EMW SmartCharge-12000 12kW EV Charger

PFC V12 Assembly Instructions

June 22, 2013

These instructions are synced to the assembly videos at
http://www.youtube.com/vmiftakhov (search for '12kW')

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 people have successfully completed this project without hurting themselves (at least to our knowledge), so you can, too.

Part 0. Before you start

1. **Required tools:**
    1. 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
    2. Screwdrivers
    3. Wire strippers
    4. Small snips for wire / lead cutting
    5. Small pliers
    6. Drill & drillbits
    7. Threading taps: 10-32 or 10-24, 6-32
        a. See http://www.wikihow.com/Use-a-Tap for some tips
    8. 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
    9. Clear Protective Goggles
    10. 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
    11. 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**
    1. 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
    2. Small soldering vise to hold pcbs while you solder
    3. 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)

1. Bolts, nuts, washers: #10 and #8, various lengths
2. 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
3. Power wiring – ideally 3-4 colors of AWG6-8 isolated multi-threaded wire (2' of each color) plus 1' of bare copper AWG10 wire

4. Assembly Tips

1. Sequence of assembly is often very important: some parts may not be able to fit after others have been soldered
2. 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
3. If you can't find some part ID on the board / PCB file, do Ctrl-F from within ExpressPCB to find the part
4. 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

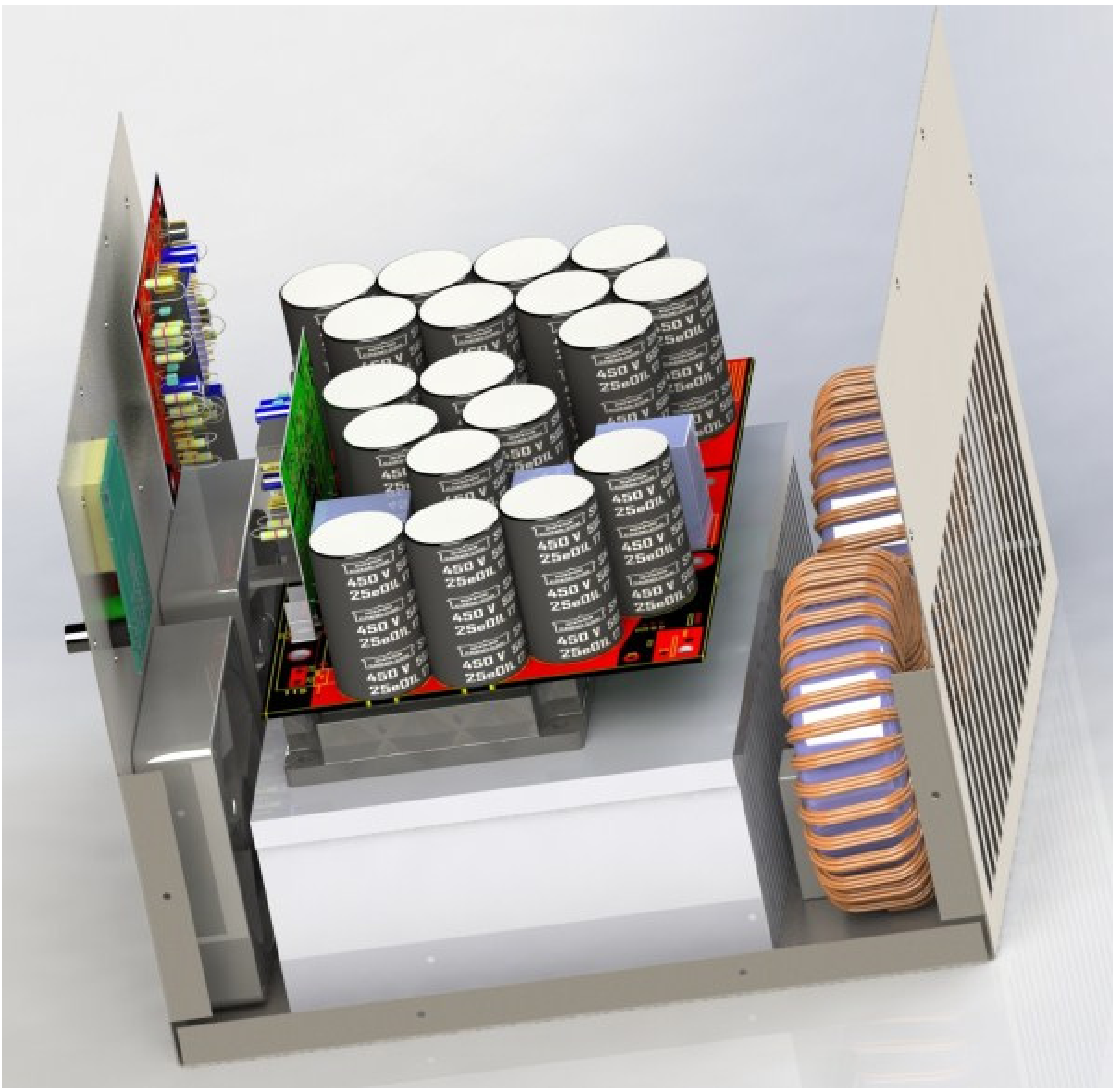
5. Education

1. Wouldn't it be nice to actually understand what you are building?
2. 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
3. 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.

Note that this is a CAD rendering and NOT an actual photo of the unit.

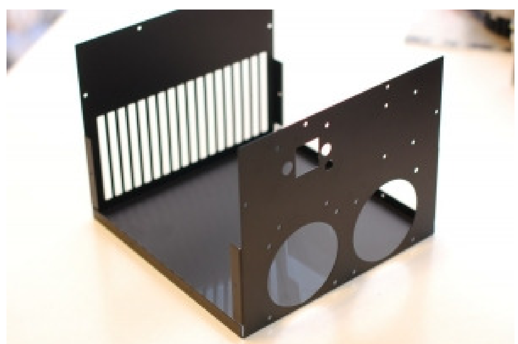


Part 1. Kit Contents

Please refer to the Bill Of Materials file on our site for more details on component lists, part numbers, etc: <BOM LINK>. 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.

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
   b. Driver Board
   c. Input Bridge Board
   d. Power Board
   e. FTDI board (used to program your Arduino, supplied with a USB cable)
3. 2 EMW High-Power Inductors
   a. Standard order has 2 Identical inductors rated for 70A DC current
   b. If you have selected a high-current option at the order time, you may also be supplied a 100A rated output inductor (the PFC inductor will still have a standard 70A rating)
4. Integrated Circuit (IC) Parts:
   a. Arduino Pro Mini 5V / 16MHz board
   b. 2 IGBT Driver chips (A3120)
   c. Comparator chip (LM211P)
   d. 2 ISO124 Isolating Amplifiers (could be substituted by other optos – see in Driver Board instructions below)
   e. Current Sensor (100A unidirectional Allegro Hall sensor)
   f. PFC chip (IR1153)
   g. 3 DC-DC convertors used to convert 12V to 15V
5. Connectors
   a. 2-3 sets of 40 male breakable pins (0.1” pitch)
   b. Assorted female .1 inch headers
   c. 5-pin white female connector + pins; 5-pin male connector. Can be omitted depending on type of driver board – see below in Driver Board instructions
   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 2 for Input AC power and 2 for Output DC)
6. Resistors
   a. Most 1/8 W
   b. 3W 10R gate drive resistors
   c. 2-3W 50-100k safety resistors
7. Capacitors
   a. electrolytic (labels directly on the caps)
   b. 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)
8. Color 1.44” diagonal 128x128 LCD programmed for the serial communication
9. 2 In-rush limiters: 50 amps max current (one goes to each AC input)

10. 3 blue film caps used on Power Board
11. Semiconductors:
    a. small N2222 transistors
    b. 2 diodes
    c. 5V linear regulator
    d. 2 Diode Bridges used on the bridge board
    e. 2 IGBTs: 150-200 amps, 600V (some IGBTs might have slightly different positions of tabs, which may need to be bent slightly to match the Power Board layout)
12. 17 Power Capacitors for The power Board (generally shipped wrapped together with power board)
13. 12V regulated AC supply rated for 2-5 amps (to power the fans and all circuitry in kit)
14. Optional Enclosure and Fans
    a. Option 1: standard enclosure – a 10x10x8" steel box. Needs to be drilled / cut to fit fans, LCD, heatsinks, etc.
    b. Option 2 (Recommended for easier and faster assembly): Fully machined enclosure matching your kit. Benefits:
        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)

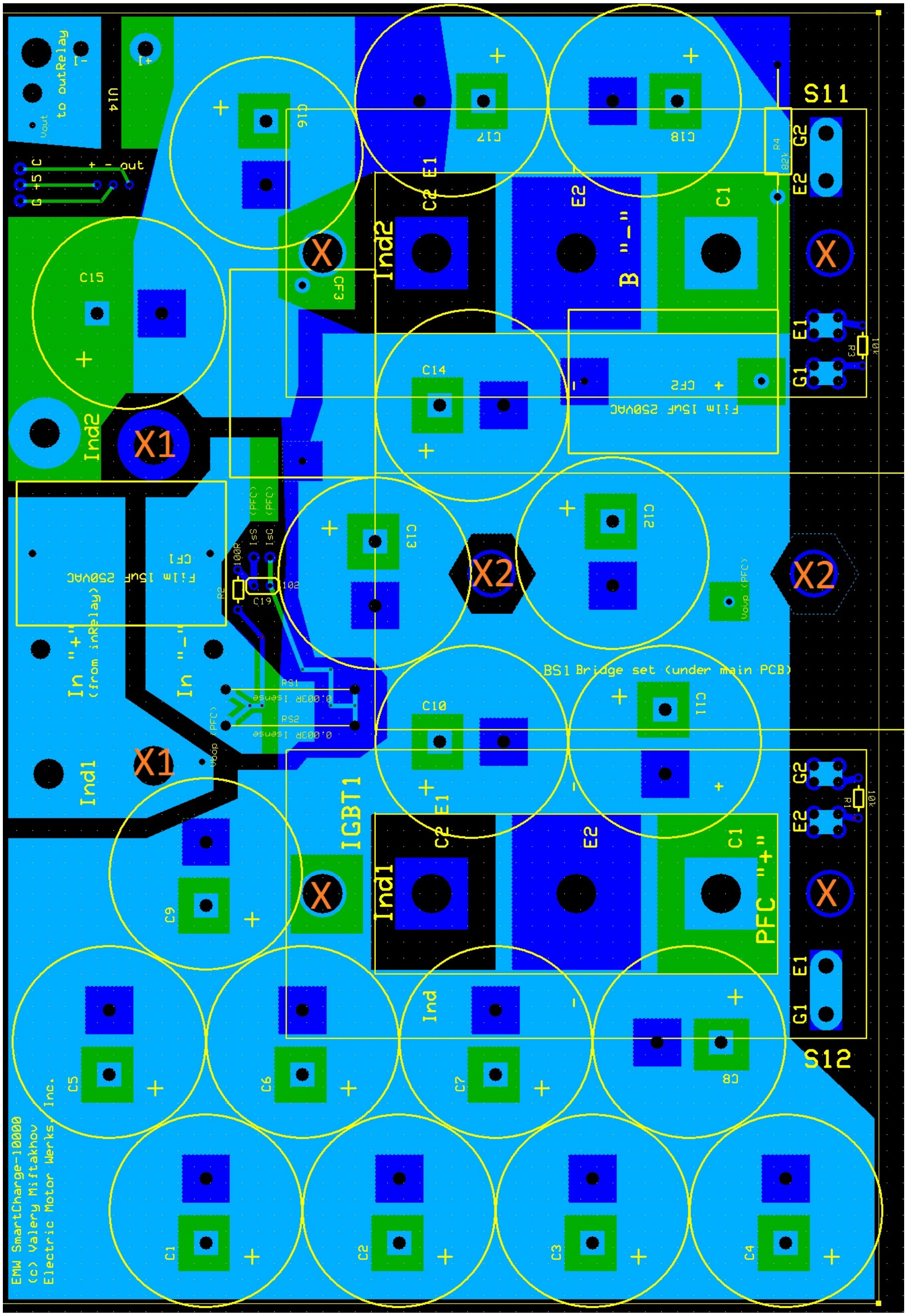
   a. Take out the Power Board and place it on top of the Heatsink (orient heatsink towards yourself with the cut lip to your right – the heatsink is cut to size from the standard 10" profile by cutting off a couple of fins on one side – that side should be on your right)
      i. Place the power board roughly in the middle with about a quarter inch space between edge of the heatsink to edge of power board from your side
      ii. Make sure that the IGBT mounting holes fall in between the fins of the heatsink (IGBTs are located on the PCB in such a way that if you match up one of the holes between the fins they all align themselves in between the fins)

   b. Use a permanent marker to mark up the hole positions
      i. 4 Power board IGBT screw holes (marked X on the picture below)
      ii. 2 Input Diode bridge holes that need to be marked up on the heatsink
      iii. 1 center hole for the thermistor (partially drilled through).

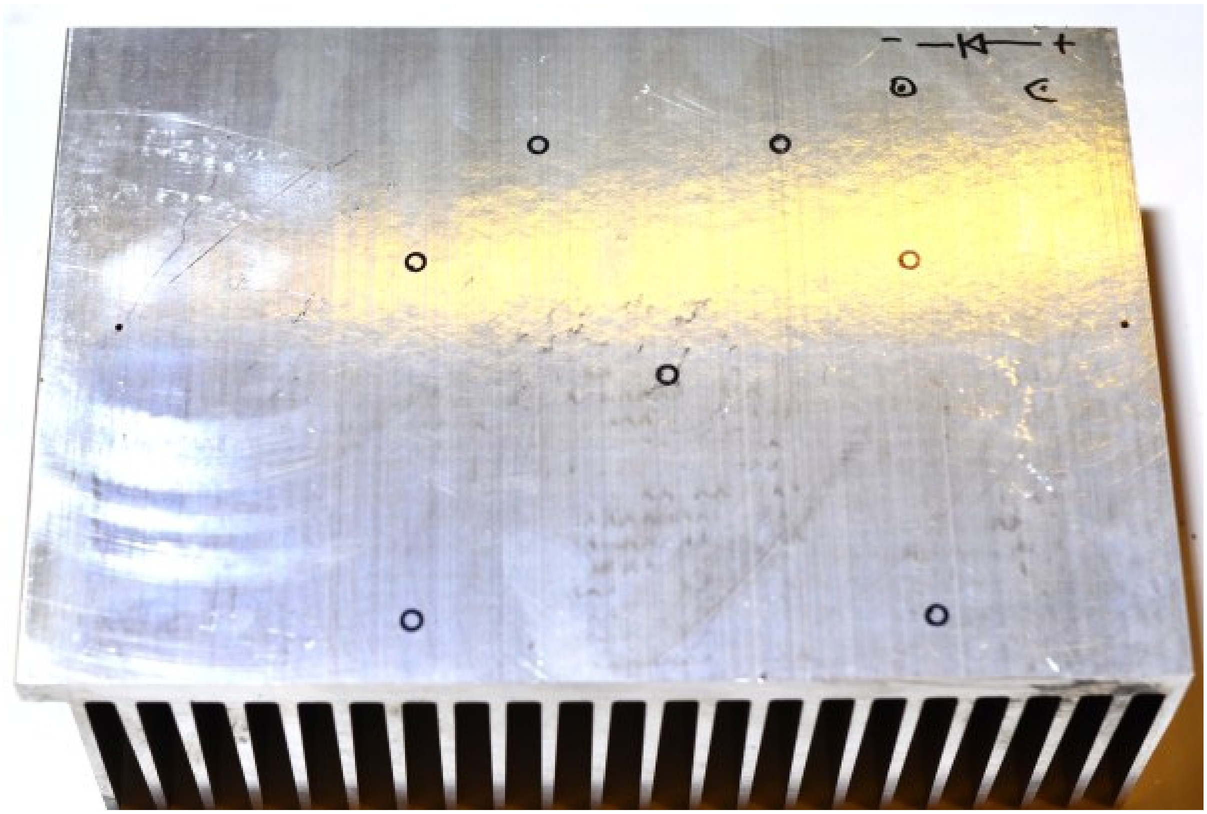
   c. If using an output diode for output inrush protection (not required for PFC but helpful if you plan to connect and disconnect your BATTERY often)
      i. if you decide not to use the diode, you will need to be more careful on the first connection of the battery to avoid inrush currents (use the supplied inrush resistors for this first connection – more details given below in 'test' section)
      ii. Place a diode on the top right corner of the sink
      iii. Align mounting holes horizontally, with an open hole facing right
      iv. Center of the open hole should be ~20mm from the top and 20mm from the right edge of the sink.
      v. Mark up the holes
      vi. 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

   d. Another option for the output inrush protection is to use an output relay. No special steps are required for this option at this stage. Details of the connection will be described later in this document. Some considerations behind the choice of relay vs. diode
      i. Relay = lower loss (~60W) but more complex connections / control (need precharge parts and Arduino relay control)
      ii. Relay = much more space required
      iii. As a result, especially for high-voltage applications, we recommend using the diode (but again, only if you really need inrush protection – NOT needed if your battery stays connected to the charger all the time)

   e. Finally, 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).


S11
S12
X1
X2
Ind1
Ind2
IGBT1
PFC "+"
B "-"
In "+"
In "-"
(from inRelay)
Film 15uF 250VAC
BS1 Bridge set (under main PCB)
EMW SmartCharge-10000
(c) Valery Miftakhov
Electric Motor Werks, Inc.

f. Here is how it should look at this point:



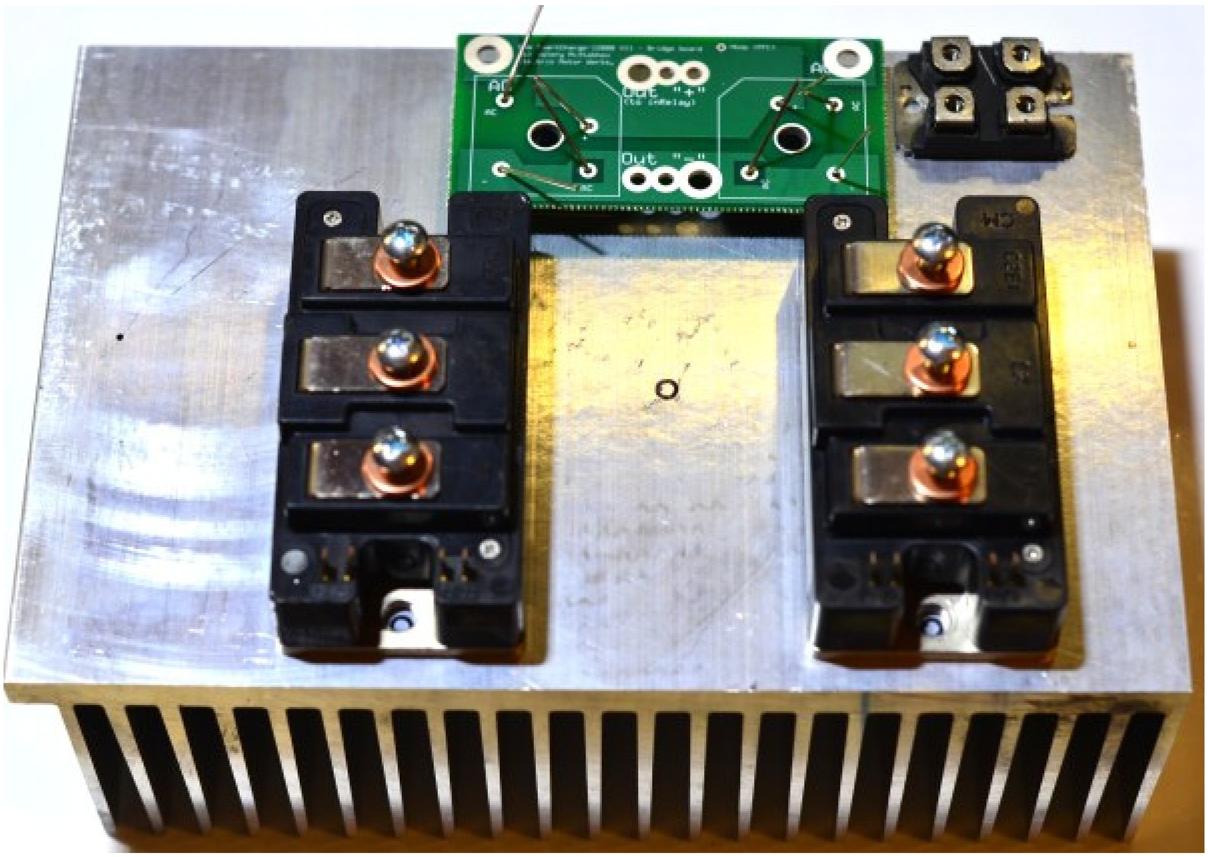
g. Next, drill and tap

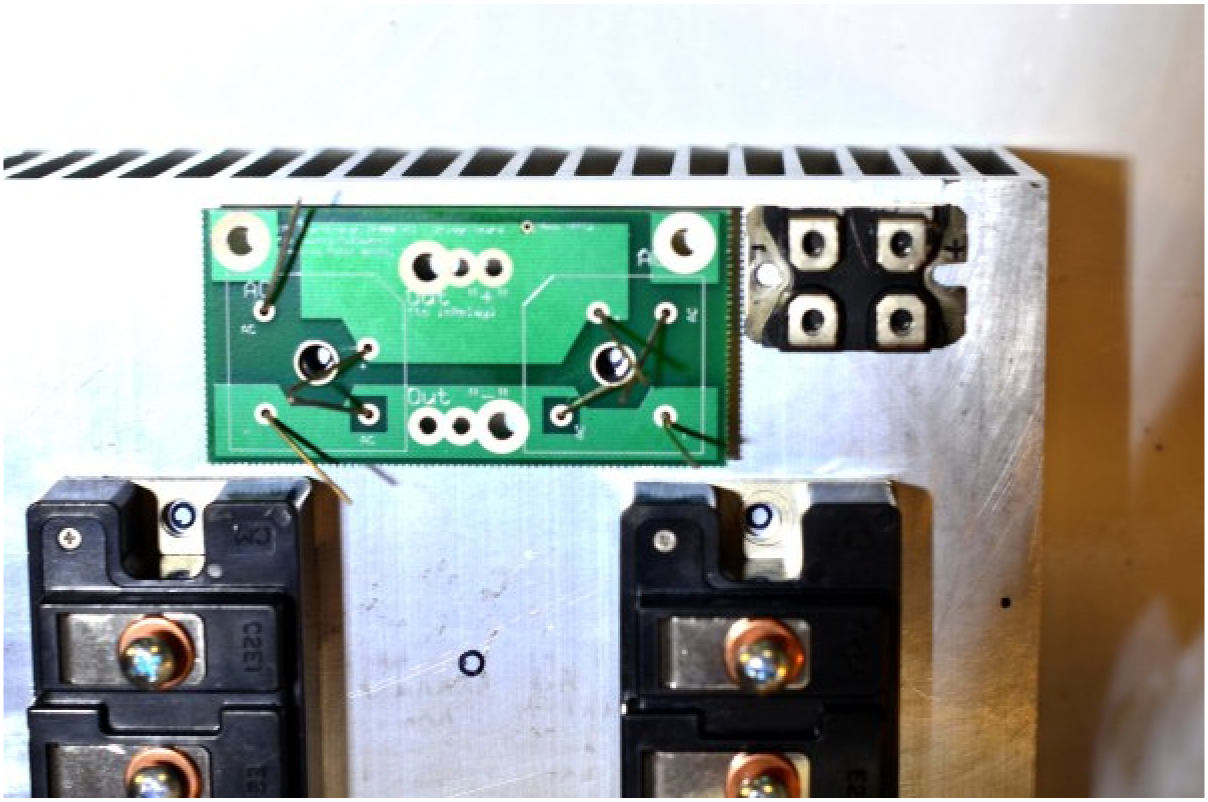
i. The first six holes (IGBT & diode bridges): drill and tap for 10/32 thread
ii. Drill the thermistor center hole ~3/8ths of an inch into heatsink (not all the way through) with same drillbit
iii. The optional diode holes: drill and tap for 6-32 thread
iv. Final 2 heatsink mounting holes need to be drilled with a slightly larger drill bit: has to allow the 10/32 or 10/24 bolts to pass through

h. Mount the output diode (if used)
  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 6” length of the bare copper wire gauge 10/12.
    ii. Make a ¾”-long oval loop on one end, solder the end to the wire closing the loop. This will be mounted to the output diode (using the double diode assembly, with 2 diodes connected in parallel)
    iii. Screw-mount the loop onto the anode terminal[s]

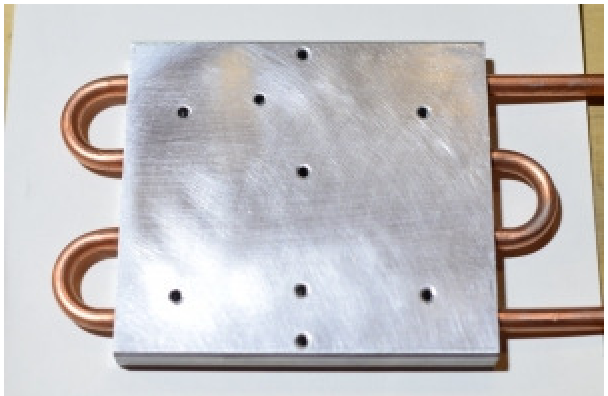
iv. Test-fit the assembly under the power board, threading the copper wire through one of the B “+” (or OUT “+”) pads in the upper right corner of the power board

i. Here’s how the components will be placed on the sink (do NOT mount anything except the output diode yet!)



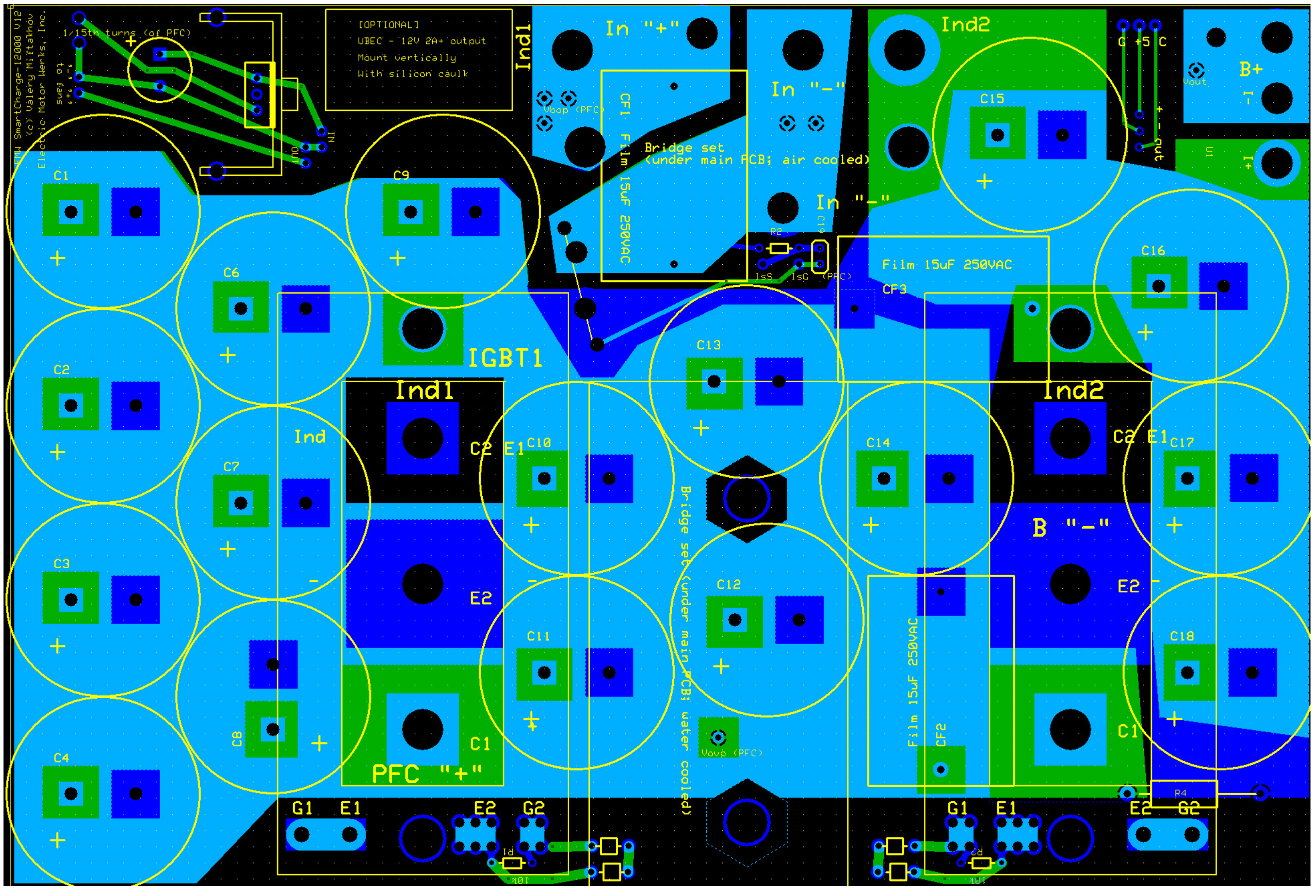


2. **LIQUID COOLED** – prepare the plate
    a. Orient 5.5”-wide, 6”-long cold plate you have received from EMW with the outlets to one of the sides
    b. Drill 6 holes using the pattern from the power board PCB. For the liquid cooled unit, use the holes marked ‘X’ and ‘X2’ on the picture above. Tap with 10-32 or 10-24 thread. Align the holes so that they do not interfere with copper tubing of the plate. These will be used to mount IGBTs and input diode bridges
    c. Drill 2 holes on the horizontal centerline of the plate, close to the right and left sides. Tap with 10-32 or 10-24 thread. These will be used to mount the plate to the bottom lid of the charger enclosure
    d. Drill the thermistor hole ~3/8” into heatsink (not all the way through) with the same drill bit
    e. Here’s how it should look when you are done:



3. Steps common to both air- or liquid-cooled units
    a. 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)
    b. Using silicon caulk, secure the thermistor in the thermistor hole

Part 3. Power Board

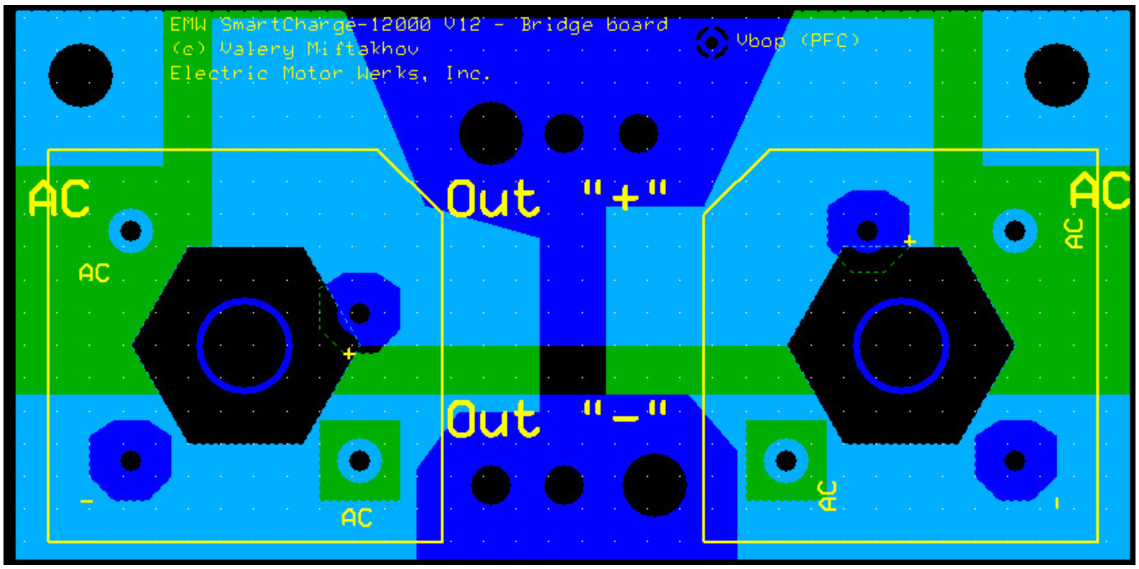


a. Place all smaller components
   i. Resistors:
      1. 10k (R1, R3) – bottom of the board
      2. 90-100R (R2) – top center
      3. 200k (R4) – bottom right
   ii. Small 103 (10nF) capacitor (C19) – top center
   iii. 100A Hall sensor: U1 - top right
   iv. 3-pin male header 0.1" pitch - next to the sensor
   v. Hold components with a carton, turn over the board, solder

4. Place first set of large components
   i. Using a short piece of gauge 10 / 12 base copper wire (~2"), connect the 'In "-"' area of the PCB (under CF1) and the main negative plane (the one running through E2 terminals of each IGBT): bend wire to fit in provided pads, solder using a 100W+ soldering gun.
   ii. 3 Blue rectangular film caps(15uF 250VAC caps supplied): CF1-CF3
   iii. Hold components, turn over the board, solder (helps to slightly bend the caps' leads so they don't fall out when you turning the board over)
5. Solder large bulk Elcaps 560-680uF 400-450V

   a. Make sure all capacitors correctly match the neg and pos leads to neg and pos PCB holes on board (positive PCB holes will be marked with a square and +)
   b. All the caps should be oriented the same way except one of the caps on the bottom left of the board
6. Solder the 5-conductor driver board harness
   a. Use a 5-pin connector (provided white housing + 5 pins or a 0.1” female pin header depending on the type of your driver board)
   b. Vbop pad on power board (on the ‘In “+”’ plane – top middle of the board) to the position marked ‘Vbop’ on the driver board
   c. Vovp pad on power board (on the main bottom plane – bottom middle of the board) to the position marked ‘Vovp’ on the driver board
   d. Vout pad on power board (on the ‘B+’ plane – top right of the board) to the position marked ‘Vout’ on the driver board
      i. If using an output diode or relay, this wire will have to be connected to the cathode of the diode or the output of the relay. Otherwise the battery voltage readings will be incorrect, potentially resulting in damage to the charger and your battery!
   e. IsS and IsG pads on power board (next to the blue CF1 cap) to the corresponding positions on the driver board
7. Solder small AC wires (to be later used to connect to AC side of the 12V adapter)
   a. 2x 6” AWG22 wire or similar
   b. One wire goes to the In ‘-‘ plane. Best location is the pad on top of the bare copper link you just placed
   c. Another wire goes to the In ‘+’ plane. Best location is the Vbop pad you just used for Vbop wire
   d. Twist wires
8. Prepare the IGBTs
   i. Solder together the E2 and G2 pins of the right-side IGBT
   ii. Solder together the E1 and G1 pins of the left-side IGBT
   iii. Fit the 2 supplied female spade connectors on the E1 and G1 pins of the right-side IGBT
   iv. Fit the 2 supplied female spade connectors on the E2 and G2 pins of the left-side IGBT
   v. Check for clearance between IGBT and the power board’s traces / component leads. If low clearance, use the copper washers / standoffs to raise the board off IGBT terminals by 1/16-1/8”. Normally, the required washers will be suppled by EMW depending on the type of the IGBT shipped
9. Place the pre-assembled power board on top of the IGBTs. Ensure the pins of the spade connector align with the correct pads on the power board
10. Secure the board to IGBTs’ top terminals only (marked ‘C2 E1’ on the board); solder the spade connector pins from the top
11. Apply silicon adhesive between all caps to protect against vibration
12. Here’s how the board should look (approximately) after you are done:

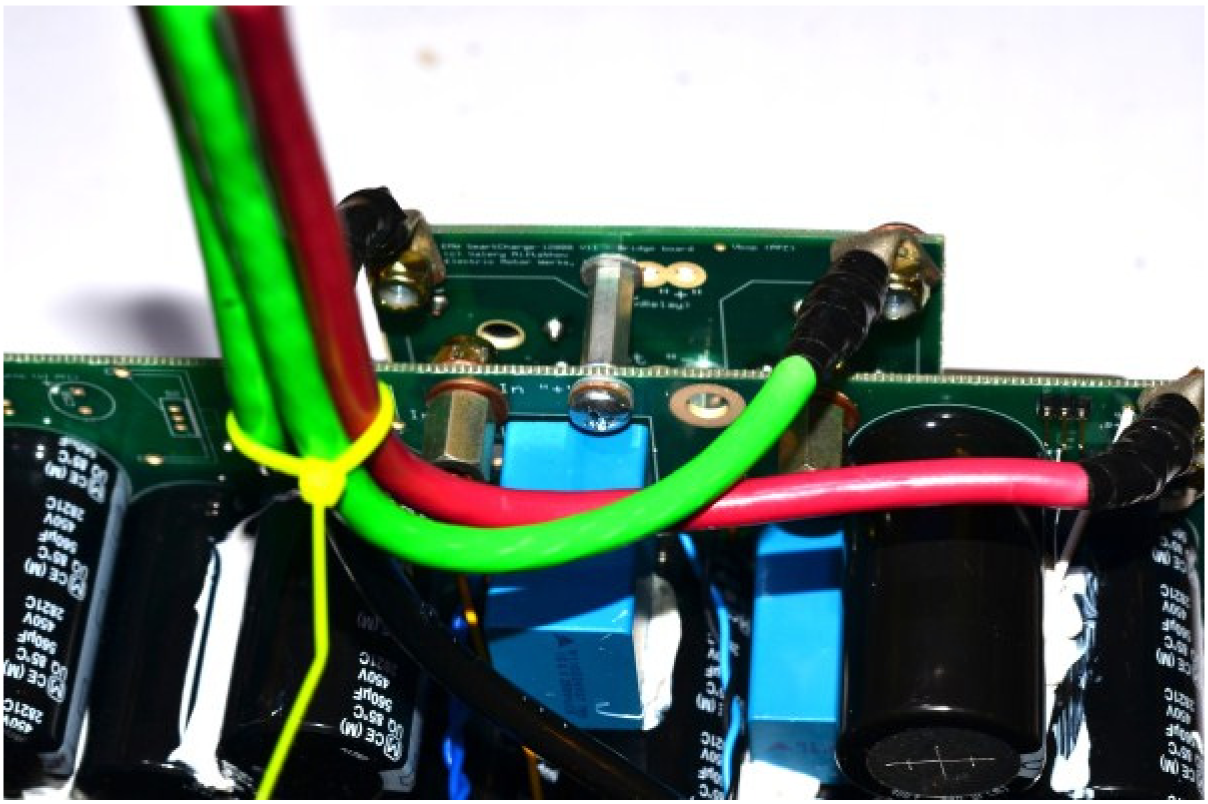
Part 4. Bridge Board



1. Mount 2 diode bridges – insert from the bottom of the board, solder from the top
2. Reinforce connections between negative bridge pins
   i. for this, it's better to keep leads sticking out ~1/4-3/8" after soldering)
   ii. cut a ~3.5" length of a 10-gauge bare copper wire and solder ends to the bridges' negative pins
   iii. make sure to not interfere with the Out "-" pads in the bottom middle of the board
2. Wire inputs and outputs
   i. Cut 10" length of gauge 6-8 wires. These will be brought outside of the box and connected to a terminal strip to AC pads of the board.
   ii. Solder these to the AC pads of the bridge board. Make sure the soldered leads will not touch the sink / plate once the board is mounted.
   iii. Solder 1.5" length of gauge 10 bare copper wire to "Out +" and "Out –" - positioned straight up.
   iv. To make sure you use the right pads, align the bridge board under the power board and note which pads fall under In "+" and In "-" entry pads on the power board
   v. Make sure the soldered leads will not touch the sink / plate once the board is mounted.
3. Mount the board to the sink
   a. Apply thermal paste to the diode bridges
   b. ensure AC pads end up on the outside of the sink
   c. Fix the board to sink using ¾" 10-24 or 10-32 machine screws
   d. Apply silicon adhesive between the board and sink/plate to secure against vibrations

Mount power board on top of the bridge-sink assembly

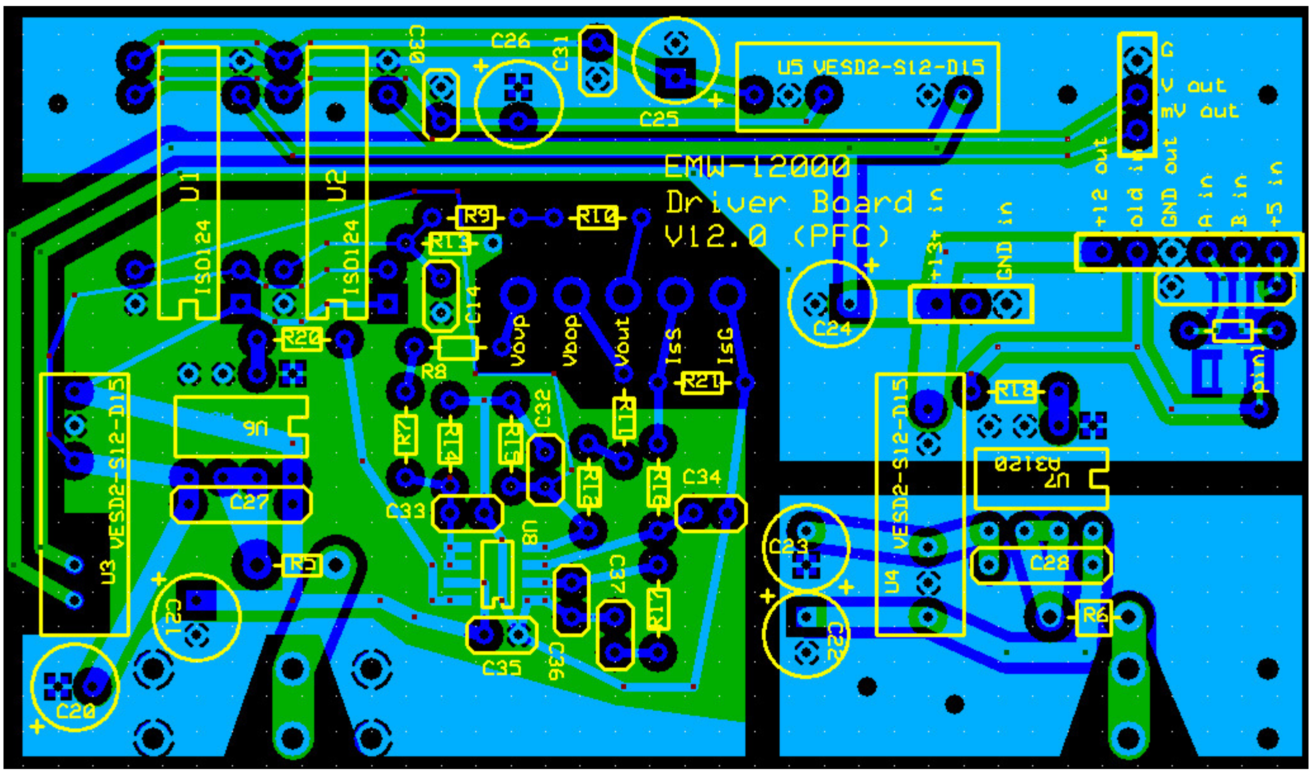
4. Apply thermal paste to the IGBTs
5. Lower the assembled power board onto the sink, aligning with the bare copper wires from the bridge board so that they pass through the correct pads on the power board
6. Solder the bare copper wires to the power board
7. Here’s how the connection should look (approximately) after you’re done:



Part 5. Driver Board

Notes:

1. There will be a number of unoccupied positions on the driver board after you are done – the “missing” parts are required only for the multi-stage systems and are not needed in this single-unit version.
2. There are several different types of Driver boards. Most likely you will get this one:

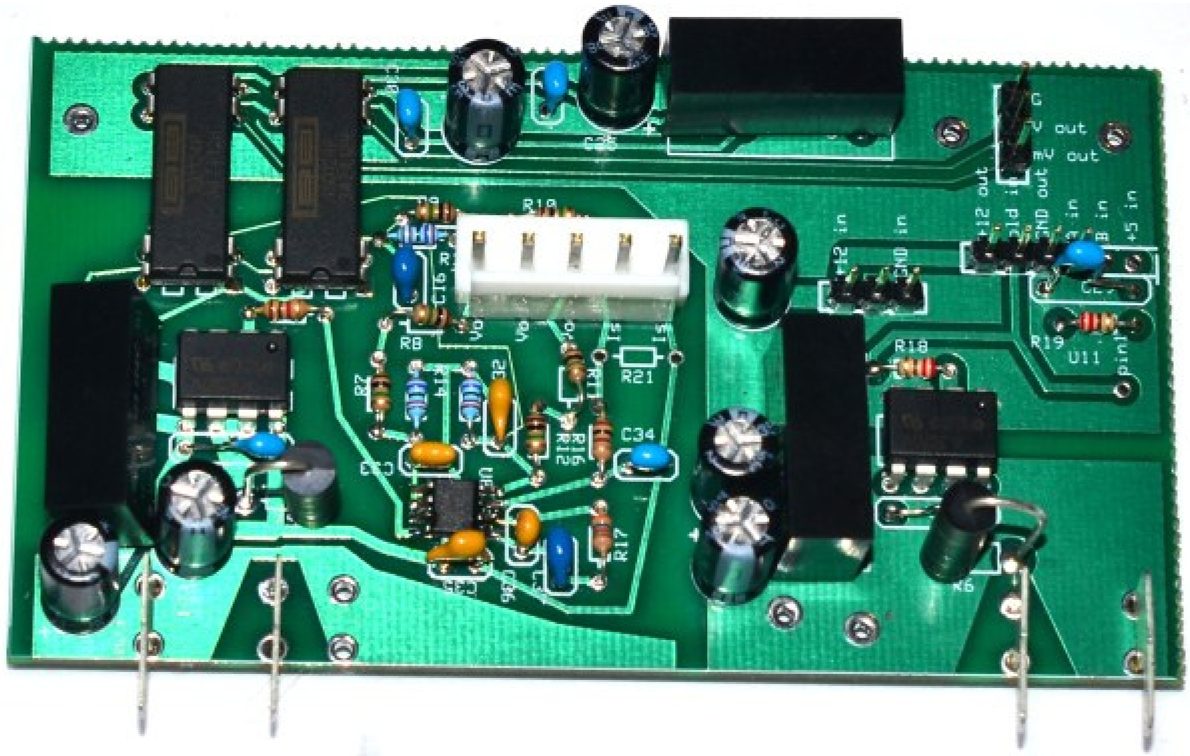


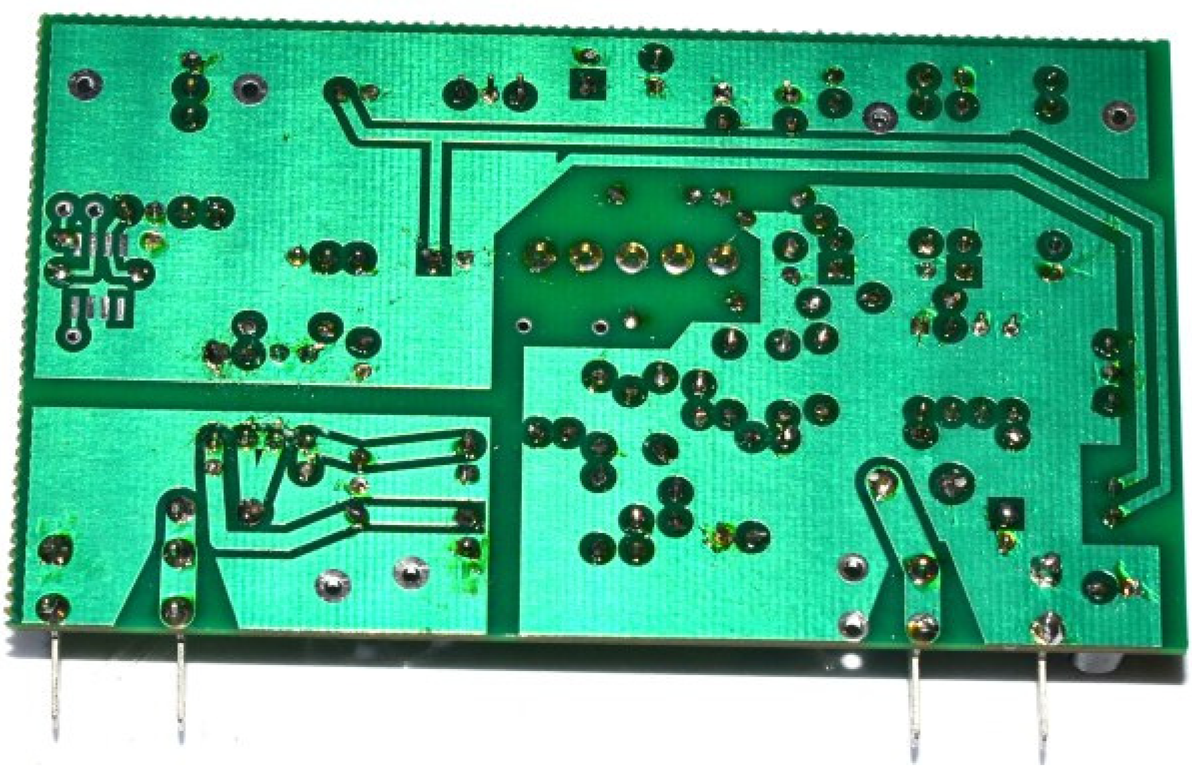
3. You may be supplied a driver board that looks different from the above. In that case, refer to the additional instructions below

Assembly sequence – standard board type

1. Group 1 – lowest-height components first
   1. Place IR1153 PFC surface-mount chip U8, solder
   2. Place 2 ISO124 isolating op-amps: U1, U2
   3. Place 8-dip sockets for U6 & U7 (if supplied) or solder A3120 chips (2)

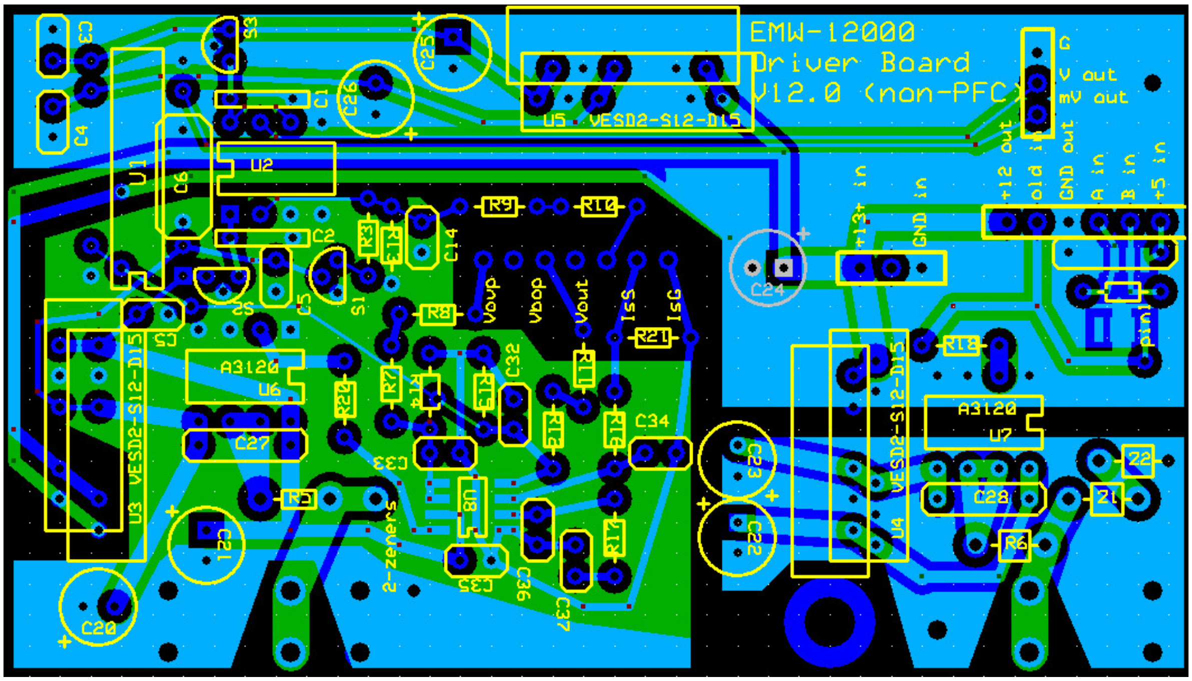
    4. Turn over, solder, trim leads
2. Group 2
    1. Place 6 1M 1% resistors: R7-12
    2. Place 3 27k 1% resistors: R13-15
    3. Place 220R-300R resistors: R18
    4. Place 90-100R resistor: R16
    5. Place 3.3k resistor: R17
    6. Place 1k Resistor: R20
    7. Place 1uF caps: C32, C35
    8. Place 100pF cap: C33
    9. Place 0.1uF caps: C27-C31, C34
    10. Place 10-12nf cap: C36
    11. Place 3.3uF-4.7uF ceramic caps: C14(could be labeled as C15, C16, or C57), C37
    12. Insert 2 3-pin and 1 6-pin male pin headers
    13. Hold parts with carton/plastic, turn over, solder, trim leads
3. Group 3
    1. Place 3 12V->15V DC-DC converters: U3-5
    2. Hold the tops of the DC-DC converters with a piece of carton/plastic and turn the board over. Solder all components, trim leads
4. Group 4
    1. Place 7 22-50uF elcaps(47uF 25V caps supplied): C20-26
    2. Place 10R power resistors: R5 & R6
    3. Place a 5-pin high voltage connector into Vbop and Vout mounting pads on the driver board. Align the connector with the locking tab facing down.
    4. Turn over, solder, trim leads
5. Insert 4 male 90-degree spade connectors into the driver board from the component side, solder
6. Insert A3120 chips into 8-pin IC sockets(U6 and U7) if not done already.
7. Insert the driver board into the female spade pins on the power board
    1. NOTE ORIENTATION – Component side of the board should face OUTSIDE.
8. Here's how the board should look after you are done:


V out
mV out
+12 in
GND in
C34
R18
R19
U11
R21
R8
R17



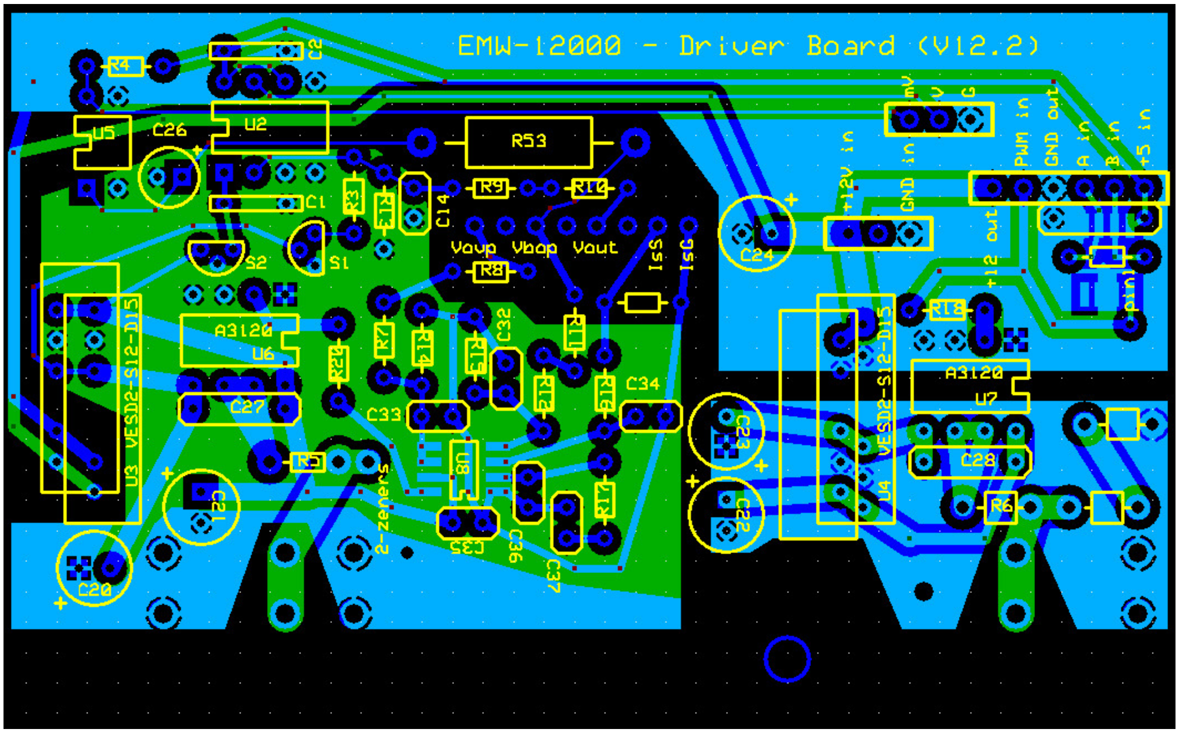
Driver board variants

1. You might receive the version of the board shown below:



   a. This is the default board for the non-PFC version but will also work in PFC setup. It uses a more noise-immune design of the battery voltage sensing circuit, based on a different opto-amplifier (A7520, U2 on the above board)
   b. The following is what's different about assembling this version of the board:
      a. Steps not required (compared to the above standard sequence)
         i. Place U2 ISO124
         ii. Place C30, C31
      b. Steps that are different
         i. R13 is now 2.7k, not 27k
      c. Additional steps needed
         i. Place 5V voltage regulators S2, S3 (78L05)
         ii. Place -5V voltage regulator S1 (79L05)
         iii. Place 0.1uF C1-C5
         iv. Place A7520 chip U2
         v. Place 68k R3

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

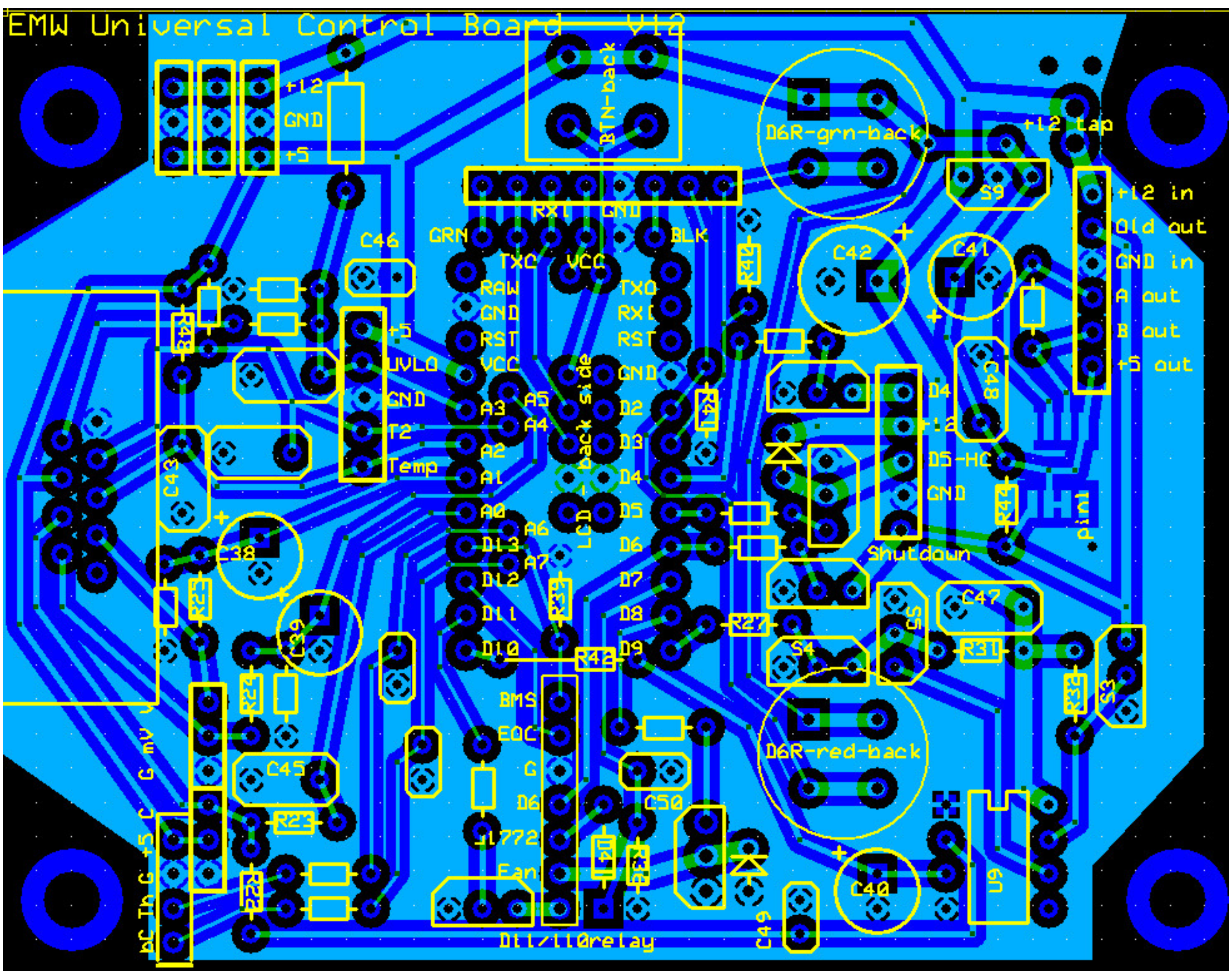


a. This version uses much simpler version of the input voltage measurement as the charger does not need precision on input voltage measurement
b. Instructions for this version will be provided as it gets released

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

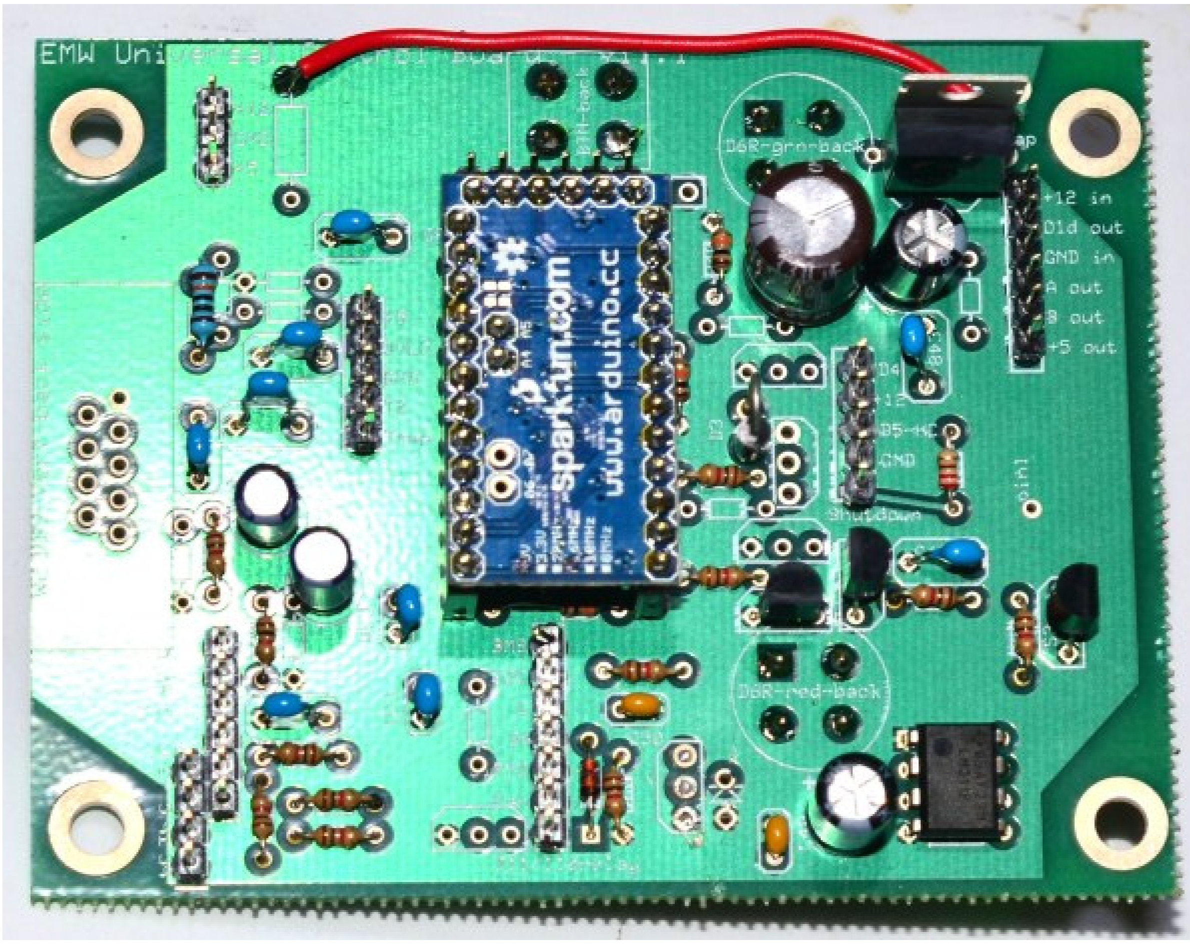


1. Solder the male pins to the Arduino board. You might receive a version of the board with all pins already placed – in that case skip this step
    1. make sure you place the pins from the component side so that Arduino board components end up facing the control board when Arduino board gets inserted 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-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-R42
3. 100k resistor: R48
4. [OPTIONAL – for UVLO shutdown – not needed in PFC]
    a. Place 6.8k resistor: R38
    b. Place 3.3k resistor: R37
5. 220R resistor: R44
6. Small signal diode: D4
7. 8-pin socket (if supplied) or LM211P chip: U9
8. 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, C45-48
    3. 10uF caps: C38-39
    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
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


EMW Universal
sparkfun.com
www.arduino.cc
D6R-grn-back
D6R-red-back
+12 in
D1d out
GND in
A out
B out
+5 out
Shutdown
GND
pin1

Part 7. Final Assembly

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 together fan leads in parallel (positive to positive, negative to negative)
            b. Solder a 2-pin female header to the resulting 2 wires (can use 3-4 pin header, just use adjacent 2 positions)
            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
            a. Flush to the fans (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
    iii. Orient with LCD facing outside, center buttons in their respective holes
    iv. Connect thermistor to the control board
        1. Use GND and Temp pins of the 5–pin connector below the Arduino board (looking at the mounted control board from the component side)

3. Mount the power board to the heatsink
    i. Pre-set 6x ¾" long #10 machine screws into IGBT / bridge mounting holes to make it easier to mount. Use lock washers
    ii. Lower the power board onto the heatsink, aligning mounting holes
    iii. If output diode is used

1. Align the bare copper lead from the output diode so it can be fastened to the output pad on the power board
2. Fasten the diode lead to the output pad using #8-10 machine screw. Use a lock washer / lock nut

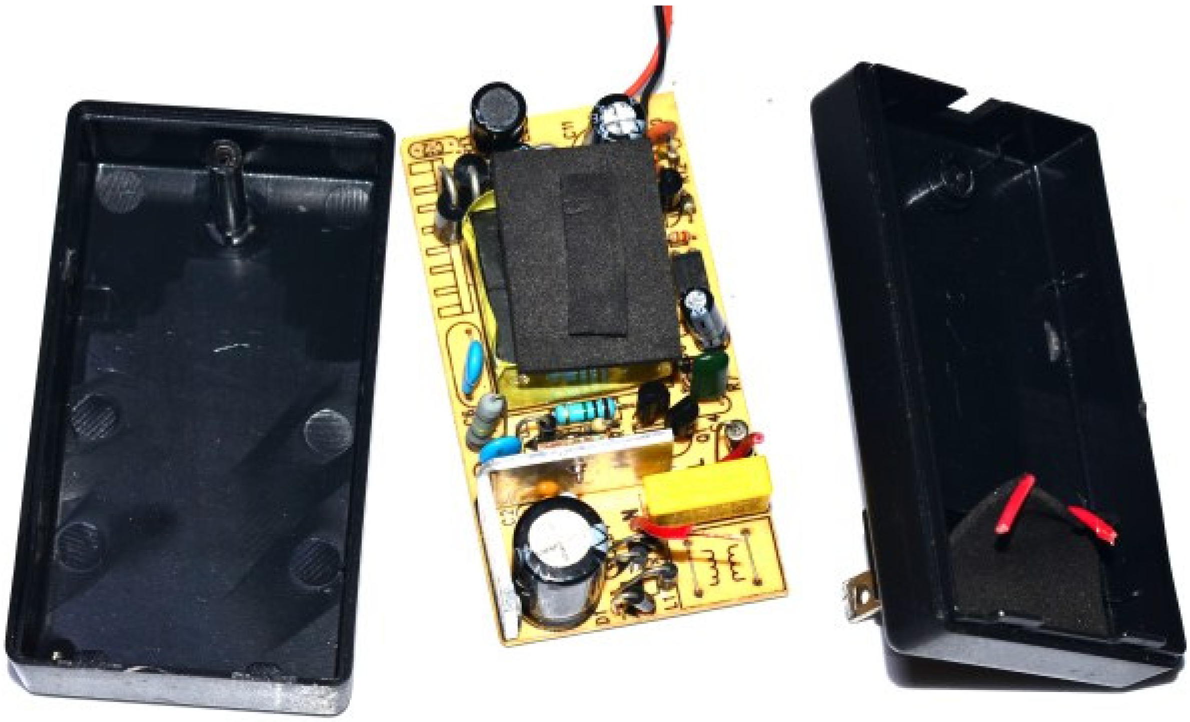
iv. Pass the screwdriver through the provided holes in the PCBs (right on top of the screws – same holes you used to mark up the heatsink earlier)

v. Fasten all screws

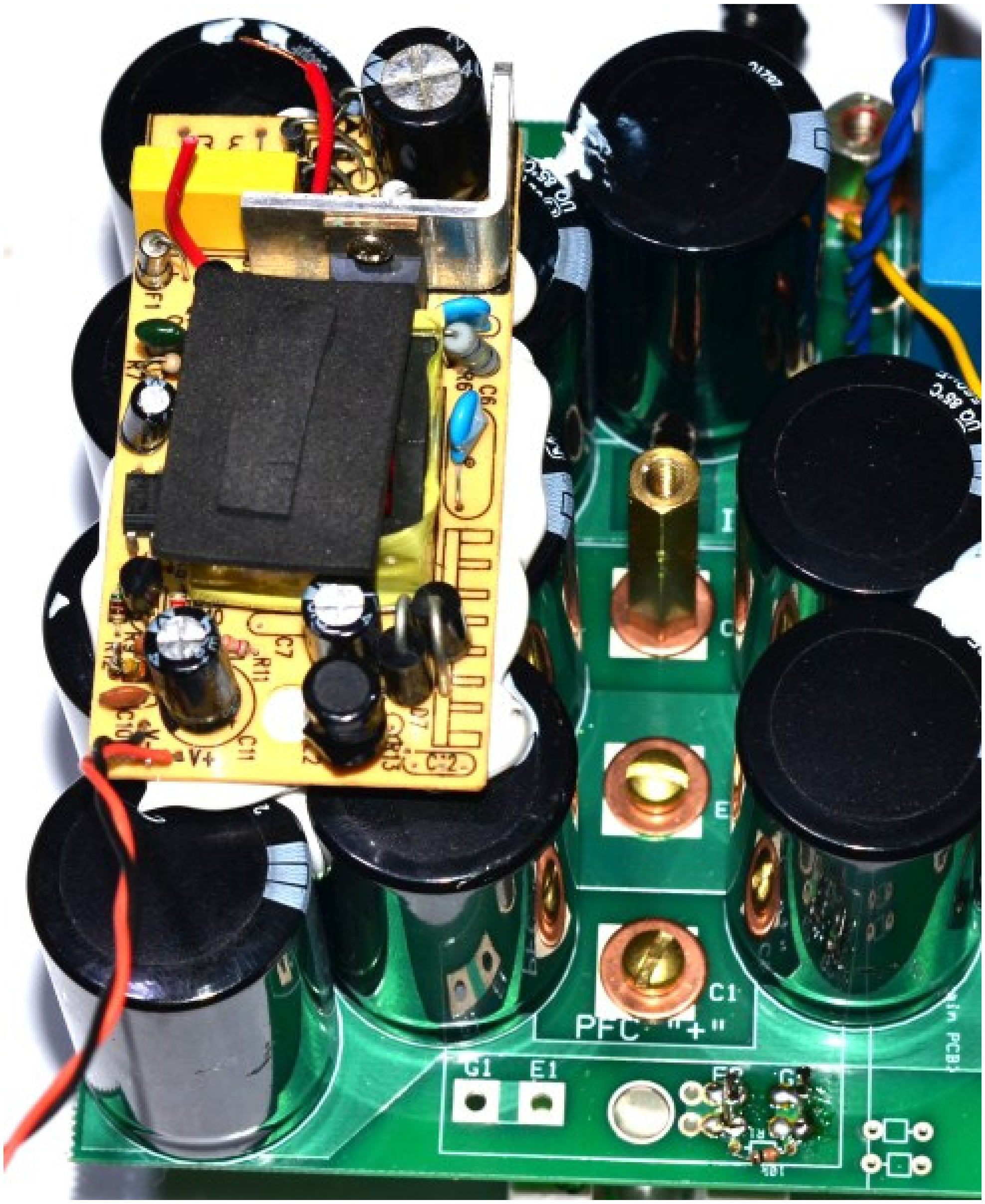
4. Wire main charger outputs
   i. Negative output
      1. 12-18” AWG6-8 black wire – connect to E2 of the output IGBT (right side)
      2. Very helpful (but not required) to use a 1.5-2” brass standoff here to elevate the mounting point off the board so the wire clears the caps easier
      3. Use lock washers on the screws
   ii. Positive output
      1. If NOT using an output relay
         a. 12” AWG6-8 red wire – connect to the output pad on the top right of the power board (or cathode of the output diode if using output diode – in that case ensure connection to both terminals so both parallel diodes are utilized)
         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)
         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 – all wires (DC and input AC) should pass in the 2-3 middle slots – this way you ensure clearing for the inductors that will be mounted later

5. 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 silicon caulk, mount the adapter on top of the elcap block on the left side of the power board. Orient the adapter so that the output DC wires go towards you
v. Set aside for 30 min to let caulk set
vi. Some pics:

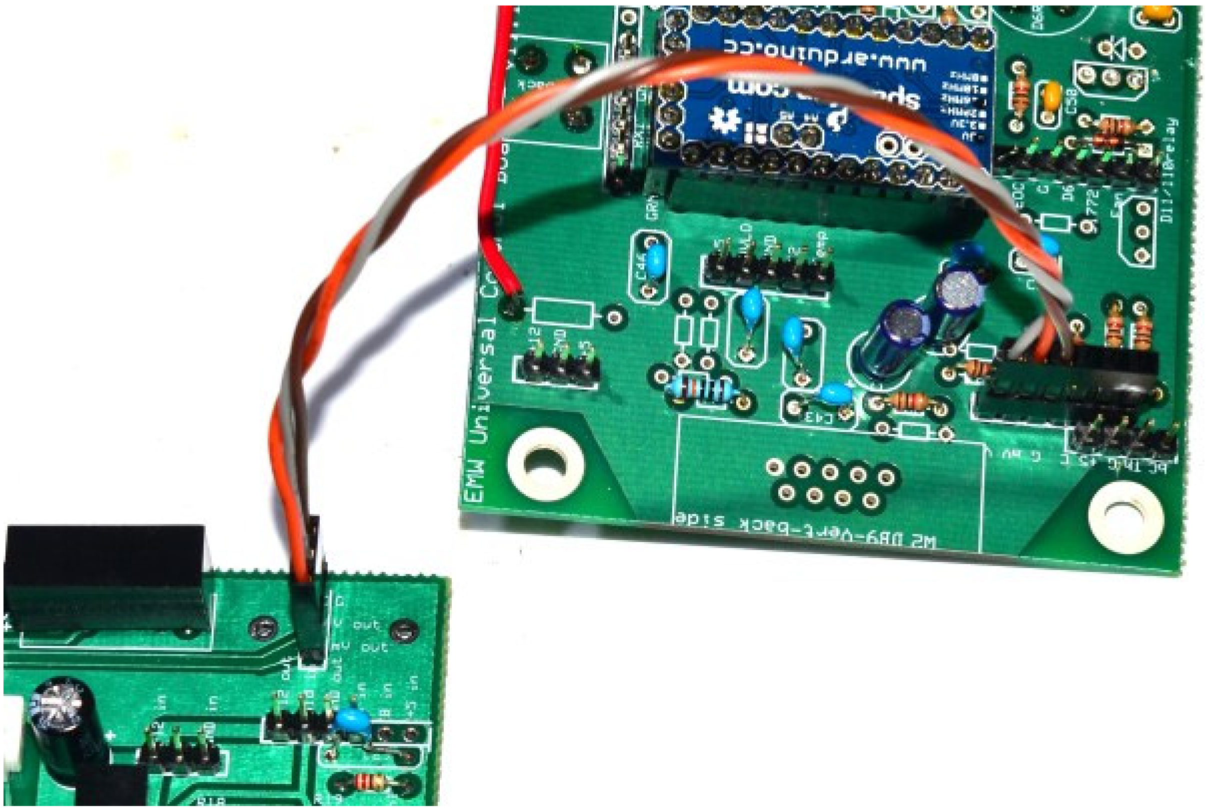



Permanently Waterproof
Supreme

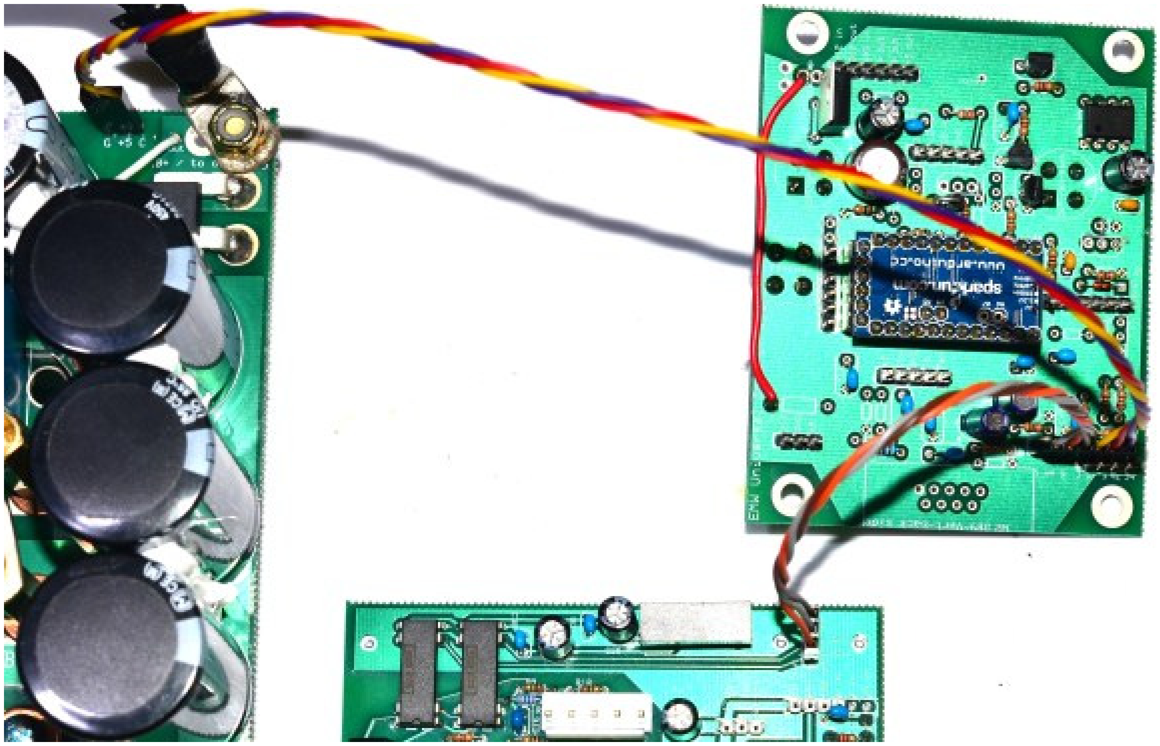


6. Prepare signal connection harnesses – you will need a total of 4 required harnesses and 1 optional. Twist all wires tightly to improve noise immunity!
    i. Voltage sensing
        1. Need: 3x 6" wires, 1x 3-pin & 1x 6-pin female headers
        2. 6-pin header will mate to the 6-pin set on the bottom 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:

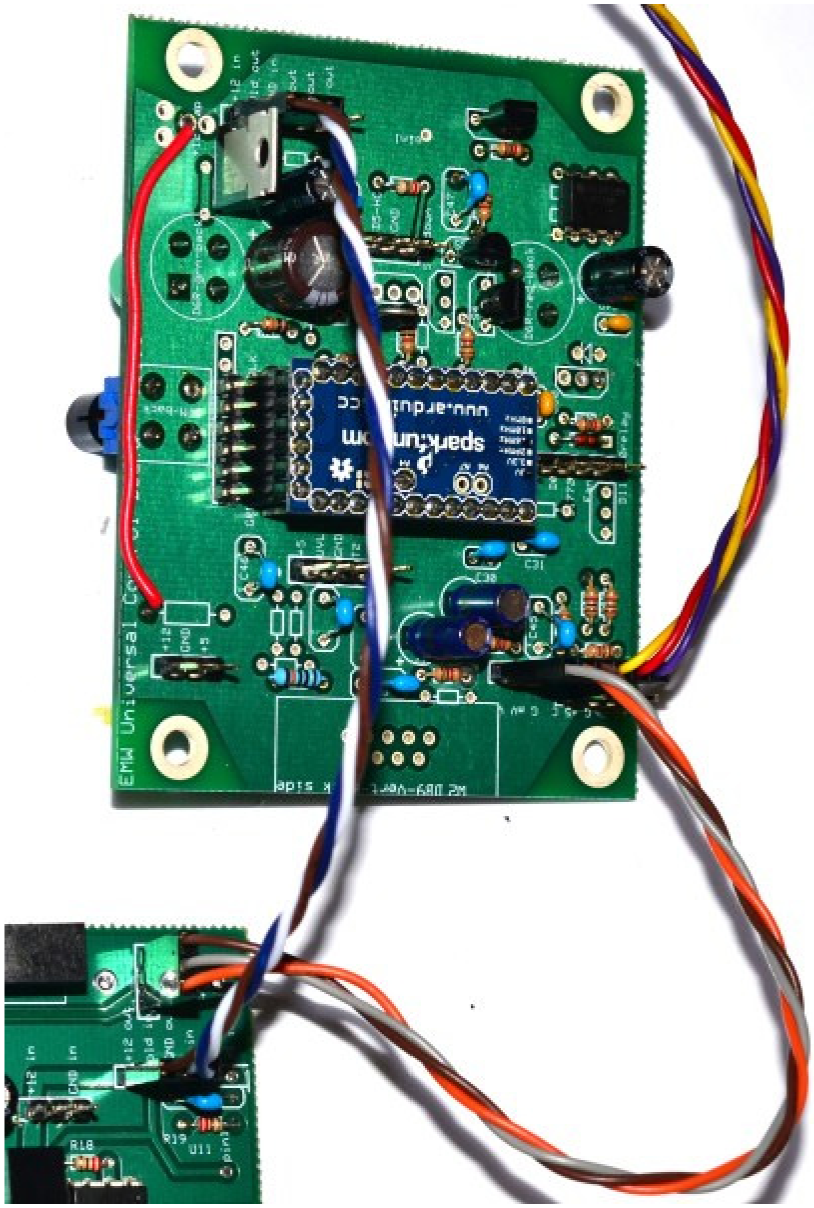


ii. Current sensing
1. Need: 3x 12" wires, 1x 3-pin female header
2. 3-pin header will mate to the current sensing pins on top right of the power board
3. The other ends of the wires will go to the 6-pin header you used in the voltage sensing step above
4. Referring to the PCBs / PCB files, solder wires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



iii. PWM signal

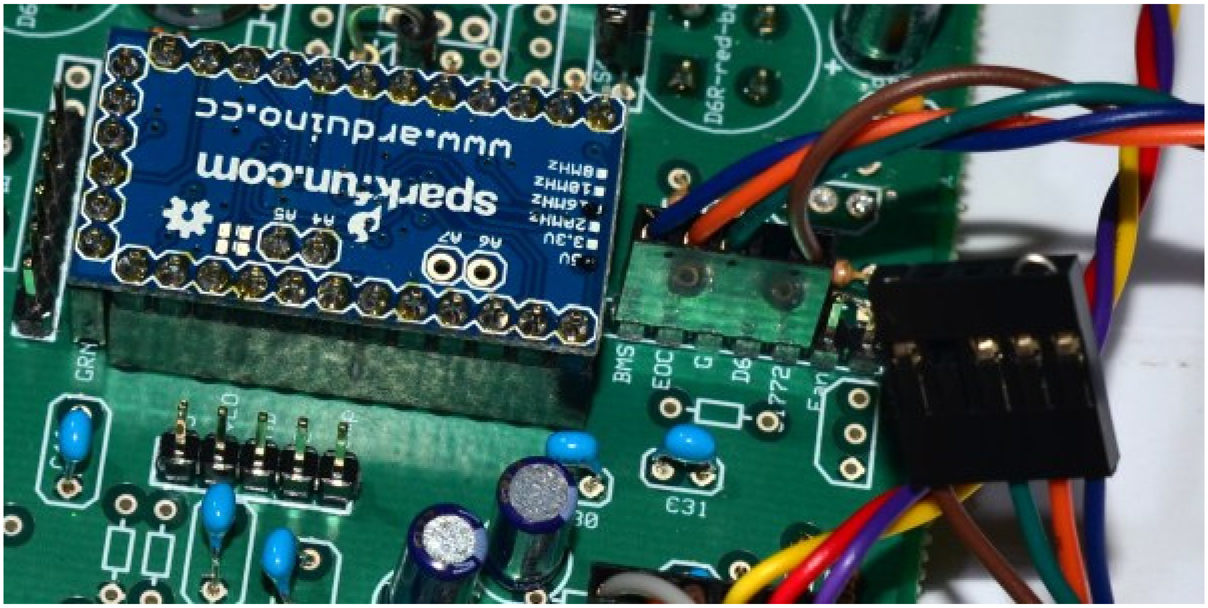
1. Need: 3x 6” wires, 2x 3-pin headers
2. One 3-pin header will mate to the leftmost 3 pins on the 6-pin connector on top left of the control board
3. Another 3-pin header will mate to the leftmost 3 pins on the 6-pin connector on top right of the driver board
4. Referring to the PCBs / PCB files, solder wires to the right positions
5. Pic here:



iv. 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:

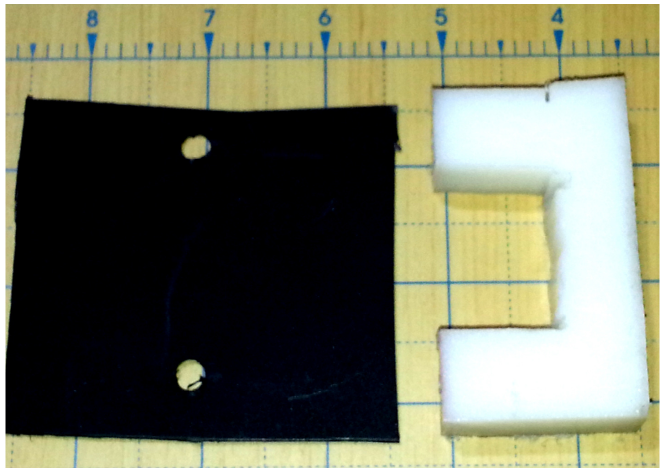


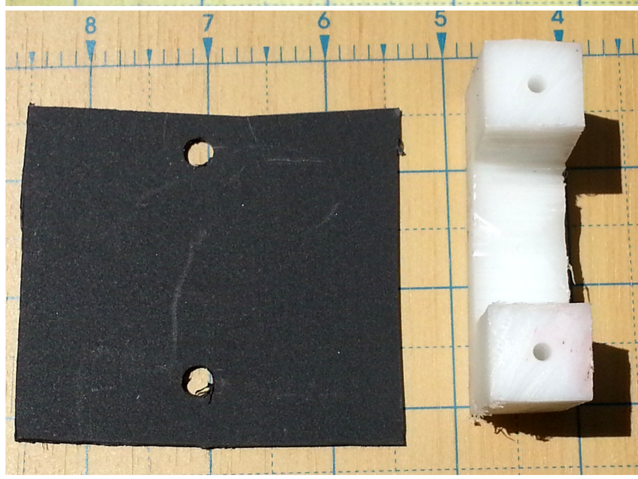
v. [OPTIONAL] Programming dongle
   1. Need: 6x 12” wires, 2x 6-pin headers
   2. One 6-pin header will mate to the bottom 6 pins on center left side of the control board (left of the Arduino board)
   3. The other header will be brought outside of the charger (through one of the venting slots on the back side) and will be used to connect the FTDI board for programming the charger in field

7. 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)
   iv. PWM (driver to control)
   v. Insert the driver board into the power board
      1. You might need to bend the male pins slightly to fit
      2. You will need to cut the leads almost flush to the driver board on the bottom of the board to make it fit with the blue film cap on the power board
   vi. Current sensing (control to power)

8. Prepare and Connect Inductors to Power Board
   1. Prepare mounting brackets

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 inductors
        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
        3. Normally, both inductors are identical so it does not matter which one goes where
        4. However, if you have ordered a 100A output inductor, that one should go to the right side. You can ID the 100A inductor by fewer turns of thicker wire bundle
    vi. Photos for reference:





ii. Connect the left-side inductor ('PFC')
   1. Cut 6" length of gauge 8/10 wire. Solder / screw one end to the Ind1 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 left-side IGBT (marked C2 E1)
      a. 1.5"-2" brass standoffs help clear the caps but not required

iii. Connect the right-side inductor ('output' or 'buck')
   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

9. Assemble a HV input/output connector

i. Use a supplied 4-6 position connector
ii. 2 positions to be used for input AC lines
iii. 2 positions – for output DC
iv. Remaining positions (if any) can be used for Ground connection and (in non-PFC units) a separate connection to 110VAC line
v. Using the connector as template, drill mounting holes in the back side of the enclosure, right above the venting slots (position connector horizontally)
vi. Connect the 4 power wires (that you have previously threaded through the venting slots) to the corresponding positions of the connector
vii. 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!
viii. Connect inrush current limiters
    1. 2x 50A limiters are normally supplied - use 1 on each AC line
    2. 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 the limiters
        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 silicon caulk to fix the leads to some metal parts (including the box of the charger. Caulk is normally good for 400F
    3. 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)

10. Prep your charger for testing
    i. 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
    ii. Check for loose bolts / connections
    iii. [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)
    iv. If desired, do same for all remaining PCBs and connections

Part 8. Power Up

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'
   v. Check the "#define …" statements in the first 2 pages of the code (marked as '--------- MAIN SWITCHES -----------')
      i. Uncomment '#define PFC' if this is a PFC unit
      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. 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)
   ii. In Arduino IDE, click Compile icon, confirm error-less compilation
   iii. 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 those '#define A7520' switch in the firmware.
        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. 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%.
h. The output voltage reading on LCD should be very close to your battery voltage. The current reading should be zero (or 1A max)
i. Check the heatsink temperature readout. Should be close to ambient
j. 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 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 AC input to the small AC wires that you earlier had attached to the power board to the input of the AC adapter (that should by now be firmly attached to the cap block with the caulk)
    ii. Connect 110V to the 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)
        2. The PFC stage will turn on, potentially causing dimming the lights for 1-2 seconds. This is normal
    iii. Let the charger time out through the calibration routine (5-second timeout)
    iv. Set the power level
        1. If the charger stops to ask for input power, set input and output power to 10A
        2. 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
        3. During the power setting routine, note the stated input voltage. It should read close to 110V
    v. Check PFC operation
        1. CAREFULLY measure voltage between C1 and E2 terminals of either IGBT. It should read between 350 and 400V. If it reads 160V or similar, PFC stage is not working. Time to debug
    vi. Measure the output of the charger. Should be close to zero (remember that you still have the lamps connected to the output)
    vii. Start the charger
        1. Using the control buttons, navigate through the menu to start the charger
        2. Confirm run
    viii. This time, the duty should ramp up from zero to ~30-35% (375V PFC stage output being reduced to ~100V) before charger should go into the

CV stage

ix. Watch the duty to go from zero to ~30-40% and stabilize there. Lamp should be reasonably brightly lit at this point
x. 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)
xi. Let the charger run for 10 min. Check the heatsink temperature readout on the screen. Should be close to ambient
xii. 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
   iii. Repeat the steps from the previous limited test, confirm normal operation
   iv. Power cycle 220VAC
   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
   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> / 380VDC)
      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