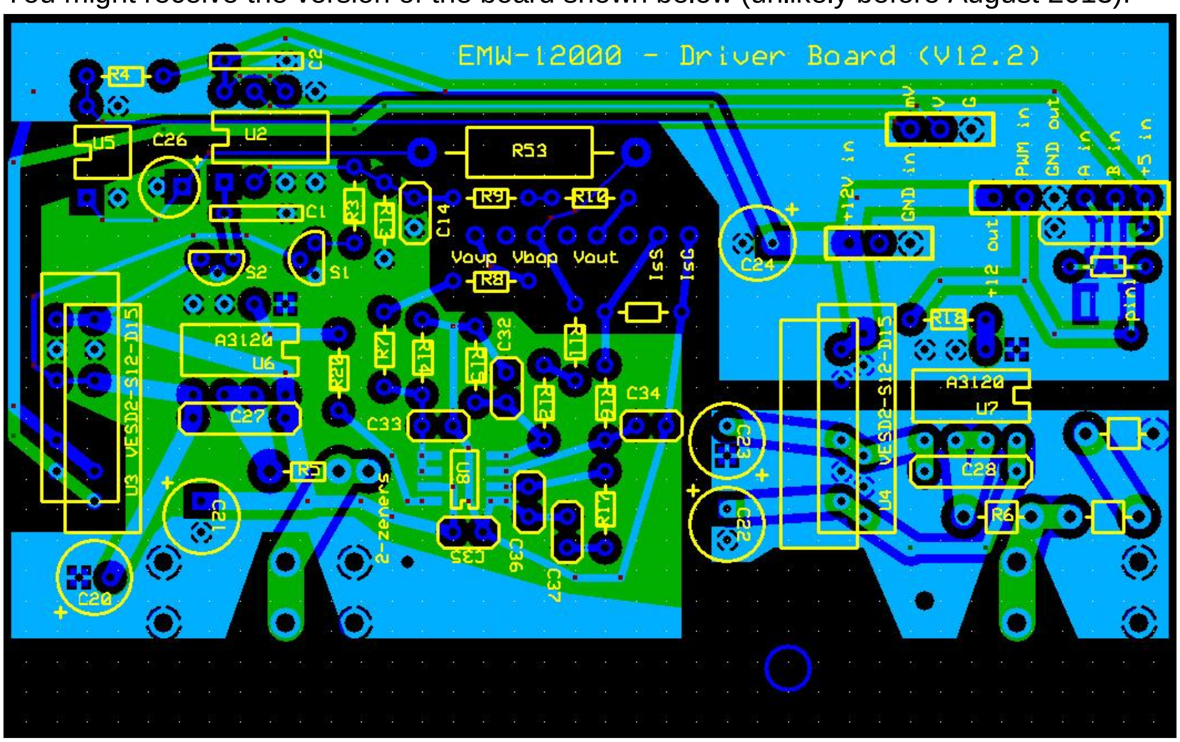
There are several different types of Driver boards. Most likely you will get this one: V2



1. **Assemble the Driver board.** Note that there will be a number of unoccupied positions on the driver board after you are done – the “missing” parts are for PFC stage control and are not needed in non-PFC version
    1. Place ISO124 isolating op-amp: U1
    2. Place HCPL-7520: U2
    3. Place 8-dip socket for U7 (if supplied) or solder A3120 chip: U7
        a. U6 is NOT used in the non-PFC version
    4. Turn over, solder
    5. Place 1uF caps: C32
    6. Place 1M resistors: R9, R8, R9, R10, R11, R12
    7. Place 27k resistors: R15
    8. Place 220R resistor: R18
    9. Place 0.1uF caps: C1, C2, C3, C4, C5, C27, C28, C34
    10. Place 12V->15V DC-DC converters: U3, U4, U5
    11. Place 10R resistor: R6
    12. Place 68k resistor: R3

13. Place 3.3-4.7uF cap(3.3uF capacitor supplied): C14(could be labeled as C15, C16, or C57)
14. Hold DC-DC converters with a piece of carton/plastic and turn the board over. Solder all components, trim leads
15. Place 50uF caps: C20, C21, C22, C23, C24, C25, C26
16. Place 2-pin high voltage connector into Vbop and Vout mounting pads on the driver board. Align the connector with the locking tab facing down.
    a. EMW may provide 5-pin connector or extra .01'' male pin headers to suit new pcb board design – in that case place all pins but use only Vbop and Vout pins later on
17. Place Positive 5V regulators-78L05: S2, S3
18. Place Negative 5V regulator-79L05: S1
19. Turn over, solder
20. FROM THE BACK SIDE
    a. Insert 2 3-pin and 1 6-pin male pin headers
    b. Hold parts with carton/plastic, turn over, solder
    c. Turn over, solder
21. Insert 3 male 90-degree spade connectors into the female spade connectors on the buck power board, mounting pins facing you. Refer to the photo below for pin placement. Solder pins in place.
22. Insert A3120 chip into a 8-pin IC socket if not soldered by itself already.

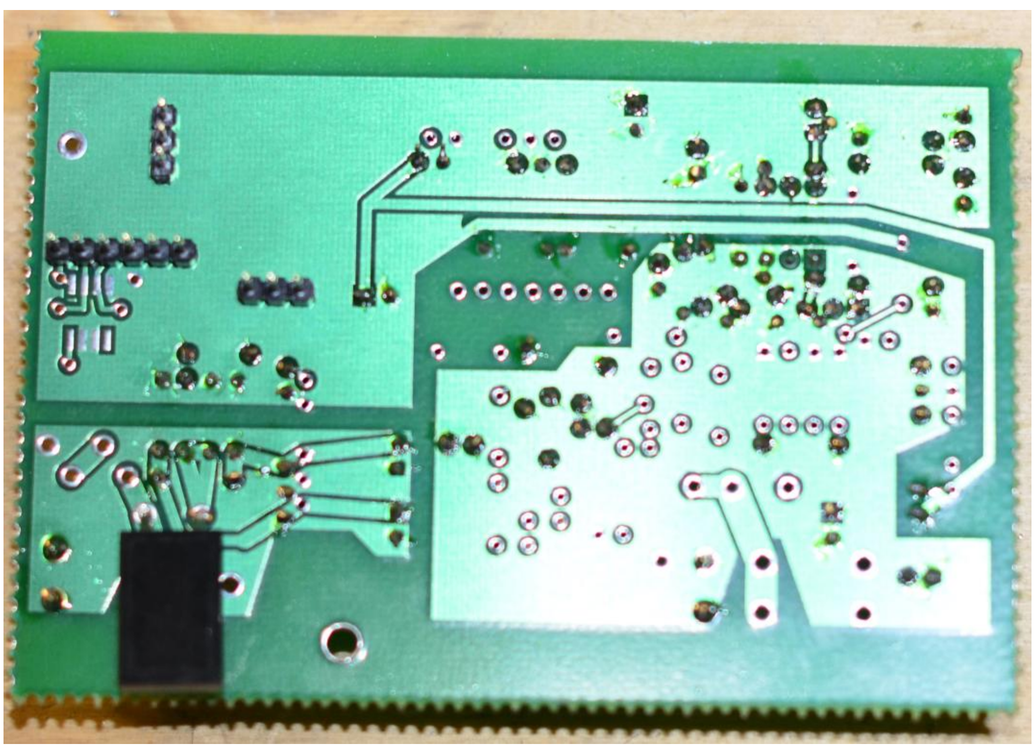
You might receive the version of the board shown below (unlikely before August 2013):

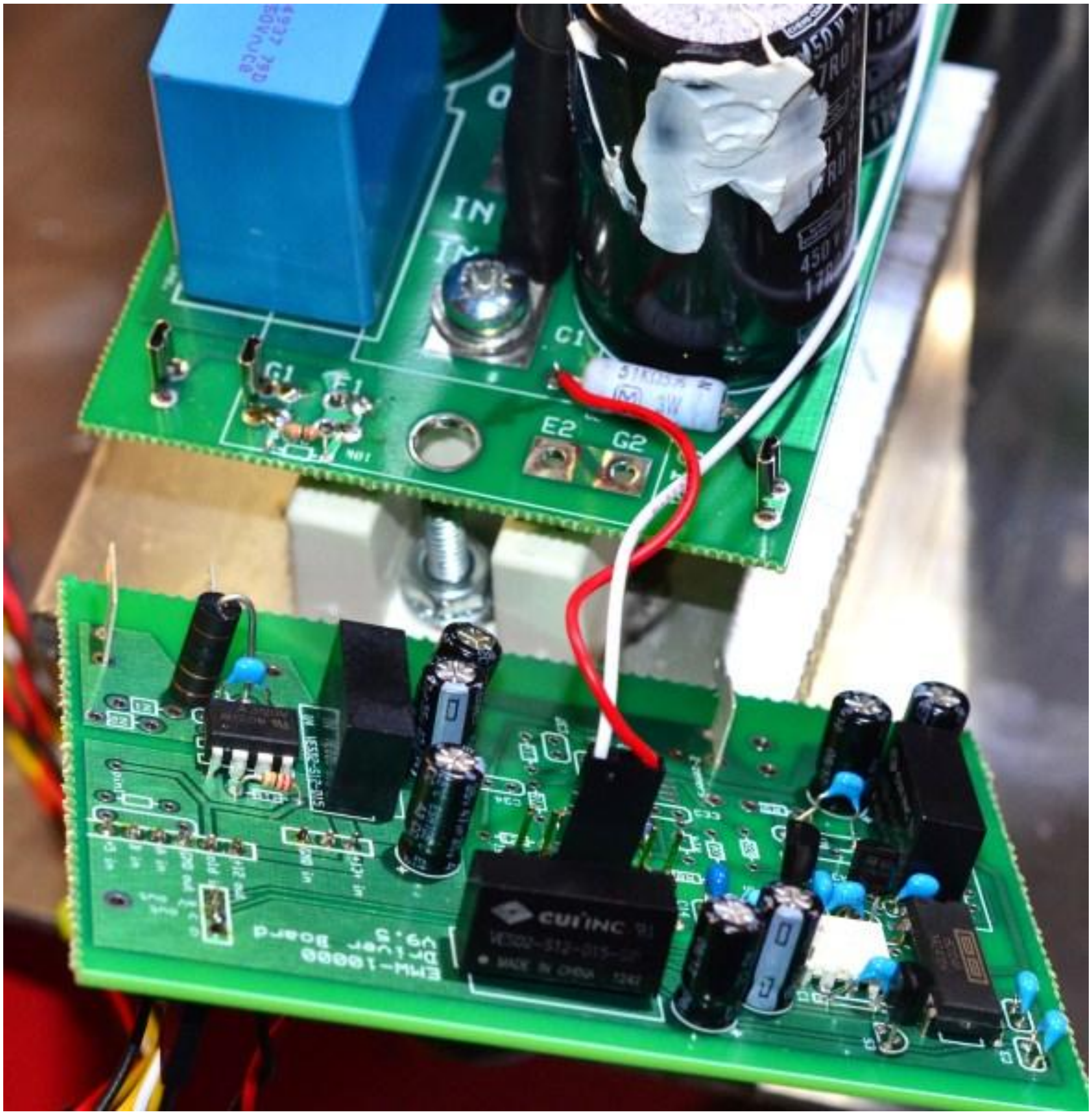


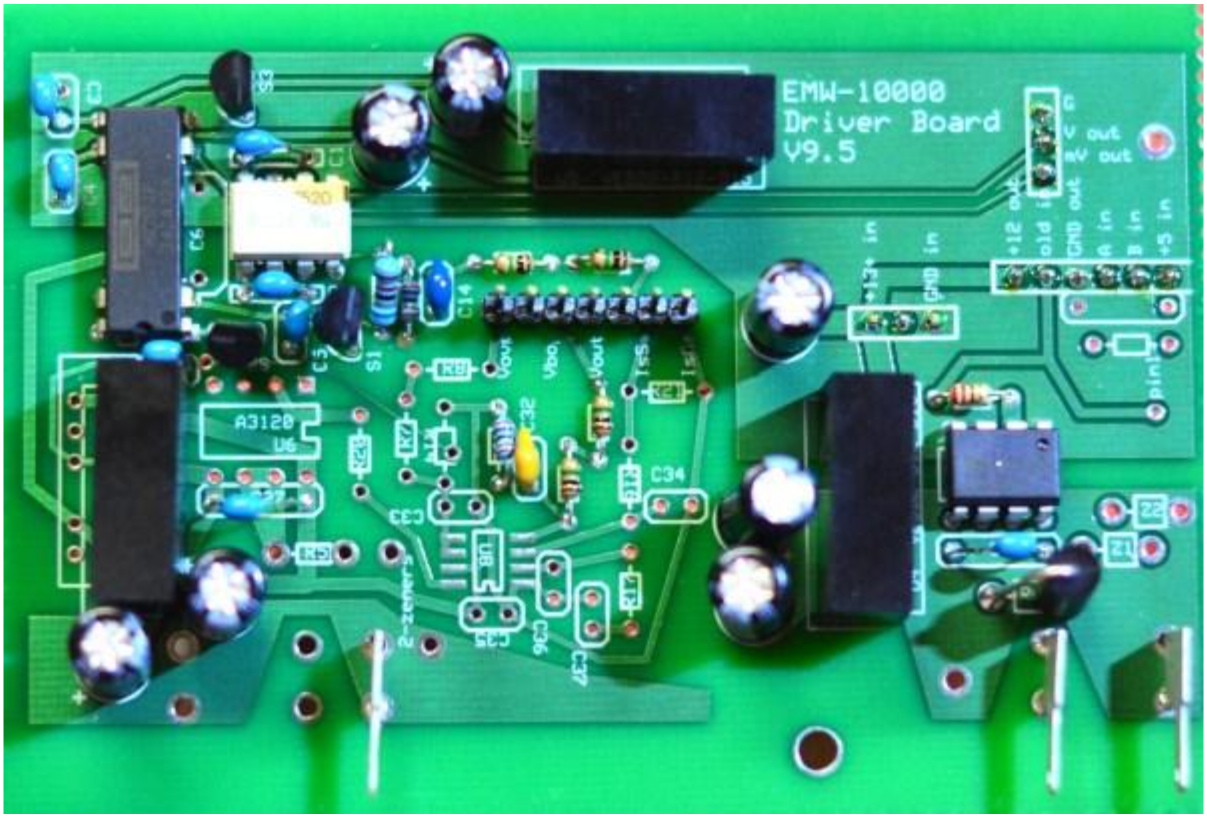
a. This version uses much simpler version of the input voltage measurement as the charger does not need precision on input voltage measurement
b. Instructions for this version will be provided as it gets released

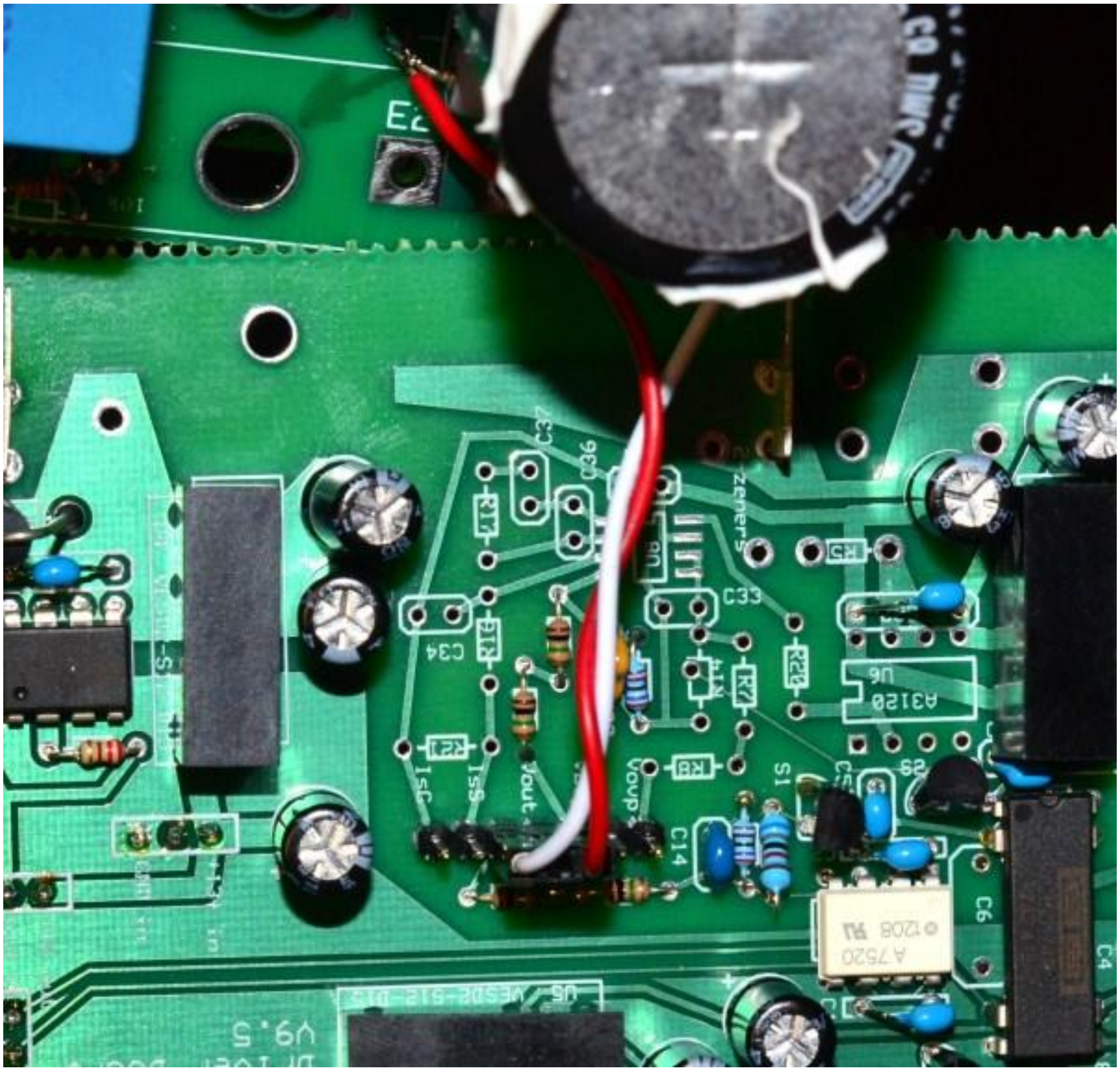
ii. Build a connection harness for HV connector
1. Cut a 12" length of signal wire (AWG 22-16), crimp one end into the female pin of the 2-pin power connector (that pin will then be inserted in the connector housing so that it will mate to the Vout male pin on the driver board)
2. Cut a 4" length of signal wire, crimp one end into the other female pin (mating to the Vbop male pin).
3. Solder Vbop wire to the left lead of the R45 resistor. This will be used to sense input voltage
4. Solder Vout wire to the cathode of output diode (or, if using output relay, to the output pin of the relay). This will be used to sense output voltage

iii. Here's how the board should look after you are done:




C1
G1
E1
E2
G2
CUI INC


EMW-10000
Driver Board
V9.5
A3120
U6

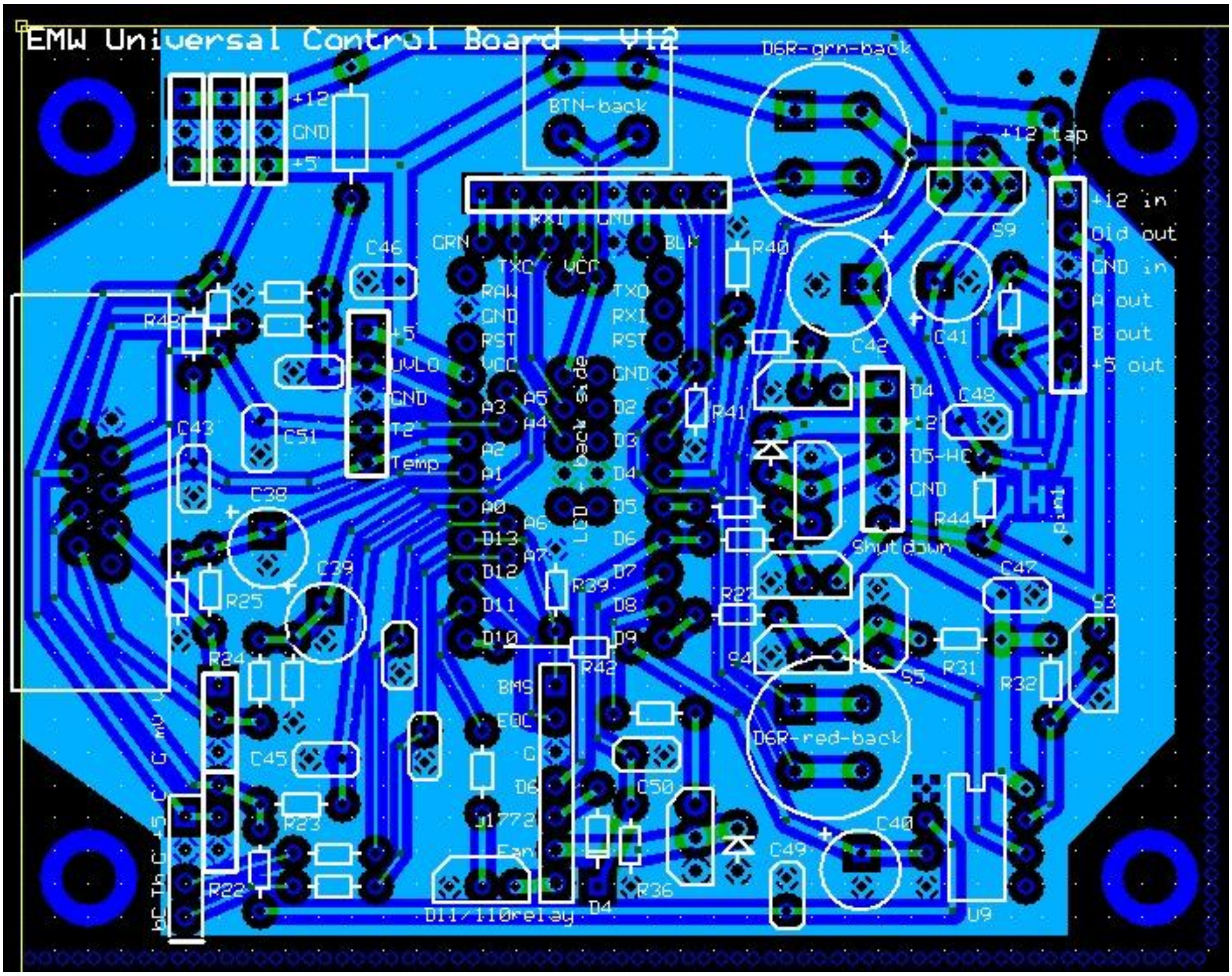


iv. Insert the driver board into the female spade pins on the power board: NOTE ORIENTATION – Component side of the board should face OUTSIDE.

Part 6. Control Board

Notes:

1. There will be a number of unoccupied positions on the control board after you are done – the “missing” parts are required only for special configurations / options / multi-stage systems and are not needed in the default version



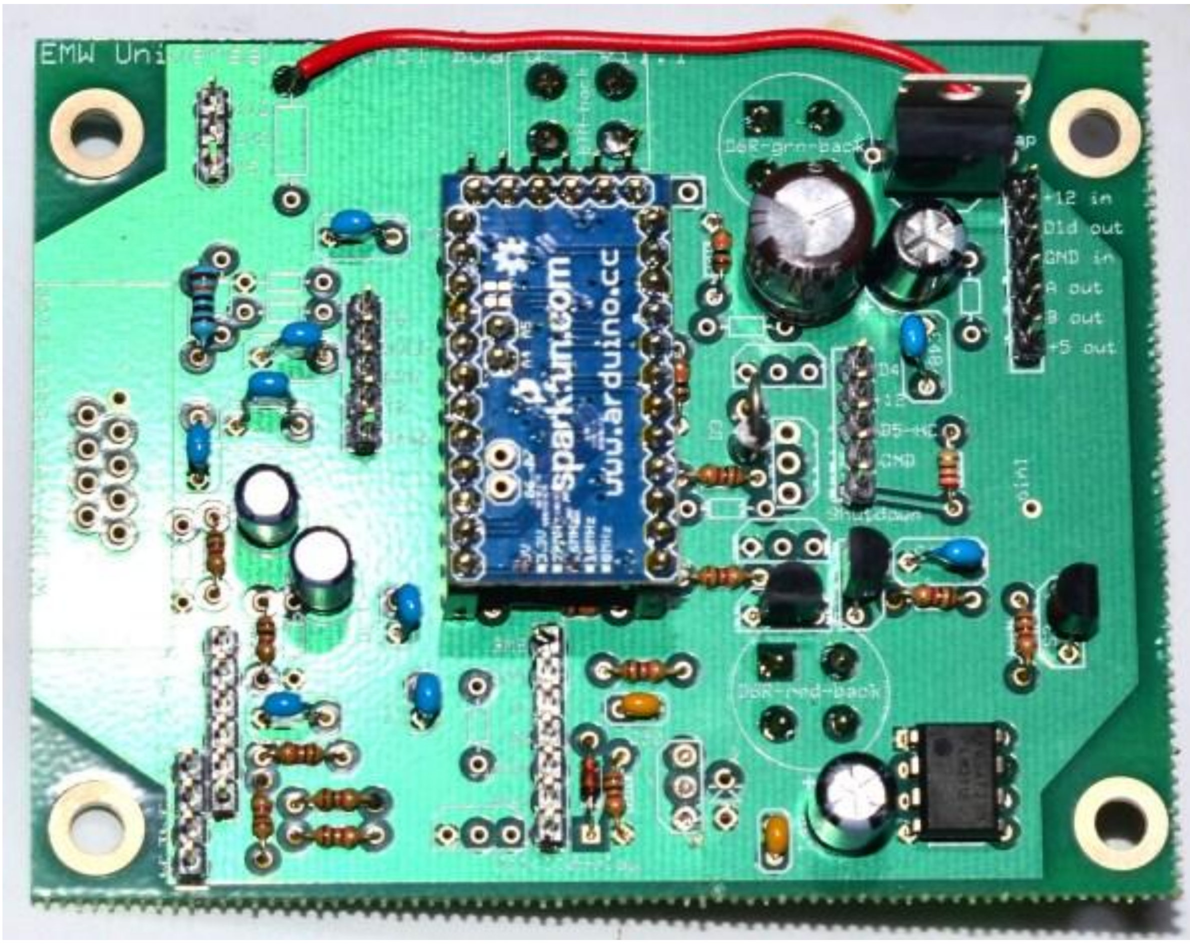
1. Solder the male pins to the Arduino board. You might receive a version of the board with all pins already placed – in that case skip this step (do make sure that all the pads on the Arduino board have pins soldered to – those A4/5 pins on the inside can be easy to miss!)
    1. make sure you place the pins from the component side so that Arduino board components end up facing the control board when Arduino board gets inserted into the control board

    2. do not forget A4 A5 pins on the inside of the board
2. Place Group 1 components
    1. 1k resistors: R22, R23, R24, R25, R27, R29, R31, R32, R36
        a. (ID may not be printed on the board – Ctrl-F in ExpressPCB software to find part on the board)
    2. 10k resistors: R39, R40, R41, R42
    3. 100k resistor: R48
    4. 6.8k resistor: R38
    5. 3.3k resistor: R37
    6. [OPTIONAL – for UVLO shutdown – not needed in PFC]
        a. Place 6.8k resistor: R38
        b. Place 3.3k resistor: R37
    7. 220R resistor: R44
    8. Small signal diode: D4
    9. 8-pin socket (if supplied) or LM211P chip: U9
    10. Turn over, solder; cut the leads
3. Place Group 2
    1. Male pins for board-to-board connectors. Tip: insert all pins, then cover board from the top with a piece of carton/plastic and then turn over and put on the table. Pins will stay in place.
        a. 7-pin under the arduino
        b. 2x 6-pin on the sides of the board
        c. 2x 5-pin on the left and right of Arduino
        d. 8-pin on top of Arduino.
    2. Turn over, Solder; cut the leads
4. Place Group 3
    1. N2222 transistors: S3, S4, S5
    2. 0.1uF caps: C43, C45, C46, C47, C48
    3. 10uF caps: C38, C39
    4. 50-100uF cap: C40
    5. 100-470uF 16V or higher cap: C41
    6. 100uF or higher 25V or higher cap: C42
    7. 5v voltage regulator(LM 7805): S9
    8. 10nF caps: C49, C50
    9. Turn over, solder; cut the leads
5. Form the Arduino socket
    1. Attach female pin headers (2x 12-pin, 6-pin, and 2-pin (latter made out of a 3 or 4-pin header)) to Arduino male pins
    2. Insert the assembly into the control board

    3. Turn over and solder
    4. Pull out Arduino board from this new connector
6. FROM THE BACK SIDE, place buttons ‘D6R-red’, ‘D6R-green’ (or black), and ‘BTN-back’. Turn over, solder
7. Using non-conductive glue, attach a piece of isolating pad to the back of the LCD board (or just place a piece of electrical tape or duct tape on the back of LCD board)
8. FROM THE BACK SIDE of the control board, place the resulting isolated LCD board through the pins in the middle of Arduino mounting area. Solder from the top of the board. To not melt or damage parts on the top board, solder the LCD pins with the solder gun in between pins rather than from the side(female pins often get in the way when soldering pins from the side).
9. Insert LM211 chip (U9) if not done so already
10. Reinsert Arduino board into the socket
11. Here’s how the board should look after you are done:


µLCD-144-G2

2.


sparkfun.com
www.arduino.cc
+12 in
D1d out
GND in
A out
B out
+5 out

Part 7. Assemble the Charger

Note that connectors are best made out of Pololu wires and connector housings (http://www.pololu.com/catalog/category/71 , http://www.pololu.com/catalog/category/70 – note that these are not included in kits).

1. Prepare the enclosure & fans
    i. For standard (non-machined) enclosure, refer to the photos of the machined enclosure to get the idea for how to fabricate
    ii. For the machined enclosure
        1. Orient the bottom part of the enclosure with round fan openings towards you
        2. Use supplied fans and grills to mount fans (will need 8x #10 ½" sheet metal screws or similar)
        3. Prepare fan connection
            a. Solder fan leads in parallel (positive to positive, negative to negative)
            b. Solder a 3-pin female header to the resulting 2 wires
            c. This header will mate to the 3-pin power connector on the control board (using +12 and GND pins only) and will feed the fans with 12V whenever the AC power is applied to the charger
        4. Place the heatsink inside of the enclosure
            a. Flush to the fans (up to 1/8" spacing is ok)
            b. Leave about the same amount of space between heatsink's left / right edges and the sides of the enclosure
        5. Using the heatsink as a template, mark & drill 2 heatsink mounting holes in the bottom of the enclosure
        6. Using 3.5" #10 machine screw, #10 fender washers, and #10 locknuts, mount the sink to the enclosure

2. Mount the control board
    i. Use predrilled location above the fans on the left part of the enclosure's side with fans
    ii. Use 4x 1-1½" 8-32 machine screws with 3/8th nylon spacers, washers, and lock washers or lock nuts.
        1. Spacers are using to create a gap between the board and enclosure – 3/8" is optimal and gets LCD and buttons as close as possible to the box without any mechanical stress
        2. Orient with LCD facing outside, center buttons in their respective holes
    iii. Connect thermistor to the control board

1. Use GND and Temp pins of the 5–pin connector below the Arduino board (all position references are made while looking at the mounted control board from the component side)

3. Wire main charger outputs
   i. Negative output
      1. 12-18" AWG 6/8 black wire (AWG 4/6 if you have opted for a high-power 100A output inductor) – connect to E2 of the IGBT
      2. Very helpful (but not required) to use a 1.5-2" brass standoff here to elevate the mounting point off the board so the wire clears the caps easier
      3. Use lock washers on the screws
   ii. Positive output
      1. If NOT using an output relay
         a. 12" AWG 6/8 red wire (AWG 4/6 if you have opted for a high-power 100A output inductor) – connect to the output pad on the top right of the power board (or cathode of the output diode if using output diode per instructions in the power board section above)
         b. Use lock washers / lock nuts
      2. If USING an output relay
         a. Use relay rated for at least 240VAC, 60A (some 2-3-pole relays rated for 30A can be used if you parallel the poles together)
            i. Since you are not planning to break connection under current, you are ok with the AC relay
            ii. You will need a 90-100A relay if you have opted for a 100A output configuration
         b. Wire the signal connection of the relay to the control board (refer to the firmware for the right pin allocation)
         c. Connect the pre-charge circuit across the relay
            i. Connect a 330R 10W resistor in series with a 1A 600V rectifier diode (anode to resistor)
            ii. Connect resistor end of the assembly to the output of the relay, cathode end of the assembly to the input of the relay
         d. Wire output pad of the charger to the input of the relay (some DC relays / contactors will have defined polarity)
         e. Connect a 12" AWG6-8 red wire to the output of the relay – this will be the positive output of the charger
   iii. Route wires through the cooling slots on the back of the enclosure

4. Cut AC adapter's cord at 12" from the adapter; solder one AC input wire to Out "-" copper bracket on the doubler board, another line input wire – to Out "+" bracket. Connect 12V adapter to the driver board
   i. Cut input/output wires of the AC adapter, leaving ~8-12" on output DC side

ii. Remove adapter from its enclosure

iii. Solder a 3-pin 0.1"-pitch female header to the output DC wires (use positions 1 and 3). This will later be connected to the power input pins on the driver board

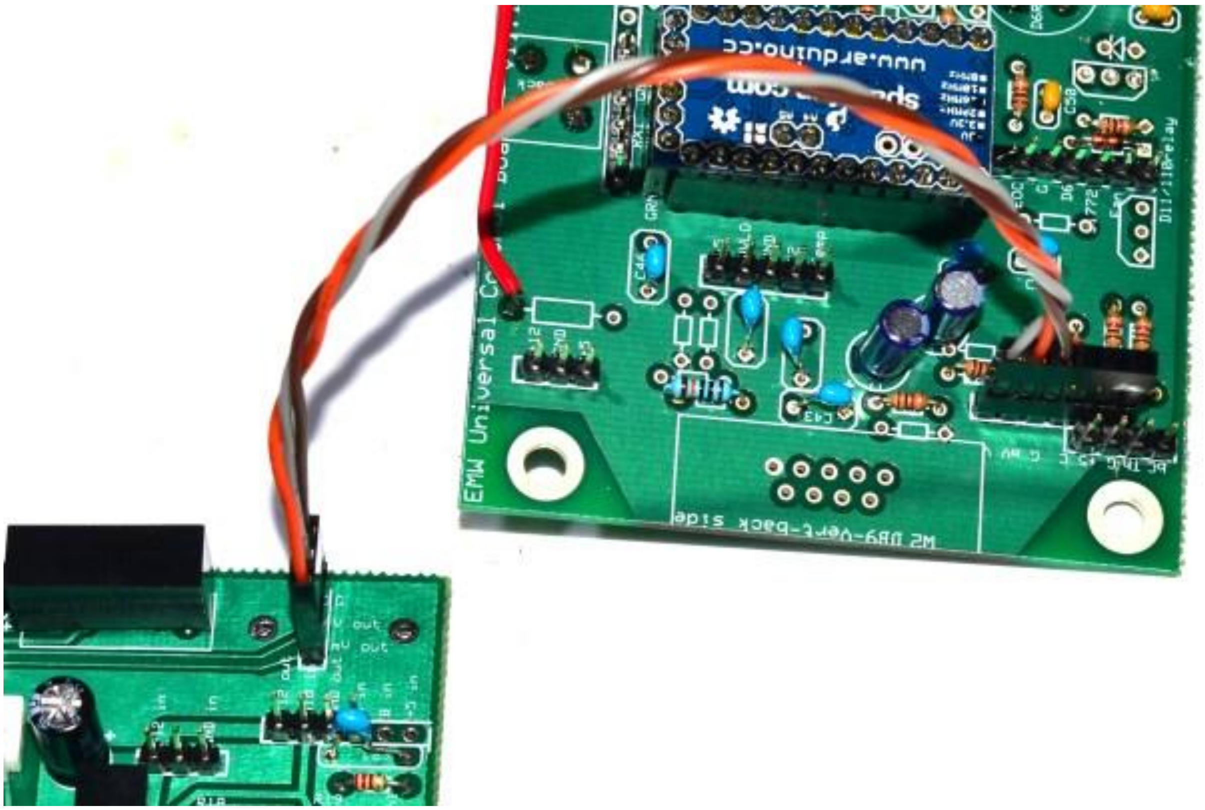
iv. Using silicone, mount the adapter on top of the elcap block on the left side of the doubler board. Orient the adapter so that the output DC wires go towards you

v. Set aside for 30 min to let silicone set

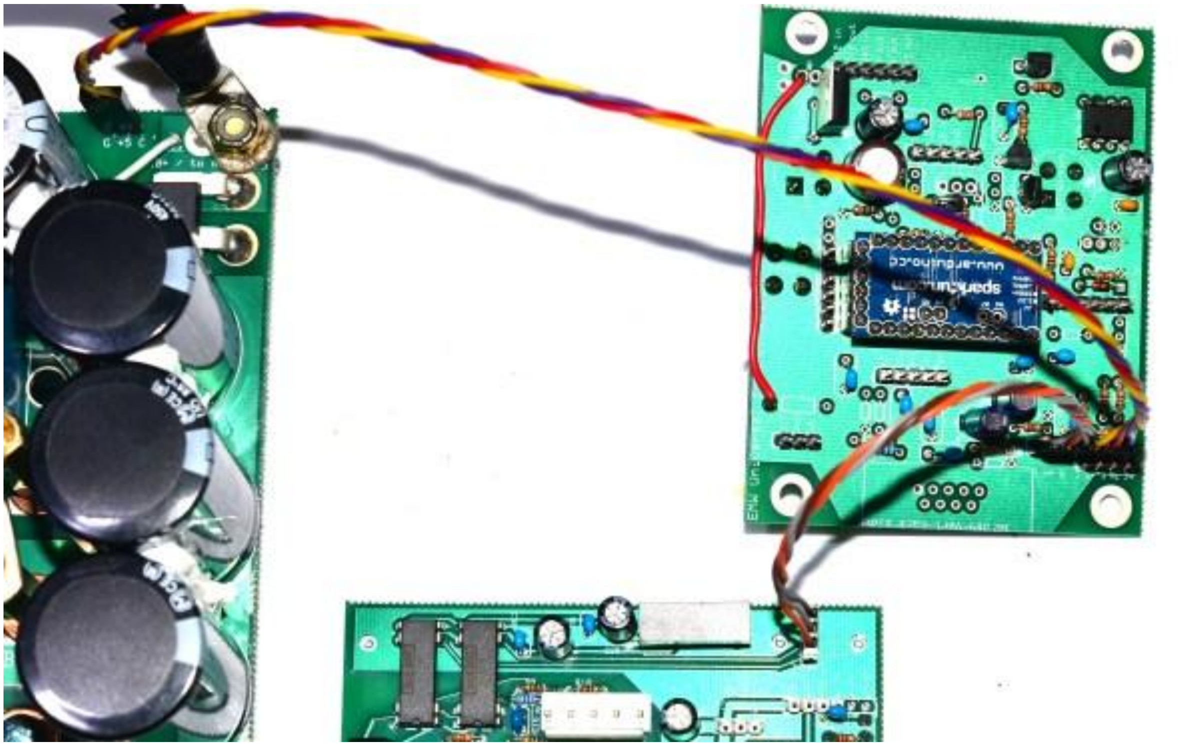
vi. Some pics:

5. Prepare signal connection harnesses – you will need a total of 4 required harnesses and 1 optional. Twist all wires tightly to improve noise immunity!
   i. Programming harness
      1. Need: 6x 12" wires, 2x 6-pin female headers
      2. Simple 6-to-6 harness, used to program the board
   ii. Voltage sensing
      1. Need: 3x 6" wires, 1x 3-pin & 1x 6-pin female headers
      2. 6-pin header will mate to the 6-pin set on the bottom right of the control board (as mounted, looking at the component side of the board)
      3. 3-pin header will mate to the 3-pin set on top right of the driver board

4. Referring to the PCBs / PCB files, solder wires to the right positions (the sequence of pins is different on control and driver boards. This tripped up many a builder…).
5. Here's a pic (PFC driver board is shown; for non-PFC, the connector pins will be located on the other side of the driver board PCB):



v. Current sensing
   1. Need: 3x 12" wires, 1x 3-pin female header
   2. 3-pin header will mate to the current sensing pins on top right of the power board
   3. The other ends of the wires will go to the 6-pin header you used in the voltage sensing step above
   4. Referring to the PCBs / PCB files, solder wires to the right positions



vi. PWM signal

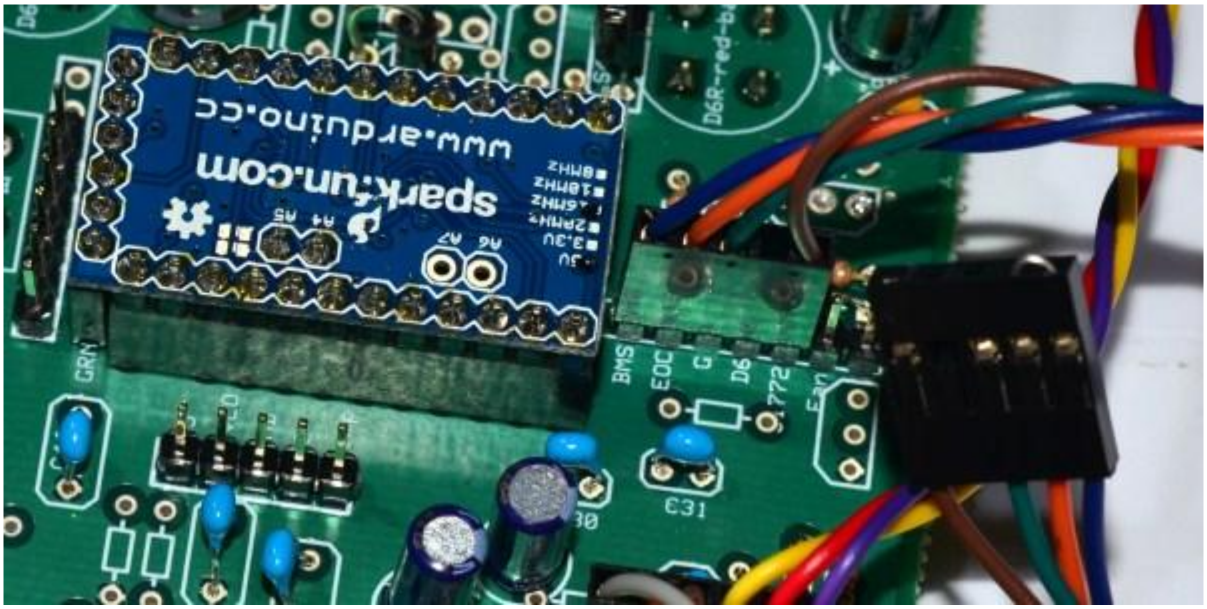
1. Need: 3x 6" wires, 2x 3-pin headers
2. One 3-pin header will mate to the leftmost 3 pins on the 6-pin connector on top left of the control board
3. Another 3-pin header will mate to the leftmost 3 pins on the 6-pin connector on top right of the driver board
4. Referring to the PCBs / PCB files, solder wires to the right positions
5. Pic here (PFC driver board is shown; for non-PFC, the connector pins will be located on the other side of the driver board PCB):

vii. BMS / J1772 dongle

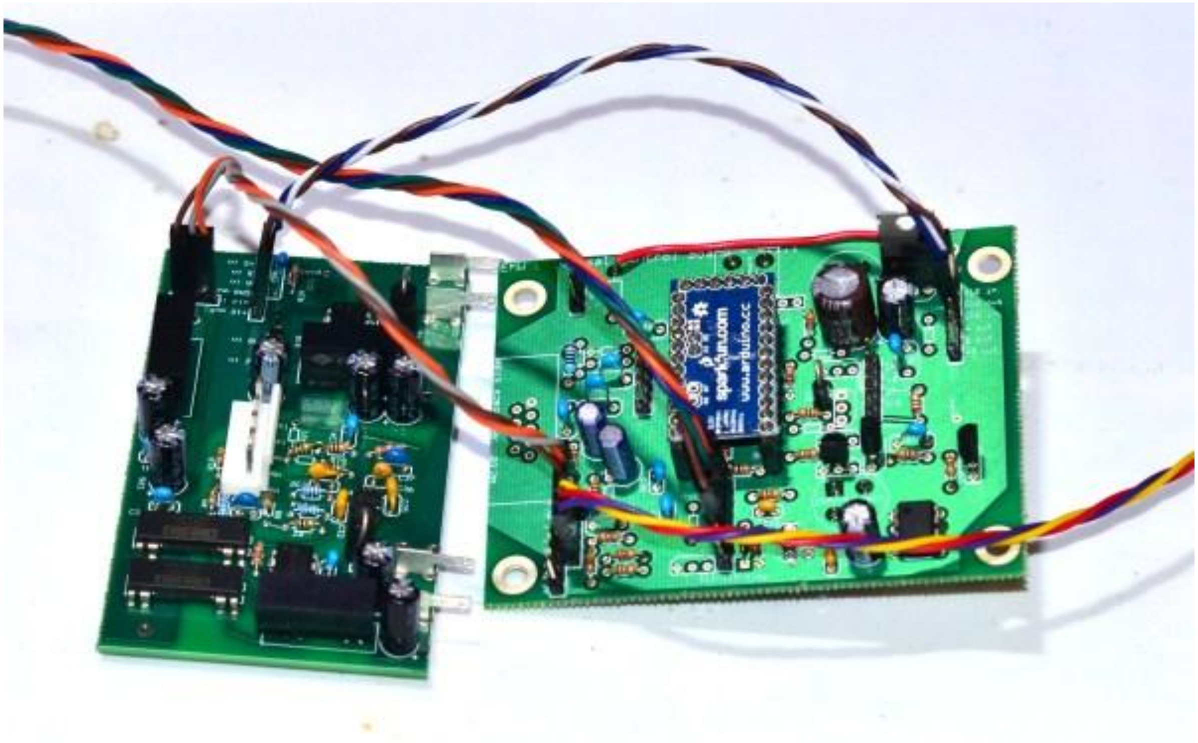
1. Need: 4x 12" wires, 2x 6-pin headers
2. One header will mate the leftmost pins of the 7-pin connector on the right side of the control board, right of the Arduino board
3. The other header will be brought outside of the charger (through one of the venting slots on the back side) and will be used to connect BMS, provide End of Charge signal (EOC), and connect J1772 pilot signal

a. Make a mini-jumper from one of the discarded component leads and insert into this header, shorting BMS and EOC wires. This will fake an operational BMS and will allow you to test the charger. Without this jumper, the charger will NOT start

4. Pic here:



And all connections together:

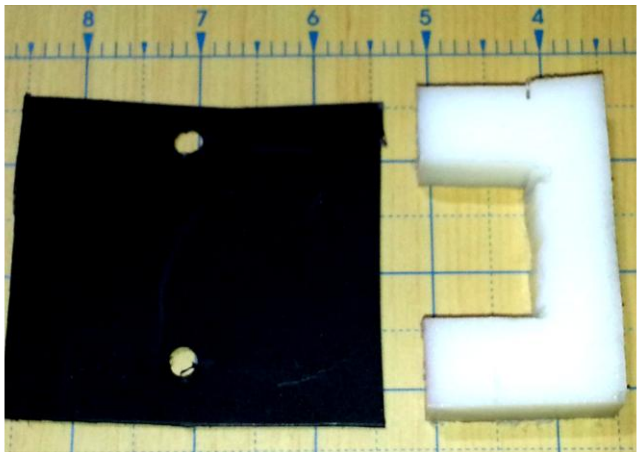


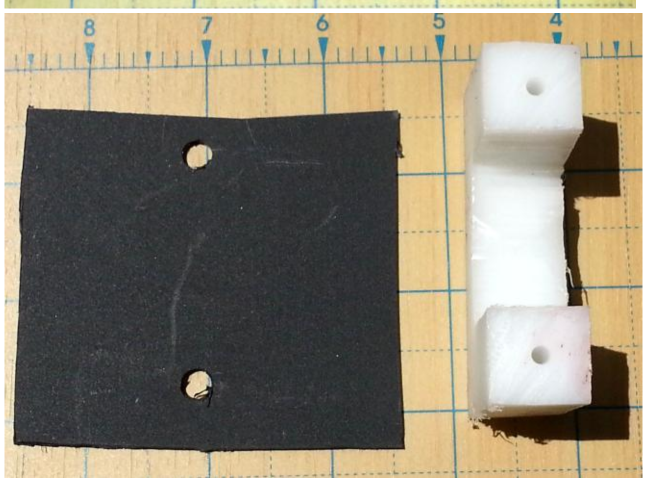
6. Connect the boards
    i. HV connector from power board to the driver board
    ii. 12V supply to driver board
    iii. Voltage sensing (driver to control board)
    iv. PWM (driver to control board)
    v. Insert the driver board into the power board
        1. You might need to bend the male pins slightly to fit
  vi. Current sensing (control to power board)

7. Prepare and Connect Inductor to Power Board
    1. Prepare mounting bracket
        i. Use ¾”-1” thick HDPE or similar material
            i. Cut a C-shape as shown below to allow the inductor core to come through
            ii. Drill 1/8” holes 1.5” deep or so from the center of the ends of the C-shape. This is where the mounting screws will go
            iii. Cut a square pad from some isolating material (1/8” rubber is great here)
            iv. Drill out holes for the screws in the isolating material (matching holes you just drilled in the C-shape)

v. Mount the inductor
    1. Use #10 sheet metal screws or similar, fender washers
    2. Fasten C-shape to the bottom of the enclosure, fitting a black isolating pad between the inductor and the enclosure to reduce chance of shorts

vi. Photos for reference:





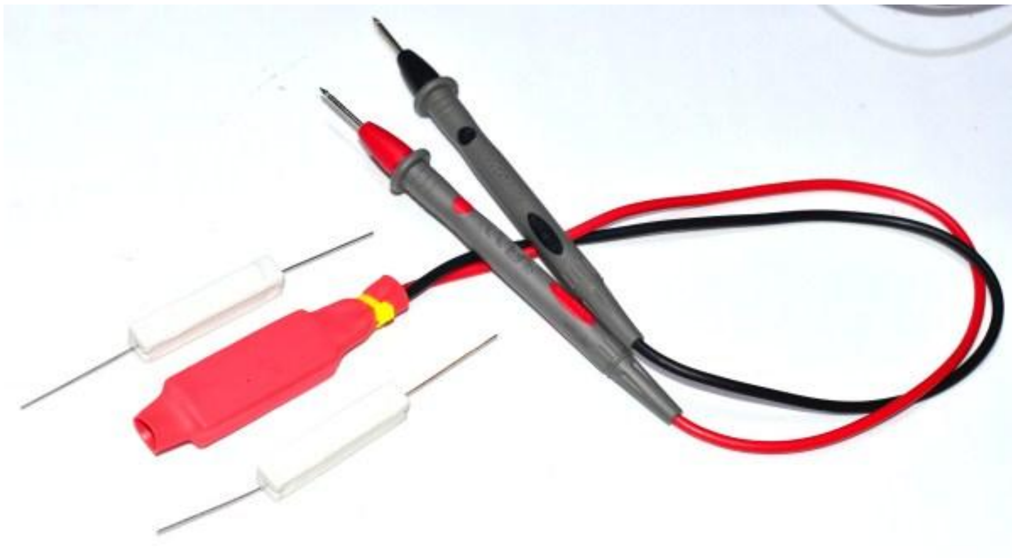
vii. Connect the inductor
    1. Cut 6" length of gauge 8/10 wire. Solder one end to the Ind2 pad on the power board, another end – to the closest lead of the inductor
    2. Cut 10" length of gauge 8/10 wire. Solder one end to the other lead of the inductor, fit eye crimp connector to another end and screw-mount to the top terminal of the right-side IGBT
        a. 1.5"-2" brass standoffs help clear the caps but not required

8. Assemble a HV input/output connector
    1. Use a supplied 4-6 position connector
    2. 3 positions to be used for input AC lines (1 for 240V ONLY, 1 for 120V ONLY, and 1 common for 120 and 240V)
    3. 2 positions – for output DC
    4. Remaining position (if any) can be used for Ground connection
    5. Using the connector as template, drill mounting holes in the back side of the enclosure, right above the venting slots (position connector horizontally)
    6. Connect the power wires (that you have previously threaded through the venting slots) to the corresponding positions of the connector
    7. Mark the outside of the box with wire designations. This is important – we had a few builders burning their chargers due to reverse polarity connection of the battery!
    8. Connect inrush current limiters
        i. 2x 50A limiters are normally supplied - use 1 on each 240V AC line
        ii. These parts heat up to 220C (450F) at max current
            a. Use caution in placing – nothing flammable should be close
            b. Leads are copper so they transmit heat very well – take that into account when mounting
            c. We suggest extending the leads with 4-6" of the AWG6-8 wire and then connecting one side to the input/output charger connector and the other side to your input AC lines
            d. You will need to prevent the limiters from moving around in your car – can use same silicone sealant to fix the leads to some metal parts (including the box of the charger. Sealant is normally good for 400F
        iii. Inrush limiters, while not strictly required, will help reduce stress on components (caps, input bridges) and prevent from tripping breakers (at home or at EVSEs)

9. Prep your charger for testing
    1. Ensure no stray material inside the charger. More than one charger was damaged by a forgotten washer shorting the PCB… Best way is to pick up the whole thing and tumble around
    2. Check for loose bolts / connections
    3. [OPTIONAL but could be useful] coat all bare high-voltage wires / connections / etc. with Corona Dope (something like http://www.mgchemicals.com/products/4226.html)
    4. If desired, do same for all remaining PCBs and connections

Part 8. Power Up & Test

Useful tool to help with multiple starts / stops of the power stage (something you will do a lot of below):



Simply a couple of multi-meter-type leads connected to a couple of 330R 10W ceramic resistors in series. Use this to dissipate the charger from the massive caps in the charger before accessing components if needed.

1. Upload the firmware
    1. Connect the supplied FTDI cable to your PC
    2. Make sure your PC recognizes the cable and assigns it a separate COM port
    3. Download from EMW site (current link http://74.208.162.121/cgi-bin/VMcharger_V9.pl?cc=gfw7iuef7) – find Software section
        i. Arduino-0022 package (zipped file, ~85MB)
        ii. Required libraries
        iii. Latest firmware
    4. Install Arduino-0022 package (simply select your installation folder and unzip the archive there)
    5. Copy libraries into the ‘libraries’ folder in your installation folder
    6. Create a separate folder somewhere on your drive and put the firmware there
    7. Make sure your downloaded code is configured correctly for your version of the charger kit
        i. Launch arduino.exe from your Arduino installation folder
        ii. IGNORE notification to update to the latest Arduino version. The firmware will NOT compile in later version of the Arduino IDE
        iii. File->Open Sketch, browse to the folder containing firmware, select .pde file

there, click Open

iv. File->Tools->Board, select 'Arduino Pro or Pro Mini (5V, 16MHz) w / ATmega328P' as the board type

v. Check the "#define …" statements in the first 2 pages of the code (marked as '--------- MAIN SWITCHES -----------')

   i. Comment out '#define PFC'
   ii. Uncomment '#define A7520_V' if your driver board uses a 7520 chip for battery voltage sensing
   iii. You should normally have '#define OUTC_SENSOR Allegro_100U'
   iv. If using a custom 100A output inductor, uncomment '#define MCC100A'
   v. Make sure '#define PFCdirect', '#define UV12', '#define NiXX', '#define buck_Ecore', '#define A7520_mV' are all COMMENTED OUT

8. Upload the code to the charger

   i. Depress the programming button on control board to disconnect LCD (to enable programming)
   ii. Connect the FTDI board to either…
      i. …the programming header on the control board (on top of Arduino board)
      ii. OR to the programming dongle you made earlier (a simple 12" 6-pin female-to-female 0.1" pitch harness)
   iii. In Arduino IDE, click Compile icon, confirm error-less compilation
   iv. Click Upload icon, confirm error-less upload

2. **Test the charger.** This is the testing sequence we use at EMW on all units. Generally, if any step fails, DO NOT proceed until you fix whatever is preventing it from passing. **USE EYE PROTECTION! WEAR SHOES WITH NON-CONDUCTIVE, SOLID SOLES! DO NOT TOUCH MORE THAN ONE BARE CONTACT AT ANY TIME!** The following sequences assume unmodified firmware (from the official EMW distribution)

   1. Prep the props
      a. 2x regular household lamps connected in series – this will be a dummy test load for the charger
      b. ~5kW, 240VAC rated load (we use standard water heater elements like http://www.amazon.com/Reliance-9000092-045-Screw-Flange-Element/dp/B000DZHAQO/ref=sr_1_1?ie=UTF8&qid=1333343402&sr=8-1)
   2. Test logic circuits
      a. Make sure programming button is pressed in (small button to the right of the LCD, below the green (or black) button
      b. Connect a separate power cord to the AC adapter, plug in to 110VAC (DO NOT CONNECT AC TO THE CHARGER – ONLY TO THE AC ADAPTER)
      c. Fans should turn on
      d. The screen should go live
      e. If you see the 5-second count-down on the LCD, press any control button

f. On the screen, setup the charger
    i. Select LiFePo4
    ii. Select 3.5 V CV cutoff ('350' setting in the CV menu)
    iii. Select 30 cells
    iv. The charger should ask you to short the output
        1. If it does not and instead goes right through this step to the confirmation screen, something is wrong. Most likely because of the incorrect setting of the '#define A7520' switch in the firmware. See above for details on how the switches should be set
            a. You need to re-calibrate your unit whenever you change any major switches in the code affecting voltage sense (#define A7520_V)
        2. If it does ask to short the output
            a. MEASURE the output before shorting
            b. If it reads more than a few volts, connect your dummy lamp load to the output. Measure again in a few seconds
            c. If it still reads more than a few volts, something is wrong. Time to debug
            d. Assuming reading is zero, short the output, press the green button
    v. The charger should now ask to connect the battery to calibrate itself
        1. In this first calibration, use 30-60V battery when requested to connect battery, ideally through a DC circuit breaker
            a. Watch polarity!
            b. There will be a spark on connection, esp if your battery is >150V. You may want to use one of the supplied inrush limiters for this step (place in line with the battery before connecting)
        2. If the charger detected the battery (within 5-10 seconds), it will show the voltage it read on the screen. Edit the voltage reading to match your measured battery voltage and confirm
        3. If the LCD does not change at all when you connect the battery, something is wrong in the sensing circuit. Time to debug

g. You might get a 'battery not connected' message. Please follow instructions on screen to ignore. This is normally caused by the battery voltage differing too much from what the charger expects (based on the cell count you specified in the setup step)

h. After calibrations are done, set the charger to max output current of 10A and let it go into the charging mode (with the battery connected), watch the duty display go up from zero to 95-97%.

i. The output voltage reading on LCD should be very close to your battery voltage. The current reading should be zero (or 1A max)

j. Check the heatsink temperature readout. Should be close to ambient

k. Disconnect AC from the adapter

3. **Test power circuits– FROM THIS POINT ON, YOU *HAVE* TO USE PROTECTIVE GLASSES**. One wrong polarity elcap soldered in reverse can mean a very violent explosion with boiling electrolyte shooting in all directions. Not fun…

   a. Limited test of Power stage only – this test is VERY important and can prevent expensive damage of the charger by testing at low voltage before applying full PFC voltage to the components (~400V!)
      i. Connect your 30-60V battery to 240V AC inputs of the charger
         1. Be ready to quickly disconnect if you see / hear / feel anything out of the ordinary (e.g. hissing / crackling sounds etc)
         2. Wait for 10-15 seconds
      ii. Measure input voltage on the IGBT (bottom and middle terminals). Should be very close to your input voltage
      iii. Measure input current if possible (clamp meter is handy here). Should be very close to zero
      iv. Connect your dummy lamp load to the charger. Measure output voltage. It should be close to zero

   b. Limited test of Power + Logic stages
      i. Connect AC adapter to the main power lines in the charger
         1. Solder AC adapter's AC input to the small AC wires that you earlier had attached to the doubler board (that should by now be firmly attached to the cap block with the sealant)
      ii. Connect 110V to the 240V input lines of the charger – ideally through a protected power strip
         1. Be ready to quickly disconnect if you see / hear / feel anything out of the ordinary (e.g. hissing / crackling sounds etc)
      iii. Let the charger time out through the calibration routine (5-second timeout)
      iv. Set the power level
         1. If you get a 'reverse polarity' warning on the LCD screen, just override as directed on the screen
         2. If the charger stops to ask for input power, set input and output power to 10A
         3. If the charger does not ask for anything and just goes into the second timeout (a 10-second power setting timeout), press any control button and navigate to 'Change Power' menu item, then set input / output power to 10A
         4. During the power setting routine, note the stated input voltage. It should read close to 110V

v. Measure the output of the charger. Should be close to zero (remember that you still have the lamps connected to the output)
vi. Start the charger
    1. Using the control buttons, navigate through the menu to start the charger
    2. Confirm run
vii. This time, the duty should ramp up from zero to ~70-75% (160V rectified AC being reduced to ~100V) before charger should go into the CV stage
viii. In the CV stage, watch the duty to go from zero to 70-75% again and stabilize there. Lamp should be reasonably brightly lit at this point
ix. Measure voltage on the lamp, confirm that it is close to the voltage displayed on the screen (up to 10% difference is ok – you will re-calibrate the circuit later)
x. Let the charger run for 10 min. Check the heatsink temperature readout on the screen. Should be close to ambient
xi. Disconnect AC input & lamps

c. Full test of the Charger on resistive load
    i. Connect the 5kW load to the output
    ii. Connect 240VAC to the input (make sure you use 240V input lines – connecting to 120V lines will damage the charger)
    iii. Repeat the steps from the previous limited test, confirm normal operation (exact duty cycle numbers will be different)
    iv. Power cycle AC
    v. Interrupt the first timeout, set the charger to 70 cells
    vi. Interrupt the second timeout, set max output current at 25A
    vii. Watch the charger ramp from zero to ~70% duty, go into CV mode, then ramp to 70% again and stabilize there
    viii. Observe the heatsink temperature for 5 min. Should not exceed 20 degrees above ambient
    ix. Disconnect AC input and 5kW load

d. Full test of the Charger on battery load
    i. Connect lamps to output. Confirm zero volts
    ii. Connect 240VAC to input
    iii. Interrupt the first 5-second timeout, set the right number of cells for your pack
    iv. Press through the 'short output' step (as you have the lamps connected which for calibration purposes is the same as shorting the output)
    v. When asked, connect your traction battery to the output of the charger (OBSERVE POLARITY). We normally connect 60VDC test battery for the

first battery load test – allows us to further limit potential problems. You may connect your full battery pack

1. If you did not use the output diode for your build, you will get a spark on the first connection. This is normal
2. If you battery voltage is above 140-160V, we recommend using a power resistor (10-100R) for the first connection to pre-charge the capacitors

vi. Follow the instructions on screen to calibrate the charger again for your actual battery voltage

vii. Low-power run

1. Set power output to 10A
2. Watch the charger ramp from zero to some duty cycle corresponding to your battery voltage (generally will be ~110% * <your battery voltage> / 330VDC)
3. Let run for 5-10 min, watching temperature. Should be minimal rise
4. Interrupt the charger by pressing and holding the red button. Press again to exit

viii. Medium-power run

1. Press and hold the green button until the charger goes back to the power setting dialog
2. Set to output current corresponding to 6kW output (e.g., if your battery voltage is 200V, set the charger output to 30A)
3. Start the charger
4. Let run for 5-10 min
    a. Watch temperature. Should see heatsink temp rise of 10-15C
    b. Confirm output voltage rise (rate of rise depends on the SOC and capacity of your pack)
5. Interrupt the charger by pressing and holding the red button. Press again to exit

ix. Full-power run

1. Press and hold the green button until the charger goes back to the power setting dialog
2. Set to output current corresponding to 12kW output (e.g., if your battery voltage is 200V, set the charger output to 60A)
    a. The charger might limit the current to lower value depending on your battery voltage (to satisfy the limits of 12kW and 70A output)
3. Start the charger
4. Let run for 5-10 min
    a. Watch temperature. Should see heatsink temp rise of 15-

20C

b. There should not be any unusual loud noises coming out of the unit. If you hear screams, loud buzz, etc – time to debug.

5. If possible, let the charger run through full charging cycle
   a. Watch temperature & voltage to prevent damage to charger and/or battery
   b. There are automatic protections built into the charger against overtemp but it always a good idea to watch things closely on the first few runs
   c. Confirm CC to CV switch at the right CV voltage
   d. Confirm CV termination at the right output current
      i. ~5% of the capacity, can be changed via firmware edits
      ii. ~1 min lag in termination is ok
6. Disconnect AC
7. Disconnect battery

Congratulations! You now have the best charger money can buy.

And you built it yourself!

Go Electric!

Yours truly,

Valery and the rest of the EMW Power Electronics Crew

MODS

This section will list a number of mods / additions to the charger that could be done. They could be beneficial in your specific circumstances. Each subsection below starts with a short description of the mod and why you would do it and then specifies how.

If you have done any interesting mods to your charger build, send us some description and we will add it here!

See a separate file in
http://www.emotorwerks.com/VMcharger_V12P/ for latest