Message from the 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:

1. You will understand the fundamentals of power conversion – something that's in high demand in today's society fueled by electrical energy.
2. You will feel the enjoyment of getting things done with your own hands.
3. 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. Your support is invaluable and helps us bring more and more innovation to you. Your money go into not only into improving this product but, even more importantly, bringing completely new, revolutionary products. Just look at our most recent products such as an Open Source 15kW EVSE (aka JuiceBox) and a CHAdeMO controller - now fully operational and tested on Nissan LEAF up to 44kW - some cool videos here.

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 Electric Motor Werks and any of its directors, employees, and partners are not liable for any damage that may result from this project and associated activities.

Furthermore, there is no explicit or implicit warranty or guarantee of applicability for any particular purpose. For Kits, E Motor Werks will only warranty the parts to be free from manufacturing defects for 1 (one) year from the date of purchase. For the assembled units, we will warranty the parts and complete assembly to be free from manufacturing defects for 1 (one) year from the date of purchase.

WARNING: This document describes circuitry that is directly connected to the AC mains, and contact with any part of the circuit may result in death or serious injury. By reading past this point, you explicitly accept all responsibility for any such death or injury, and hold Electric Motor Werks, Inc. harmless against litigation or prosecution even if errors or omissions in this warning or the document itself contribute in any way to death or injury. All mains wiring should be performed by suitably qualified persons, and it may be an offence in your country to perform such wiring unless so qualified

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.

Table of Contents

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

The EMotorWerks SmartCharge-12000 is a fully programmable, open-source (for non-commercial use only) 12kw intelligent PFC charger. Designed & manufactured by Electric Motor Werks.

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 our 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 Electric Motor Werks and not direct from DigiKey, etc.

A: Few reasons:

- You will save money by buying from us. We recently 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 (as of September 2013). 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 Electric Motor Werks 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… We do expect you to be a proficient electronics builder, however - if this is the first complex electronics project for you, DO NOT DO IT!

Design notes:

The original charger design was created early 2011 and was the basis for our kits up until May 2012. Information on that design is still available from Electric Motor Werks on request.

The version described in this document 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
- 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 120/240V switching, under-voltage protection is included (some of that is optional in default kit configurations)
- Firmware optimized for variety of applications - from standalone charger use to use in CHAdeMO and CCS DC Fast Charger applications

Specs

Input voltage:

- 90-260VAC for the PFC version (output power automatically limited to ~1.5kW if connected to 120VAC)

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

Output current:

- Up to 70A continuous (100A option available on request with custom inductors)

Output power:

- 12kW rated
- Tested to 18kW in our labs
- Up to 35kW possible in DC-DC mode! (see section MODs in the end of this document)

Efficiency at full power:

- 93%+ for PFC version

Switching frequency:

- 22kHz (PFC stage)
- 15-20kHz variable depending on load, etc (buck output stage)

Additional features:

- [new as of August 2013] Serial control of the charger is now implemented – the charger can now be automatically controlled via a UART interface. See interface details in our Quick Start document (link on the first page of this doc)
- BMS stop-charge TTL input (active LOW)
- End-of-Charge TTL output (active LOW)
- Hardware current limiting
- SAE J1772 support with automatic power regulation based on J1772 pilot signal
- 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:
    - Number of batteries
    - Battery capacity

Part 0. Before you start

1. **Required tools:**
    a. Soldering setup
        i. A low-power soldering station with a relatively fine element – something like this (what we use, $20 from Amazon)
        ii. A high-power station / gun (100W minimum, 250W or more preferred) – something like this (what we use - $70 from amazon)
        iii. Electrical solder (make sure you never use 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
        iv. 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
    b. Screwdrivers
    c. Wire strippers
    d. Small snips for wire / lead cutting
    e. Small pliers
    f. Drill & drill bits
    g. Threading taps: 10-32 or 10-24, 6-32
        i. See http://www.wikihow.com/Use-a-Tap for some tips
    h. Multimeter with Capacitance / Resistance measurements. This thing is awesome and is what we use ($130 at Amazon – pricey but worth it!)
        i. Also download / print the resistor color coding reference: http://en.wikipedia.org/wiki/Electronic_color_code
    i. Clear Protective Goggles
    j. 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
2. 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 1 MHz bandwidth (10 MHz or higher preferred).
        i. Our favorite is this 100MHz OWON scope ($440 at Amazon). Probably an overkill if you just want to assemble one charger
        ii. You can also 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)
        iii. Finally, you can get one of these small units – they are open source and generally don't have huge bandwidths / feature sets but will do the job just fine. In fact, we love these due to their portability – you can take this

battery-powered unit anywhere!

2. **Required software**
    a. In order to read the schematic / PCB layout files, you will need to install ExpressPCB software from http://www.expresspcb.com
    b. You will need to download and install the right version of the Arduino IDE software – see the FIRMWARE section below for specific recommendation on the version you will need

3. **Helpful aids**
    a. 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
    b. Small soldering vise to hold pcbs while you solder
    c. Magnifying glass to read small parts' markings

4. **Required additional commodity parts** (not supplied with the kit, available at any hardware / electronics store or (soon) as an add-on option in our online store)
    a. Bolts, nuts, washers: #10 and #8, various lengths
    b. Signal wiring
        i. Ideally, a set of Pololu pre-crimped female-female wires and housings
            1. Housings (http://www.pololu.com/catalog/category/70): 5x 3-pin, 4x 6-pin (with 2 of them used as 5-pin)
            2. Wires (http://www.pololu.com/catalog/category/71): 3x 12", 10x 6", 3x 24"
        ii. Alternatively, could use 5-6 colors of AWG22 isolated wire (~3' of each color)
    c. Power wiring – ideally 3-4 colors of AWG 6-8 isolated multi-threaded wire (2' of each color) plus 1' of bare copper AWG10 wire
    d. Silicone sealant.
        i. 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.
        ii. 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.

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

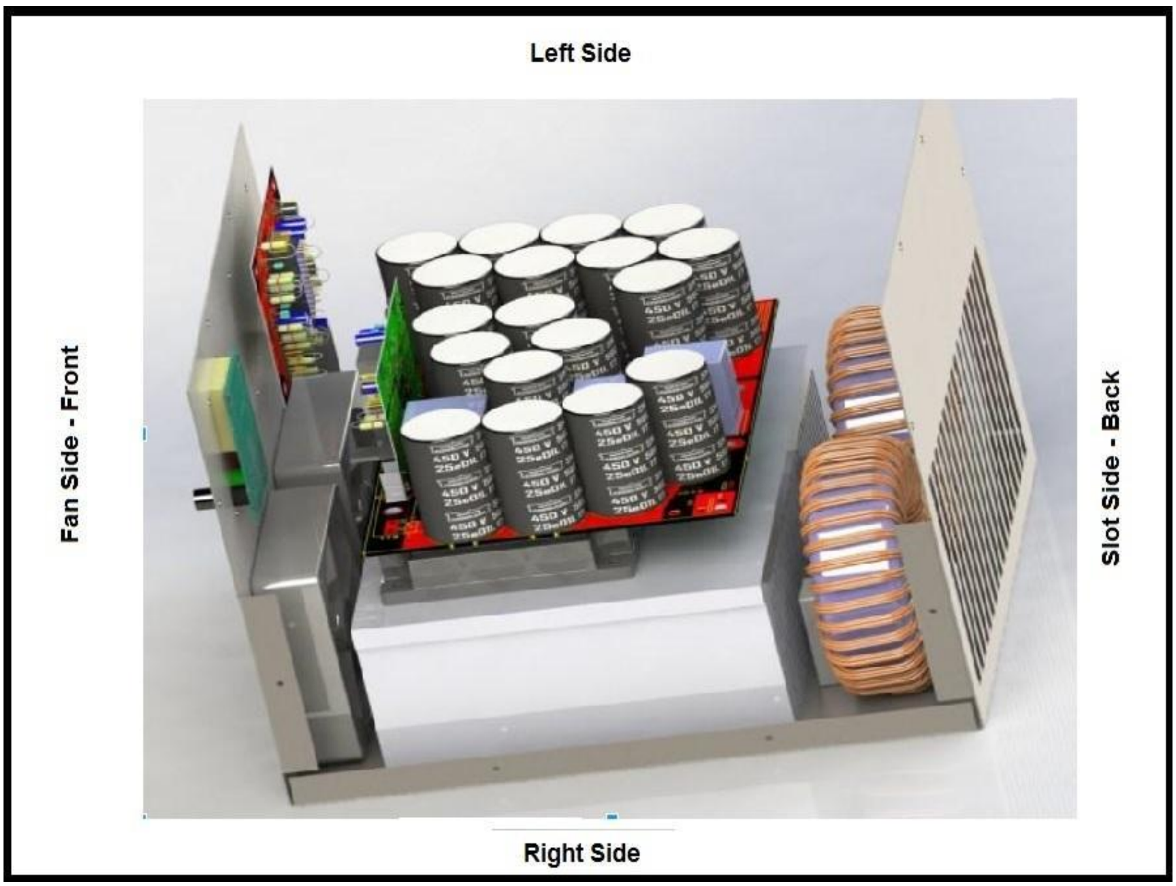
d. Place many parts at a time, bending pins on the other side of the board so that the parts stay in place when you turn over the board to solder
e. It is recommended to use a good flush cutter and then go over each soldered connection with another application of solder (or just flux and touch with the iron), so that you can run your finger over the bottom of the board and feel any sharp edges. Each connection should be a smooth "bubble" of solder. And this procedure also double checks for possible missed solder joints.
f. 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.

6. **Education**
   a. Wouldn't it be nice to actually understand what you are building?
   b. 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
   c. A couple of good ones are:
      i. **Switching Power Supplies A - Z, Second Edition** by Sanjaya Maniktala (Apr 18, 2012) - $60 on Amazon
      ii. **Switching Power Supply Design, 3rd Edition** by Abraham Pressman et al (2009) - $65 on Amazon

Overall build reference:

The ‘FAN-SIDE’ of the chassis is considered the ‘FRONT’. The ‘SLOT-SIDE’ is on the back. “Left” and “Right” terms in the rest of this document are referenced with respect to the front ‘FAN-SIDE’. 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.



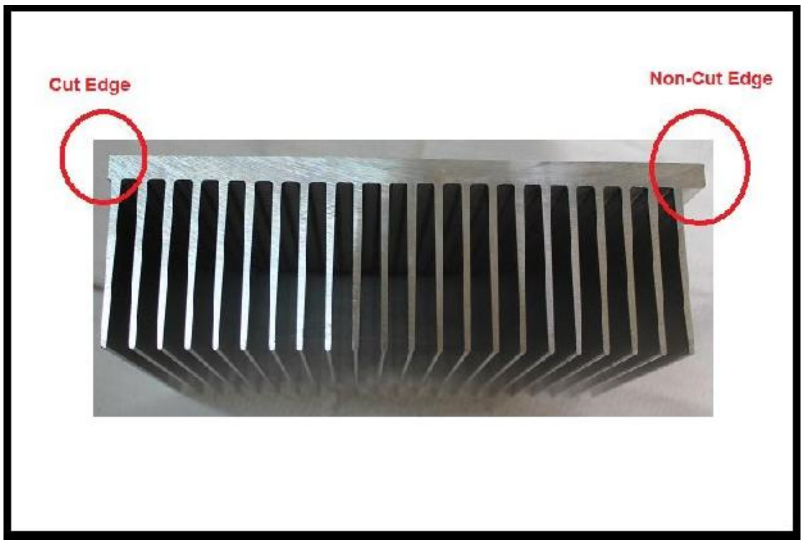
And these below are the actual photos:

Part 1. Kit Contents

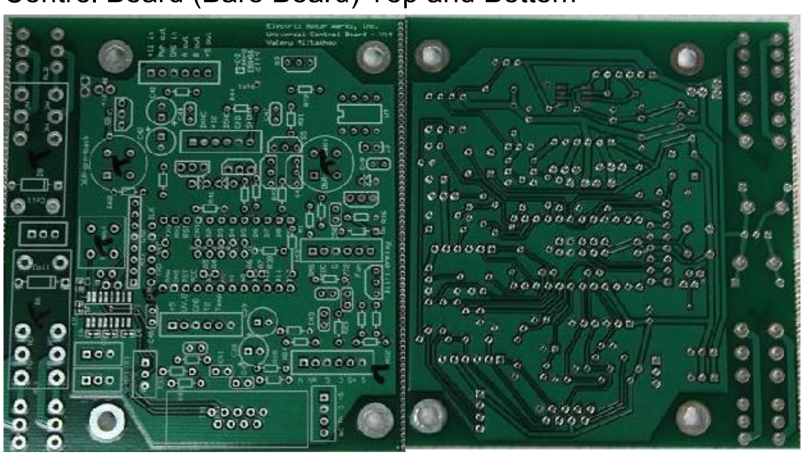
Please refer to the Bill Of Materials (v14) for a complete list of parts. For more details on component lists, part numbers, etc, please refer to the 'Quick Start Manual' . Tip: enter part number into http://www.DigiKey.com search box to get a detailed part info page with full datasheets, photos, etc.

Most of the smaller parts will be placed in anti-static bags, 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).

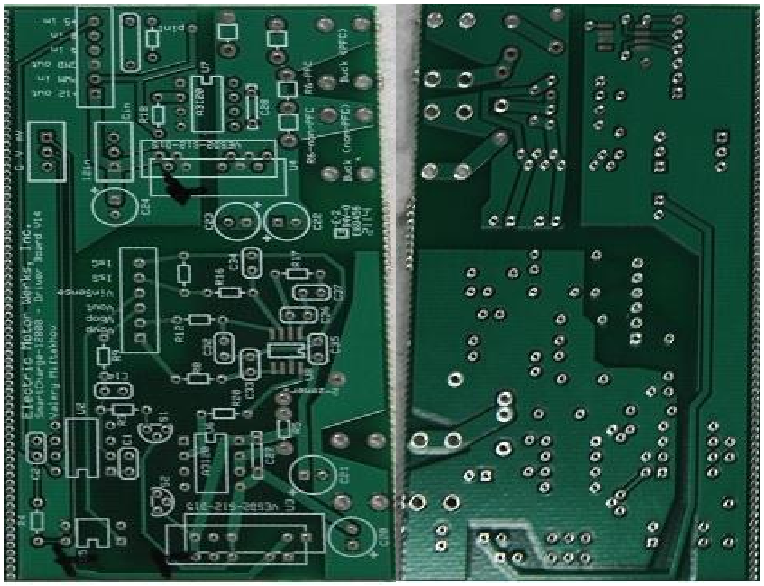
1. Air cooled Heatsink:



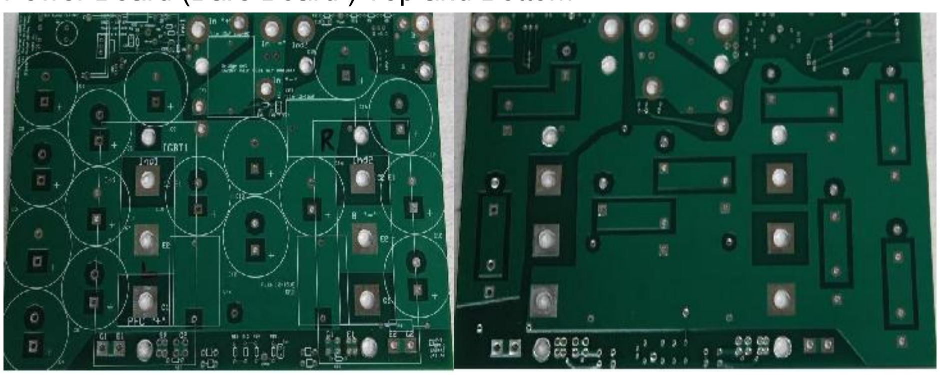
2. V14 PCBs. (these slightly change from version to version!). Pictures not to scale.
    1. Control Board (Bare Board) Top and Bottom



2. Driver Board (Bare Board) Top and Bottom



3. Power Board (Bare Board) Top and Bottom

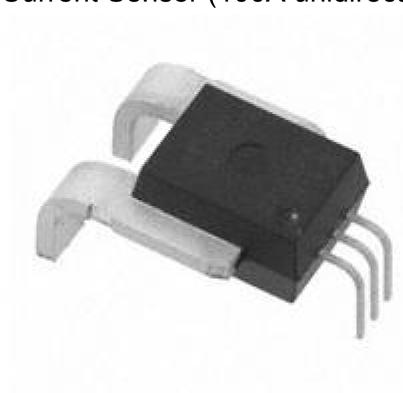


4. FTDI board and USB cable (to program Arduino)

3. x2 Electric Motor Werks High-Power Custom Inductors:
    1. Standard order has 2 Identical inductors rated for 70A DC current
    2. Picture of a 70A Inductor

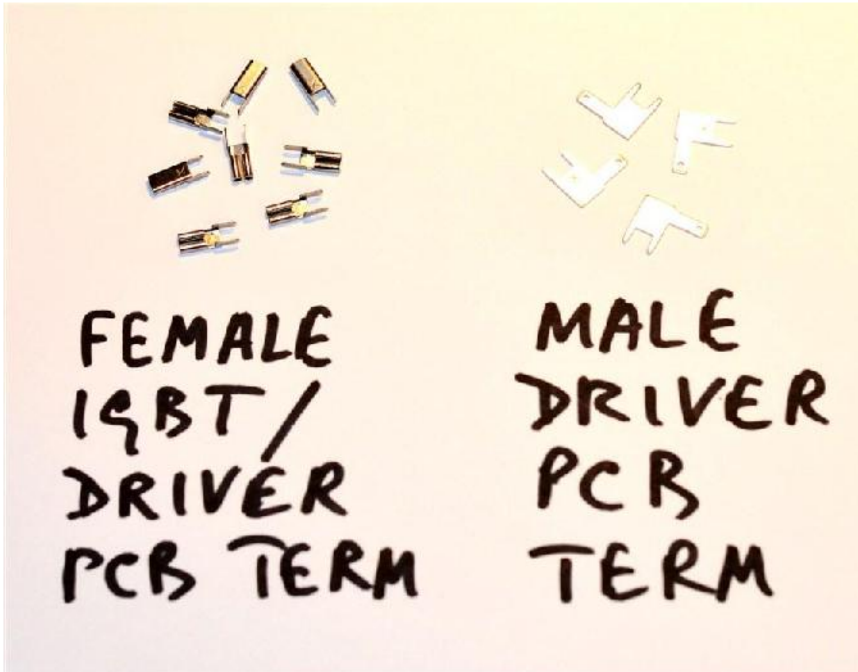
4. Integrated Circuit (IC) Parts:
    1. Arduino Pro Mini 5V / 16MHz board

    2. IGBT Driver Chips (A3120)
    3. Comparator chip (LM211P)
    4. A7520 opto amplifier
    5. Current Sensor (100A unidirectional Allegro Hall sensor ACS758xCB)



    6. PFC chip (IR1153)
    7. x2 DC-DC converters - used to convert 12V to 15V

5. Connectors:
    1. 3 sets of 40 male breakable pins (0.1” pitch)
    2. Assorted female .1 inch headers
    3. Various terminal types for Power Board and Driver Board



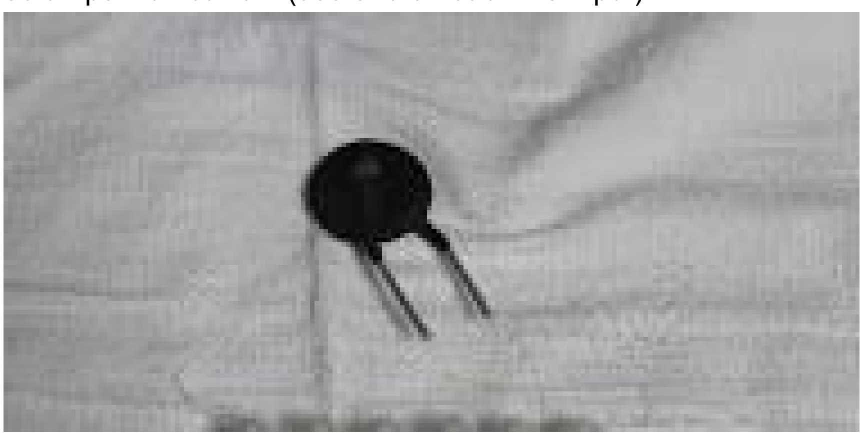
4. Control & programming / LCD buttons
5. High-power Input Connector: 6 positions (minimum of 2 for Input AC power and 2 for Output DC)

6. Resistors:
    1. Most are 1/8 W (1/4W will also work by mounting them vertically)
    2. 3W 10R gate drive resistors
    3. 2-3W 200k safety resistors

7. Capacitors:
    1. ~10x small electrolytic (labels directly on the caps)
    2. ~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)
    3. 3x rectangular film caps used on Power Board
    4. 17x Power Capacitors for the power Board (generally shipped wrapped together with the power board)

8. Color LCD:
    1. 1.44" diagonal 128x128 resolution
    2. Pre-programmed for using a simple serial protocol

9. 2 in-rush limiters:
    1. 50 amps max current (use one on each AC input)



10. Semiconductors:
    1. Small N2222 transistors
    2. 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 on the body
    3. 5V linear regulator (7805, TO-220 body)
    4. 1x small signal diode
    5. Integrated Diode Bridge module (3-phase, can be used in a single-phase configuration)

6. 2x IGBTs
    i. 145 amps, 600V
    ii. Some IGBTs may have different tab positioning, bend them to match layout if so
    iii. 

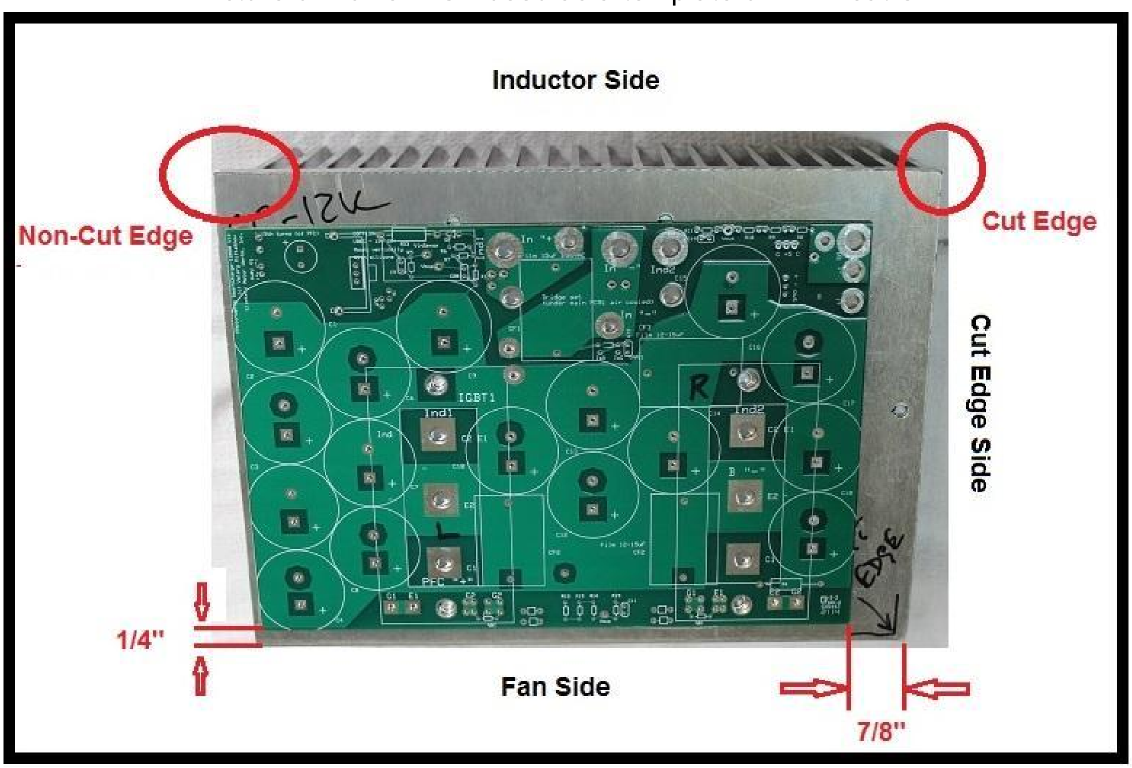
11. 12V regulated AC supply rated for 4-5 amps (to power the fans and all circuitry in kit)

12. Optional Enclosure and Fans. Fully machined enclosure matching your kit.

1. No drilling / cutting required. Save yourself 20+ hrs of metal work
2. Cooling is further optimized by a precisely slotted back
3. Make your charger build look professional. Chances are, you'll bang up the standard enclosure a bit during your build. With the fully machined enclosure to start with, you'll have a pristine build when finished.

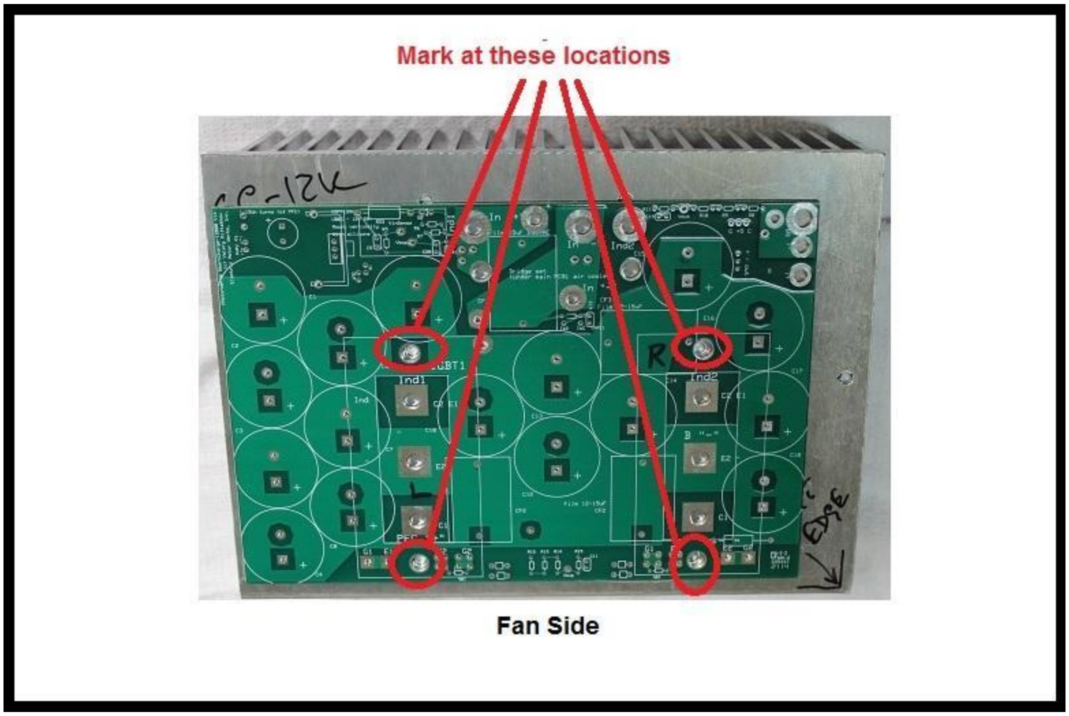
Part 2. Marking up the Heatsink

1. Using the Power Board PCB as a template, prepare to mark the top of the heatsink with a permanent marker.
    a. (Note: make sure the majority of holes are drilled between fins)
    b. Place the Power Board 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)
        iii. Picture of Power PCB used as a template on Air Heat Sink

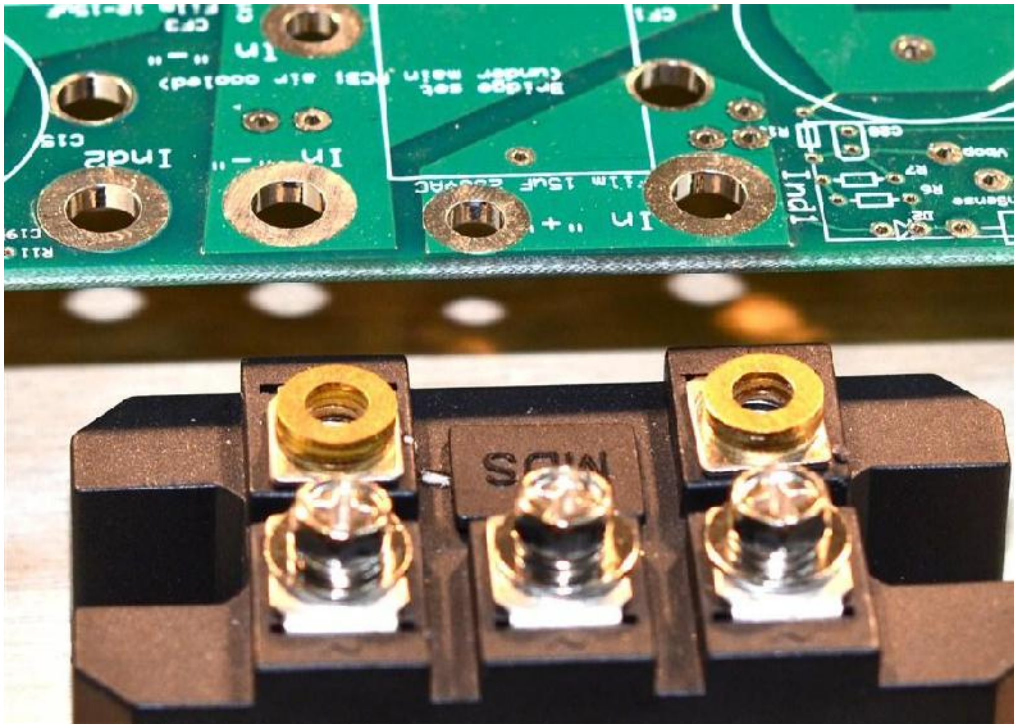


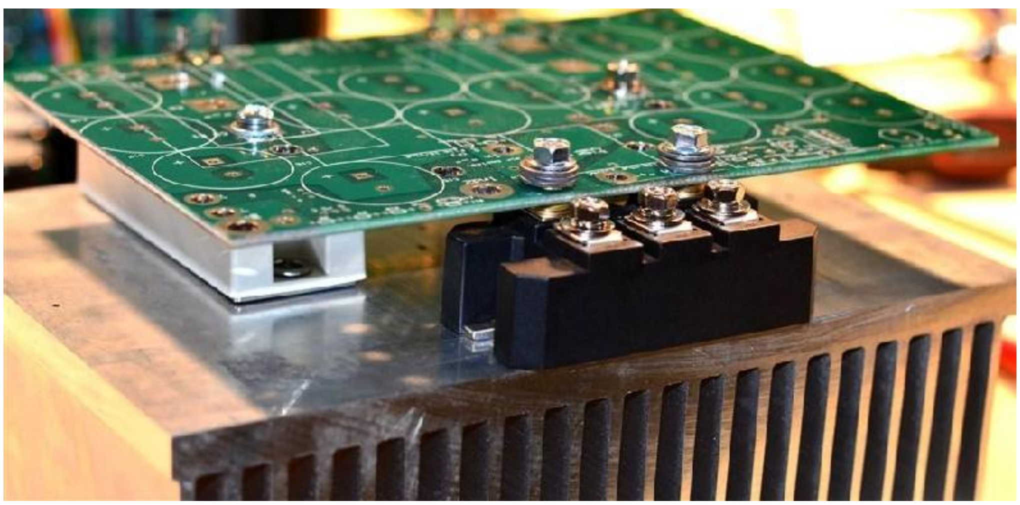
2. Use a permanent marker to mark up the hole positions:
    1. Use power board PCB as a template. (Note: make sure the majority of holes are marked between fins)

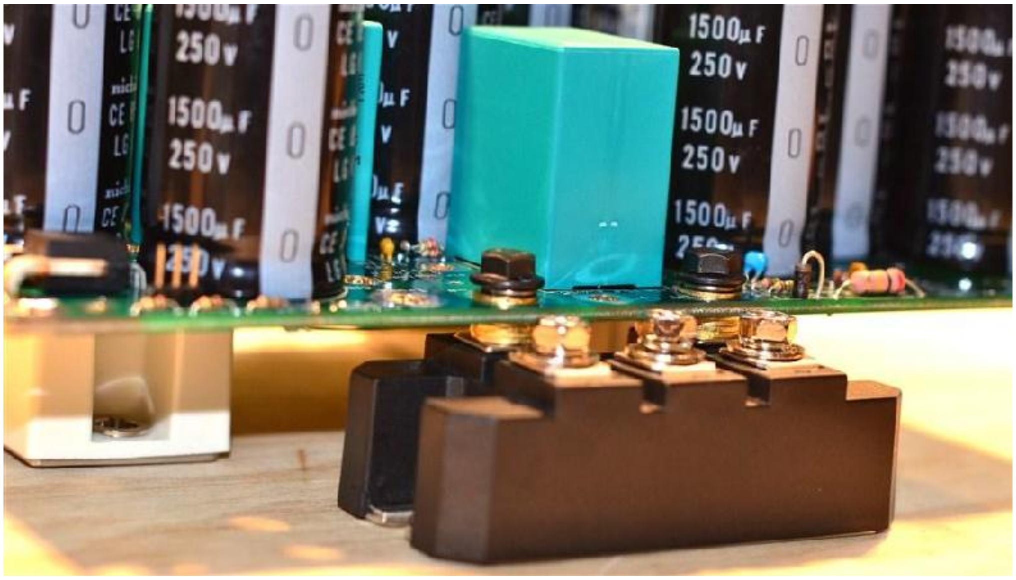
2. x4 Power board IGBT screw holes as shown in the picture below.

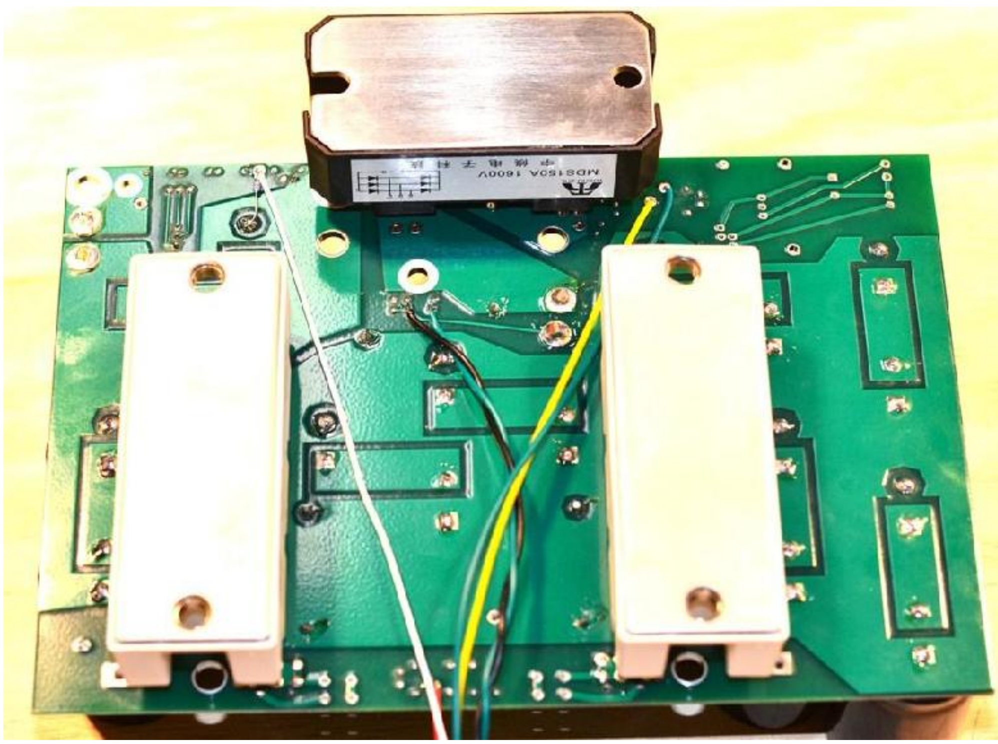


3. 1 center hole for the thermistor (partially drilled through)
4. 2 holes for Diode bridge module. Best way to do this is to (see photos below)...:
   b. ...take your 2 IGBTs and mount them onto the board with at least 2 screws per IGBT
   c. ...turn over the board and place on a heatsink
   d. ...take a diode bridge module, remove 2 screws from the output terminals (market + and -), place module on heatsink right under the board, next to the mounting points for +/- terminals
   e. ...take several small copper washers and insert them between +/- terminals of the bridge and the landing pads on the board. Total thickness of the washer stack should be around 0.14-0.15”
   f. ...using ½”-long 10-32 machine screws, mount the board on top of the bridge
   g. ...align IGBT mounting holes with the markings you made in the previous steps
   h. ...tighten all screws
   i. ...mark diode bridge mounting locations (it’s ok if the bridge slightly - by 1/8th of an inch - overhangs outside of the heatsink)









5. Inrush protection using an Output Diode Markup - Option 1
    b. Using an output diode for output inrush protection (options for PFC but could be helpful if you plan to connect and disconnect your BATTERY often). 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)
    c. Place a diode on the top right corner of the sink
    d. Align mounting holes horizontally, with an open hole facing right

e. Center of the open hole should be ~20mm from the top and 20mm from the right edge of the sink
f. Mark up the holes
g. 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

6. (Optional Markup)
   b. Inrush protection using a Relay - Option 2
   c. Use an optional 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
   d. Relay = lower loss (~60W) but more complex connections / control (need precharge parts and Arduino relay control)
   e. Relay = much more space required
   f. As a result, especially for high-voltage / low-current (on the output) 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)

7. Mark up 2 holes for mounting the heatsink itself
   b. Mark in between the two final fins on each side of the heatsink (Center Left side and center right of the heatsink)

Heatsink with part positions marked

3. Next, drill and tap:
    f. The first six holes (x4 : IGBT & x2 : Integrated Diode Bridge):
        i. drill and tap for 10/32 thread
    g. Drill the thermoresistor hole (center) ~⅜" into heatsink (not through!) - with same bit as before
    h. The optional diode holes (top right): drill and tap for 6-32 thread
    i. Final 2 heatsink mounting holes (next to right & left edges) need to be drilled with a slightly larger drill bit: has to allow the 10/32 or 10/24 bolts to pass through

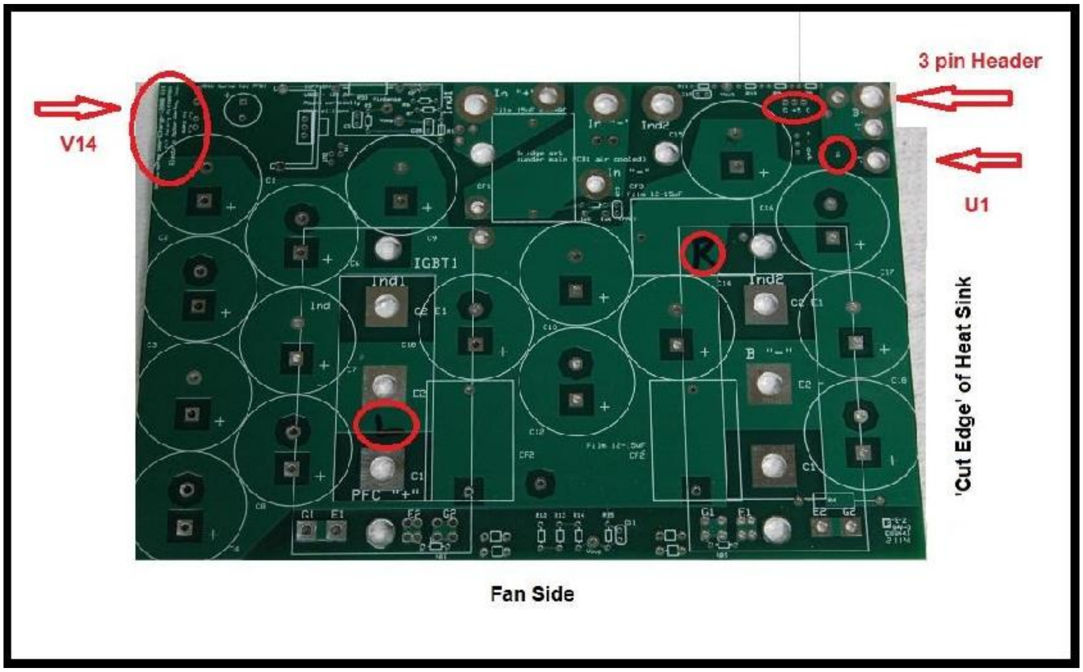
4. Mount the output diode (if needed):
    g. Make sure you use thermal paste between diode and sink/plate
    h. Note that the diode supplied is a dual package in which you have to use both diodes connected in parallel
    i. Prep the diode connection wires:
        i. Cut 6" length of the bare copper 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

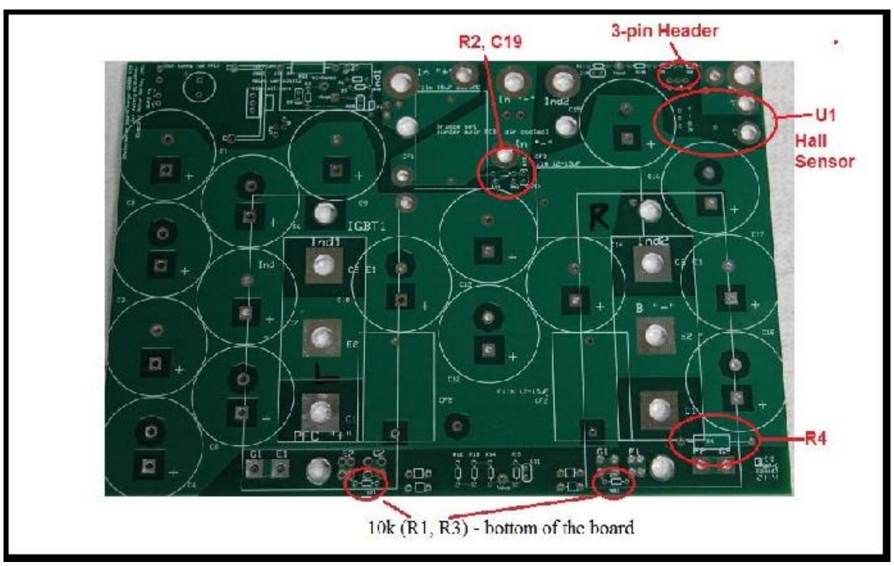
5. Thermistor Assembly
    a. Cable assembly
        i. Make a 3-pin (only 2 side pins used) thermistor connector
        ii. Take x2 12" signal wires and solder one to thermistor
        iii. Heat-shrink wires and solder one end to thermistor
        iv. Solder other ends of the wires to the ¾-pin female 0.1" pitch header (use positions 1 and 3)
        v. Using silicone sealant, secure the thermistor in the thermistor hole in the heat sink. Refer to the Heat Sink hole markup picture for the location.
        vi. Here's a visual walkthrough:

Part 3. Assemble Power Board

Major Landmarks



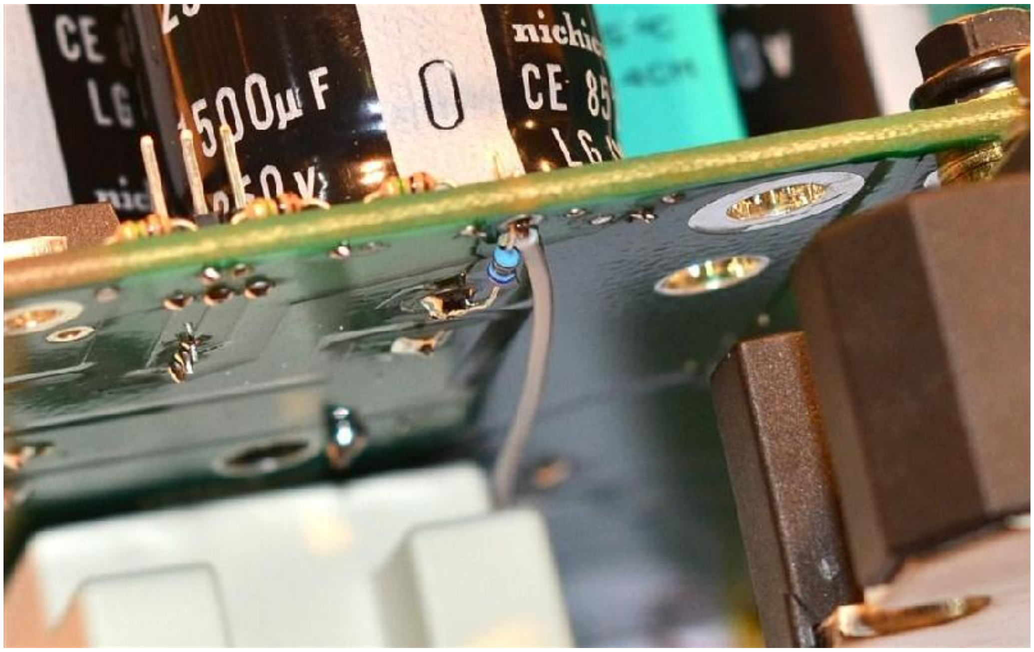
1. Place all smaller components:
    a. Resistors:

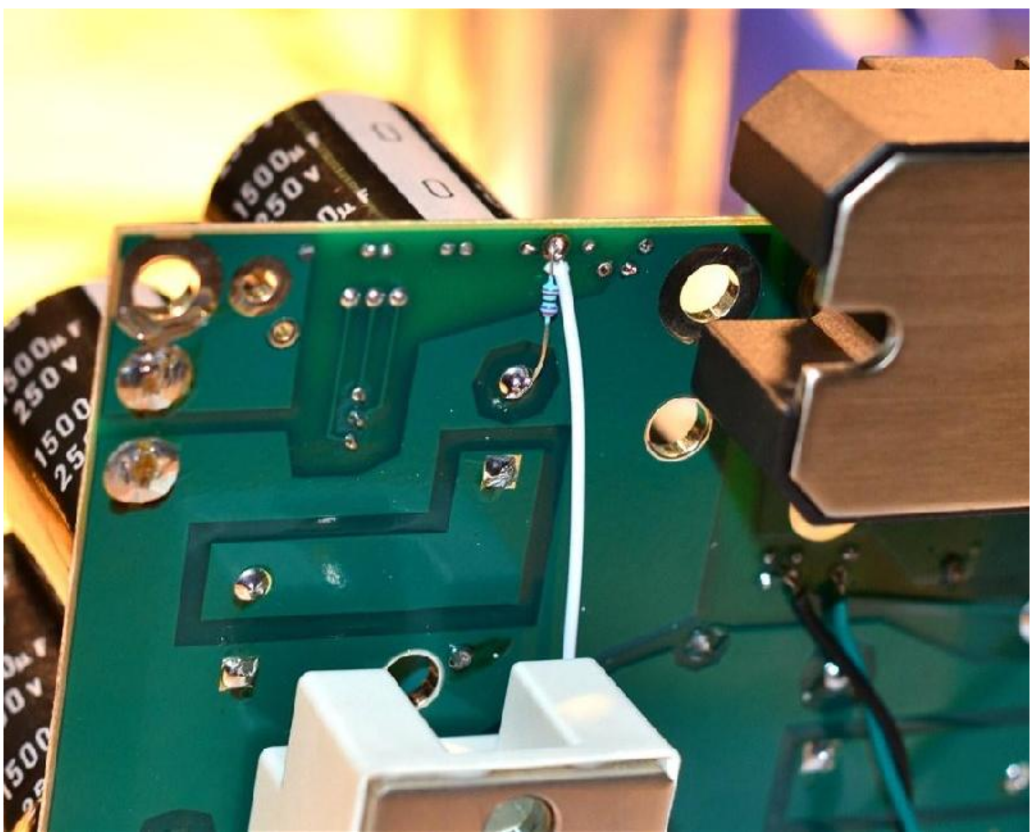


        i. 2.7k (R1, R3) - bottom of the board
        ii. 90-120R (R2) - top center

        iii. 200k 2W (R4) - bottom right
    b. Capacitors
        i. 10nF (C19) - top center. Note: PCB is marked with 2 C19's. Both are 0.01uF
        ii. x2 10nF - unmarked on the board, installed next to R1 and R3 in parallel with resistors. Use pads to the right of R1 and the ones to the left of R3 to install. Confirm by tracing the board. Next version of the PCB will have these capacitors marked on the board
    c. 100A Hall sensor: U1 - top right
    d. 3-pin male header 0.1" pitch - above the sensor, on the boards edge
    e. Hold components with a carton, turn over board, solder

2. Place Divider Resistors and Caps
    a. Top Left - dividers for input voltage sensing
        i. D2 : 600-1,000V rectifier diode, 1N4005 (600V) or 1N4007 (1,000V)
        ii. R53 : 200k / 2W
        iii. R5, R16 : 27k
        iv. R6, R7 - vary with the input voltage supplied
            1. up to 300VAC: 1M each
            2. 300VAC-600VAC: x2 1M each
        v. C5 : 0.1uF
        vi. C20 : 1uF
    b. Top Right - Resistor divider for Vout sensing
        i. Upper half of the divider
            1. R8 - varies with output voltage rating of the charger:
                a. up to 300V: insert a jumper (0R)
                b. 300V-450V: 1M
                c. 450V-900V: x2 1M in series
            2. R9 - varies with output voltage rating
                a. up to 600V: 1M
                b. 600V-900V: x2 1M in series
            3. R10 - varies with output voltage rating
                a. up to 750V: 1M
                b. 750V-875V: x2 1M in series
            4. Remember the resistor composition - you will need to enter the total resistance you installed into the firmware (upperR0_bV variable circa line 125)
        ii. Lower half of the divider
            1. VERY IMPORTANT: there is a 'bug' on the V14 Power PCB - R11 / C19 landing positions are INCORRECT. Installing these components into the pads marked on the PCB will result in failure of the charger. Instead, these components need to be installed between Vout pad and the top (negative) pad of the large C15 cap - see photo below



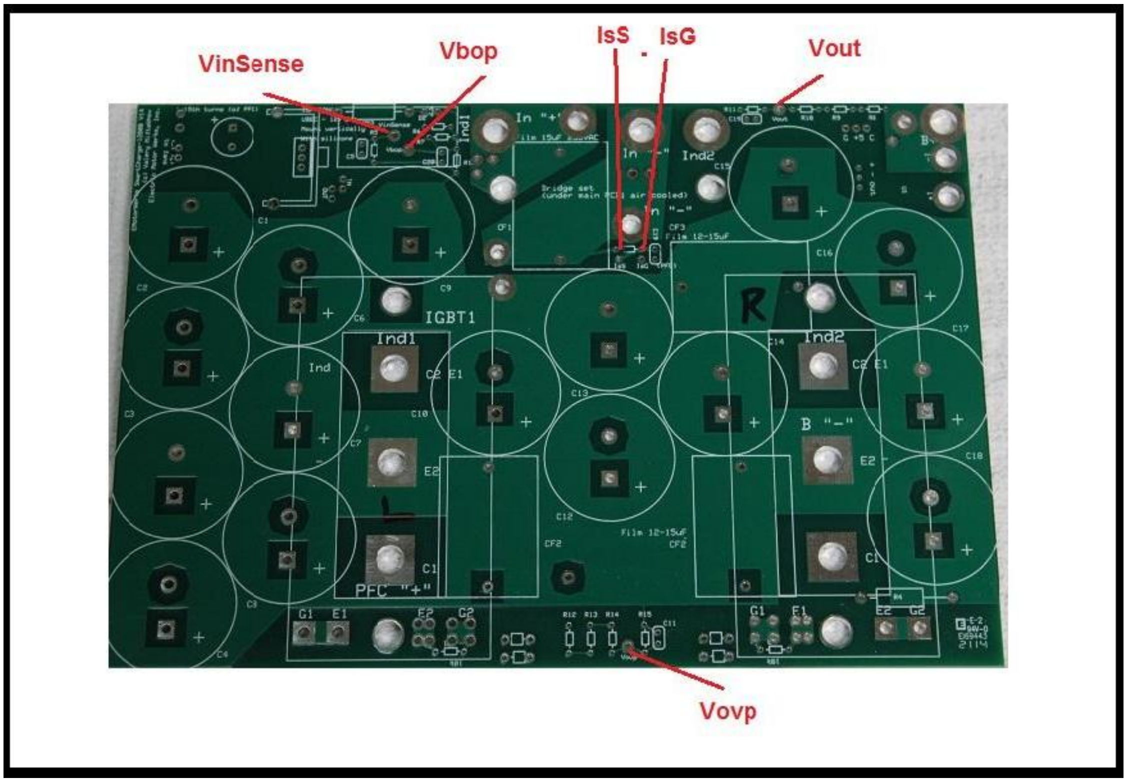


2. R11 : 2.7k (lower half of the resistor divider for Vout sensing)

3. C19 : 0.01uF (Note: PCB is marked with 2 C19’s. Both are 0.01uF)

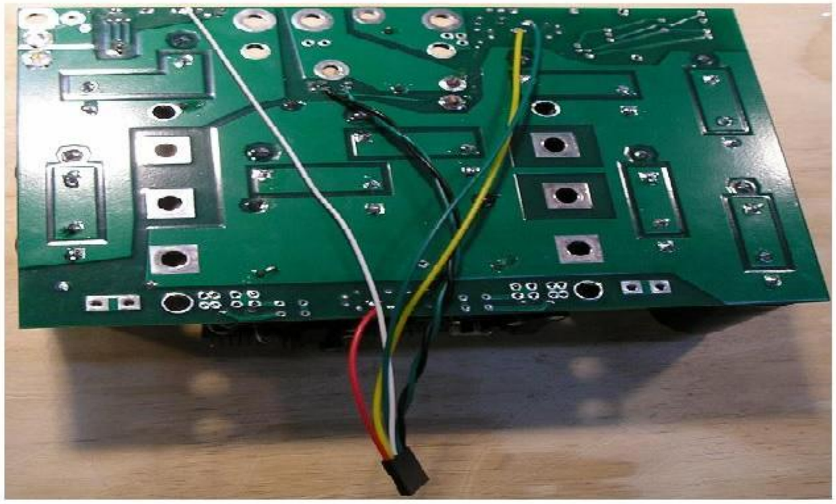
c. Bottom Center - Resistor divider for Vovp sensing. This divider is used to set the PFC rail voltage so it is VERY important to install the right amount of resistors here. Failure to do so may result in catastrophic failure of the charger.
    i. Upper half - The resistance varies with the expected output voltage of the charger
        1. R12
            a. up to 350V: insert a jumper. THIS IS VERY IMPORTANT IF YOU HAVE 600V IGBTS
            b. 350V-525V: 1M - NOTE THAT THIS WILL RESULT IN PFC RAIL VOLTAGE OF ~560V AND WILL REQUIRE INSTALLATION OF 1,200V IGBTs!
            c. 525V-875V: x2 1M in series
        2. R13
            a. up to 725V: 1M
            b. 725V-875V: x2 1M in series
        3. R14 : 1M
    ii. R15 : 27k (lower half of the resistor divider for Vovp sensing
    iii. C11 : 0.1uF

3. Create and Solder a 6-conductor driver board harness:
    1. There is no connector on the power board for the harness. 6 Individual wires are soldered directly to the power board and then soldered to a 6-pin female connector (0.1” female pin header)



2. Power Board through hole signal names and corresponding header connector pin number, wire color and wire length.
    i. Vovp -> pin 1 : red (3” inches long)
    ii. Vbop -> pin 2 : yellow (9”)
    iii. Vout -> pin 3 : white (9”)
    iv. VinSense -> pin 4 : green (9”)
    v. IsS -> pin 5 : green (7”)
    vi. IsG -> pin 6 : black (7”)

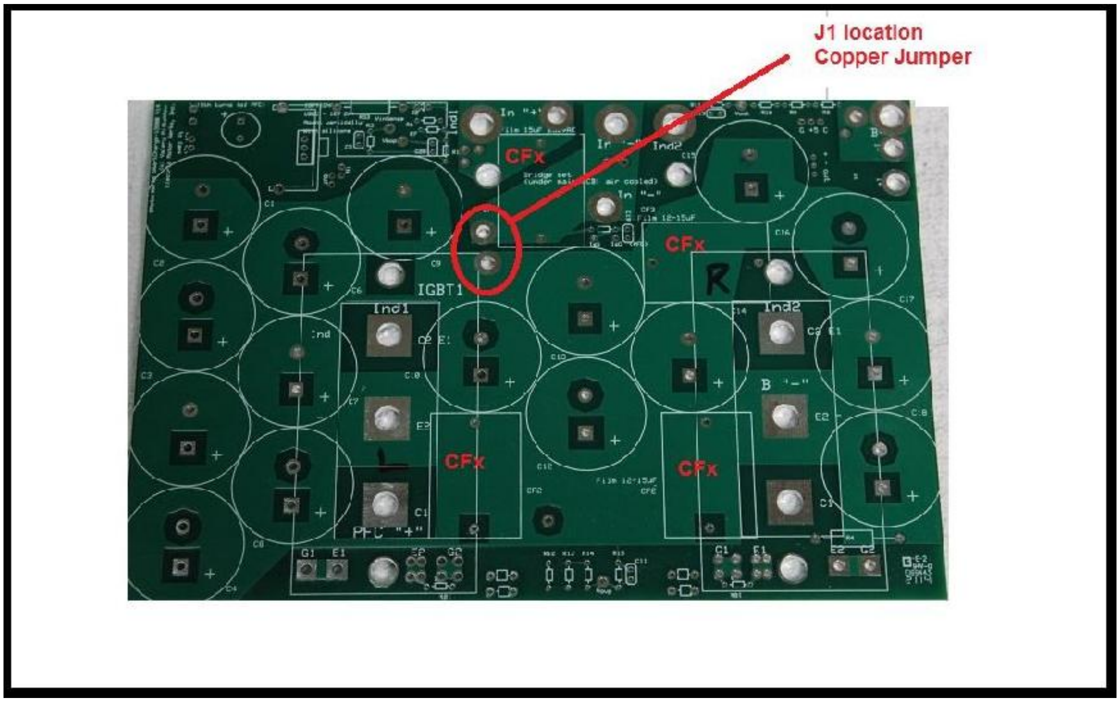
3. On the power board, solder the respective wire using the wire color and length above. Solder each wire to the respective power board signal pad. Route wires to enter from the lower side of the board. For example, solder a 3” red wire to ‘Vovp’ and solder the other end to pin 1 of the female connector (not on the inside). Do similar for the remaining 5 signals. Make sure to twist together the IsS and IsG wires.



6. Solder small AC wires to the bridge input locations on the power board (to be later used to connect to AC side of the 12V adapter). Do this ONLY if your input voltage is below 260VAC. If your input voltage is above that, you will have to either provide a separate 100-260VAC input to the 12V AC adapter OR provide an independent 12V 4A supply to the charger
    1. 2x 6” AWG22 wire or similar
    2. One wire goes to the In ‘-‘ plane
    3. Another wire goes to the In ‘+’ plane
    4. Twist wires

7. Place first set of large components:
    1. Connect the negative copper planes on the power board
        i. General area: ~1” up and left from the board’s center, between IGBT1 and CF1
        ii. Using a short piece of gauge 10/12 bare 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) with jumper (denoted as R5 on the schematics).
        iii. Bend wire to fit in provided pads, solder using a 100W+ soldering gun
    2. 4 rectangular film caps: CF1, CF2, CF3, CF4. Note PCB has 2 CF2’s. Picture below shows all 4 marked with ‘CFx’. The types of caps installed vary with the input and output voltage of the charger. First, calculate max(1.4 * input voltage, output voltage). Let’s call that Vmax
        i. Vmax < 450V: x4 20uF 450V caps
        ii. Vmax > 450V:
            1. CF1

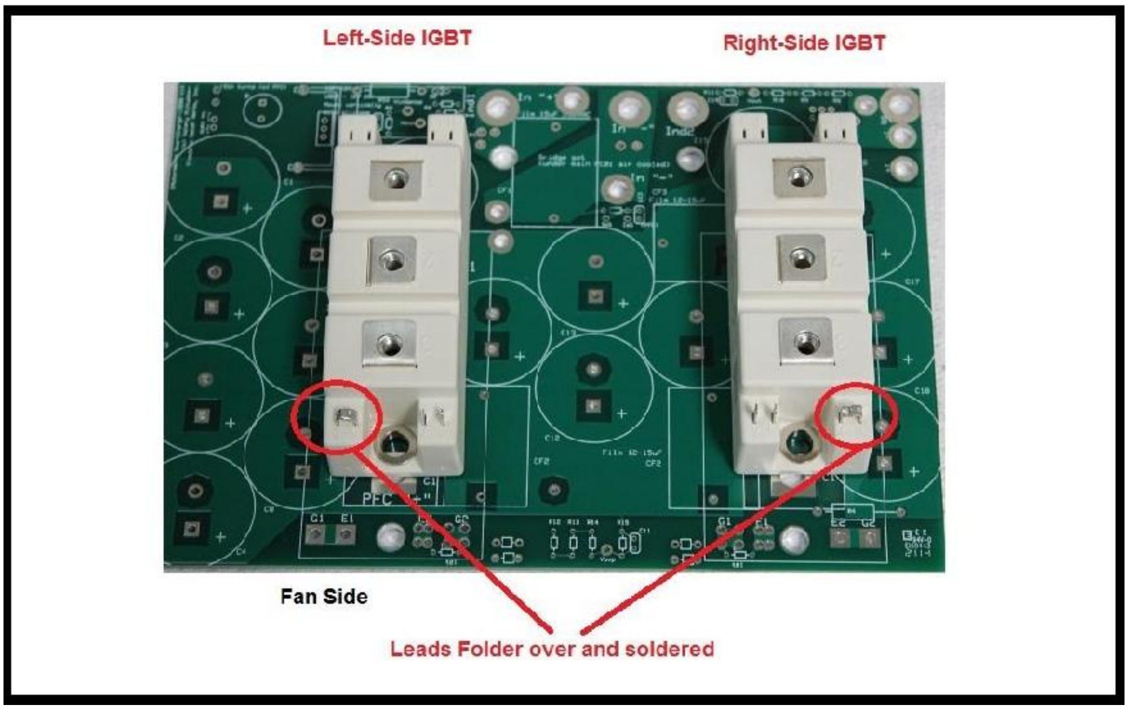
a. input voltage < 260VAC: 20uF 450V
b. input voltage > 260VAC: 12uF 800V
2. CF2, CF3, CF4: 12uF 800V
3. Hold components, turn over the board, solder (helps to slightly bend the caps' leads so they don't fall out when you turning the board over
4. Photo of Jumper and caps:



8. Solder large bulk Elcaps (Second set of large components)
   4. 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 '+') - dotted area means negative connection. All the caps should be oriented the same way.
   5. See picture and the PCB for the location of the caps. It is the 16 large circles. Starting from the top left corner, moving to the right and ending in the bottom right corner, the PCB circles are marked : C1, C2, C3, C4, C6, C7, C8, C9, C10, C12, C13, C14, C15, C16, C17, C18 (Note : C5, C11 NOT in the seq).
   6. The types of caps will depend on the output voltage rating of the charger - all arranged as 8 in parallel pairs of 2 in series
      i. 350V version (default)
         1. up to 240VAC input (single or 3-phase)
         2. 2200uF, 200V caps
      ii. 450V version
         1. suitable for CHAdeMO output
         2. up to 240VAC input (single or 3-phase)
         3. 1500uF, 250V caps

iii. 650V version
   1. up to 415VAC input (single or 3-phase)
   2. 1000uF, 350V caps
iv. 850V version
   1. up to 600VAC input (single or 3-phase)
   2. 560uF, 450V caps

9. Prepare the IGBTs
   1. If IGBT pins have metal ring ‘keepers’, these can be discarded.
   2. Solder together the E2 and G2 pins of the right-side IGBT (these are the 2 pins on the right side of the IGBT)
   3. Solder together the E1 and G1 pins of the left-side IGBT (these are the 2 pins on the left side of the IGBT)
   4. Picture of IGBTs with leads folded over and soldered



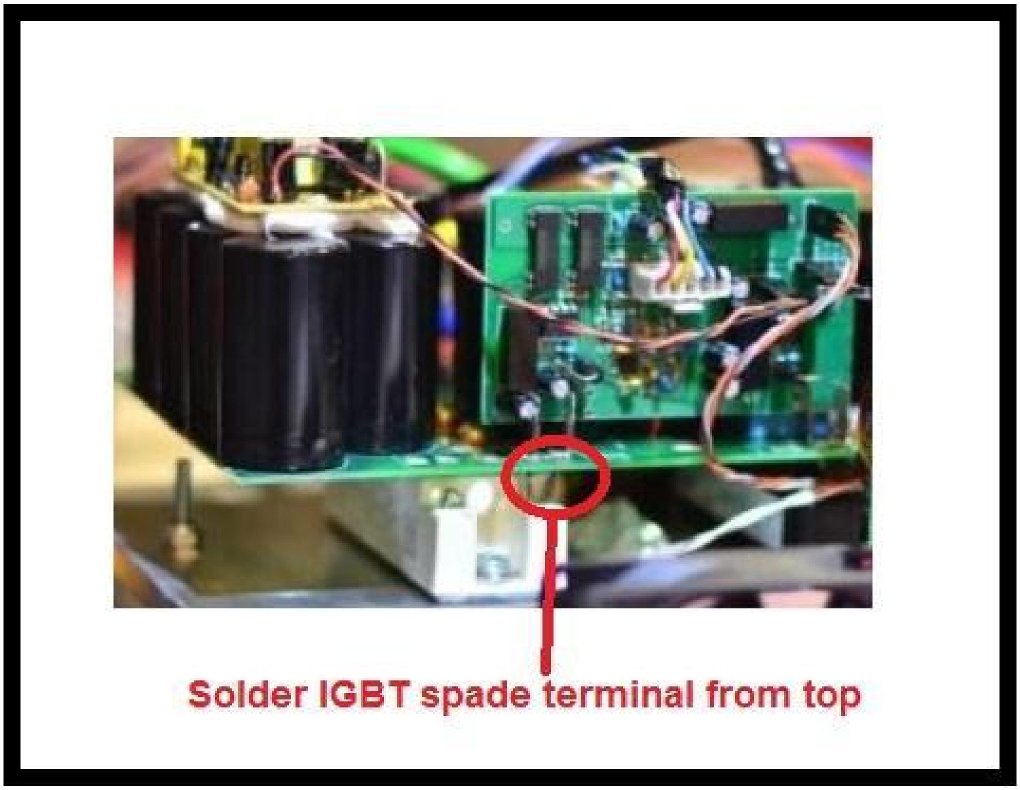
   5. Snip the top 2mm of the remaining IGBT pins

6. Fit 2 of the supplied female spade connectors on the E1 / G1 pins of the right-side IGBT
    i. orient connectors so that mounting pins of the two adjacent connectors are facing away from each other (see picture)

7. Fit another set of 2 female spade connectors onto the E2 / G2 pins of the left-side IGBT
    i. same orientation of connectors as for the right-side IGBT

8. 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 supplied by us - dependent on the type of IGBT shipped

10. 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.
    a. Picture of Power Board mounted on top of the IGBTs



11. Secure the board to IGBTs' top terminals only (marked 'C2 E1' on the board) using ½" 10-32 machine screw (from an optional SmartCharge-12000 [parts kit](#)); solder the spade connector pins from the top. Note that it takes a lot of heat to make a good solder joint for these connections - avoid 'cold-solder' joints. Make sure not to get solder in any of the other PCB holes as they will be used later on to mount the driver board onto the power board.

12. Apply silicone sealant between all caps to protect against vibration. See picture below.

13. Use x3 1/2" 10-32 machine screws and x3 2 inch 10-32 male-female standoffs (½" thread length; standoffs can be obtained as part of the SmartCharge-12000 parts kit) to secure power board to IGBTs, placed according to orientation on picture below

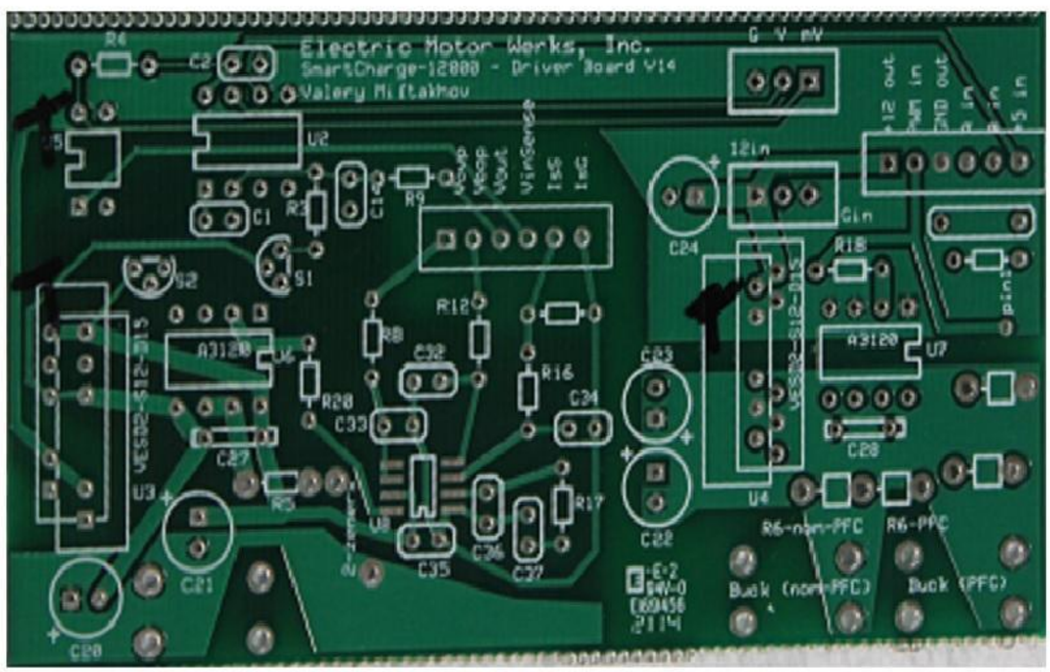
a. C2E1 of the left-side IGBT and C2E1 & E2 of the right-side IGBT get the standoffs (it is possible to do everything without the standoffs - at the expense of a bit harder wire routing)
b. Do not tighten screws; some space for movement will be helpful later on when mounting IGBTs to heatsink (you may need to slightly change the alignment of the IGBTs in order to align them with the heatsink mounting locations later).

14. Here's how the board should look (approximately - Vovp divider shown is configured for 525V output / 550V PFC rail) after you are done:

Part 4. 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. As you assemble the board, make sure to orient all ICs’ correctly. Make sure pin 1 of a chip mates to pin 1 of the board.
3. Several builds were hurt by incorrect orientation of ICs.



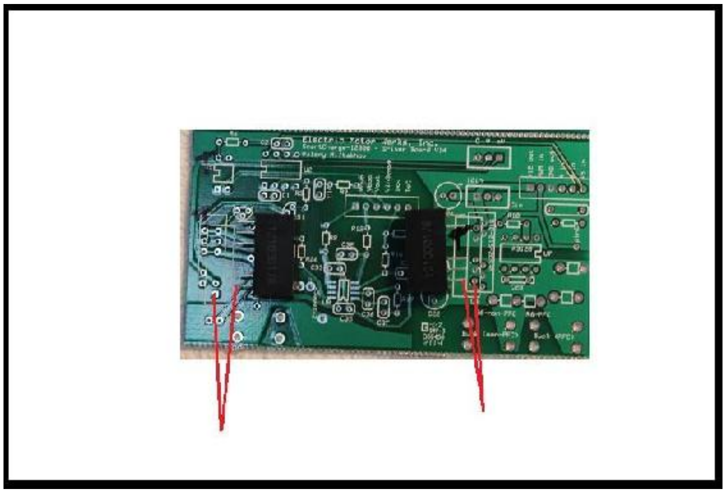
Assembly sequence – current board type

1. Group 1: (lowest-height components first) - OBSERVE IC ORIENTATION!
    a. Place IR1153 PFC surface-mount chip U8, solder
    b. Place A7520 chip U2
    c. Place PC817 (opto-isolator) U5
    d. Place U6, U7 : A3120 / 3184 / J312 chips (2)
    e. Turn over, solder, trim long leads

2. Group 2:
    a. Place 68k resistor R3
    b. Place 2.7k resistor R4
    c. Place 220R resistor R18
    d. Place 120R resistor R16
    e. Place 3.3k resistor R17
    f. Place 1k resistor R20
    g. Place 1uF caps C32, C35
    h. Place 100pF cap C33
    i. Place 0.1 uF caps C1, C2, C27, C28, C34, C14
        i. C14 is obscured on PCB. It's to the right of R3
    j. Place 0.01uF cap C36
    k. Place 3.3uF ceramic cap C37
    l. Place -5V (NEGATIVE -5V) voltage regulator S1 (79L05)
        i. Be VERY careful - 79L05 and 78L05 (next item below) look IDENTICAL - the only way to tell them apart is to look at the label on the device - for which you will most likely need a magnification glass!
        ii. Mixing up these two components during assembly will most likely result in charger failure
    m. Place 5V voltage regulator S2 (78L05)
    n. Insert x2 3-pin & x2 6-pin male pin headers on the component side.
        i. 3-pin : G, V, mV
        ii. 3-pin: 12in, NC, GND
        iii. 6-pin: +12out, PWM in, PWM out, A_In, B_In +5in
        iv. 6-pin: Vovp, Vbop, Vout, VinSinse, IsS, IsG
    o. Hold parts with carton/plastic, turn over, solder, trim leads

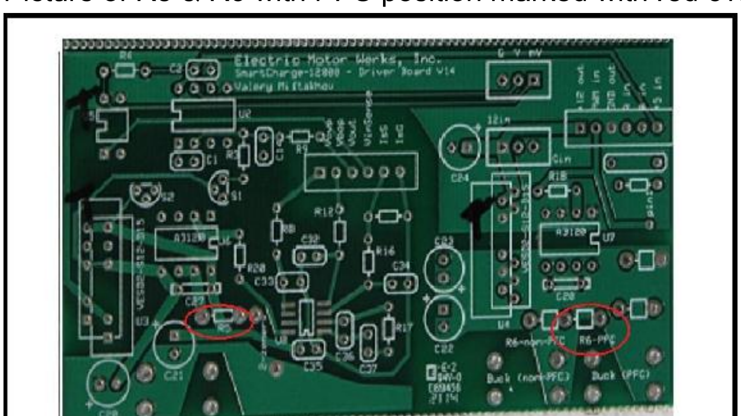
3. Group 3:
    a. Place 2x 12V->15V DC-DC converters U3, U4

    b. There are TWO silkscreen footprints for each converter. Place each converter into the footprint matching the pin layout.
    c. Picture of Converters and Footprints



d. Hold the tops of the DC-DC converters with a piece of carton/plastic and turn the board over; solder all components and trim leads

4. Group 4:
   a. Place 5x 47uF 35V elcaps C20, C21, C22, C23, C24.
      i. These are polarized! Make sure + of cap goes to + on PCB and - of cap goes to - on PCB. See cap Polarization Marking for more info.
   b. Place 18v zener diodes D1, D2 (bottom right corner). The PCB is not marked with the polarity. White band on diode goes on the left for D1(bottom). White band on diode goes to the right for D2 (top). See diode Polarity Marking for more info. The objective is to install these back-to-back
   c. Place 5-10R power resistors R5 & R6 (observe board markings for the correct location of R6 for the PFC version)
   d. Picture of R5 & R6 with PFC position marked with red oval



PFC/Buck Driver R5, R6

   e. Turn over, solder, trim leads
5. Group 5:
   a. R9 : 120R
   b. R8 :120R
   c. R12 : 120R

d. Place 2 more 18v zeners D3, D4 as shown in picture.
    i. Place on PCB marked ‘2-Zeners’
    ii. Insert lead into PCB with white band next to the board for both D3 and D4
    iii. Solder D3 white band lead to PCB
    iv. Solder D4 white band lead to PCB - the objective is to install these back-to-back
    v. Twist ‘in-air’ leads together, solder and trim

5. Install x4 male 90-degree spade connectors into the driver board from the component side. Solder while trying to ensure all connectors are fixed at 90 degrees to the PCB

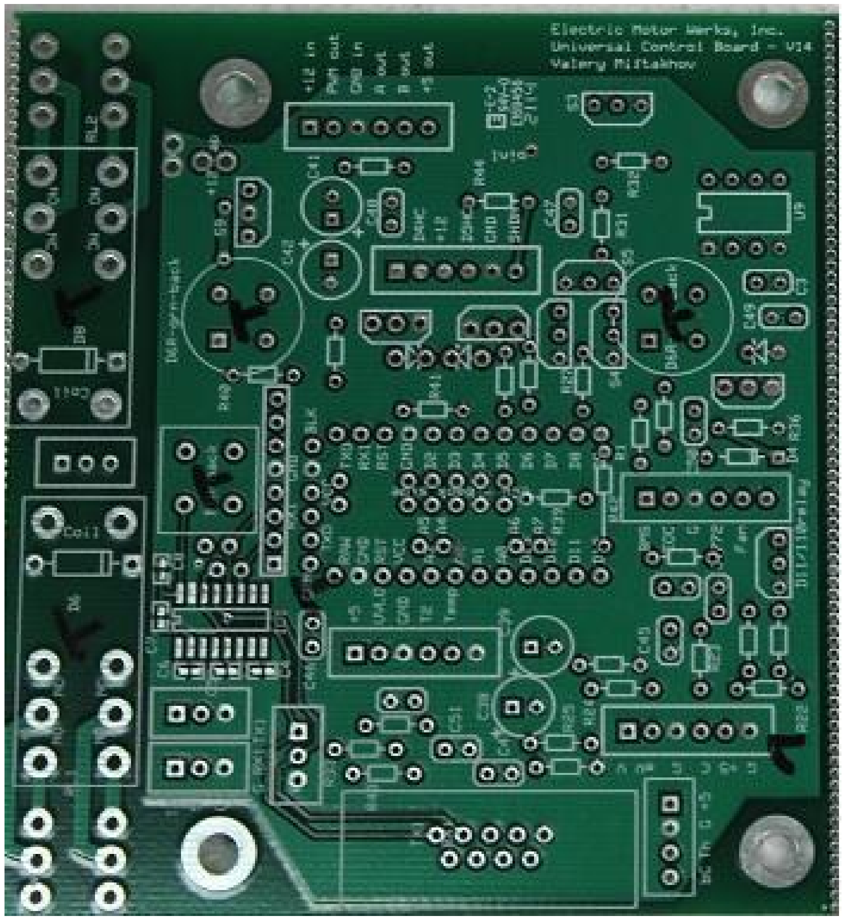
6. Fit x4 female spade terminales on top of the 90-degree male connectors
7. Insert the driver board with female spade terminals into the power board

8. NOTE orientation - component side of the board should face OUT (for PFC version)
9. Turn over the power+driver board assembly, solder female spade terminals to the power board (from the bottom of the power board). Be especially careful not to produce cold solder joints!

Part 5. 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
    a. Be 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
    b. Do not forget A4 & A5 pins on the inside of the board

2. Place Group 1 components:
    a. 1k resistors R22, R23, R24, R25, R27, R31, R32, R36
        i. If any of these IDs are not printed on the board - Ctrl-F in ExpressPCB software to find part on the board)
    b. 10k resistors R1, R39, R40, R41, R42 (immediately under R39)
    c. 100k resistor R48
    e. 220R resistor R44
    f. LM211P chip U9
    g. Turn over, solder, cut leads

3. Place Group 2:

a. 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. Pin 1 is the first (leftmost) item in each list below.
    i. 6-pin under the Arduino:
        1. BMS, EOC, G, J1772, FAN, NC
    ii. 2x 6-pin on the sides of the board
        1. +12_in, PWM_out, GND_in, A_out, B_out, +5_out
        2. V, mV, G, C, +5, G
    iii. 2x 6-pin on the left and right of Arduino
        1. +5, UVLO, GND, T2, TEMP, NC
        2. NC, D4HC, +12, DSHC, GND, SHDN
    iv. 8-pin on top of Arduino
        1. These are the FTDI-Bus programming pins
    v. 2x 2-pin
        1. Both 2-pin headers located at left, above 8-pin FTDI-Bus header and to the left of ‘BTN-back’ silkscreen outline for the programming button
        2. Used for 2 jumpers
    vi. 2x 3-pin External power headers
        1. 12, 5, G (marked on PCB)
        2. Not marked on PCB but immediately to right of header above
b. Hold pins with a piece of carton or plastic, turn over, solder

4. Place Group 3:
    a. Small signal diode D4
    b. N2222 transistors S3, S4, S5
    c. 0.1uF caps C43, C45, C46, C47, C48
    d. 10uF, 50v caps C38, C39
    e. 0.1uF C3
    f. 330uF 16V-25V cap C41
    g. 330uF 16V-25V cap C42
    h. 5V voltage regulator (LM7805) S9
    i. 0.01uF caps C49, C50
    j. Turn over, solder, cut leads

6. Make the Arduino socket
    a. Attach female pin headers (2x 12-pin, 6-pin, and 2-pin (latter made out of a 3 or 4-pin header - just pull out excessive pins from the header) to Arduino male pins
    b. Insert the assembly into the control board
    c. Turn over and solder
    d. Pull out Arduino board from this new connector

7. FROM THE BACK SIDE, place buttons ‘D6R-red’, ‘D6R-green’ (or black), and ‘BTN-back’
    a. Flat indentations in the button bodies should align parallel to the shorter side of the control board. If unsure, measure resistance of the button in on and off position to ensure correct placement
    b. Turn over, solder

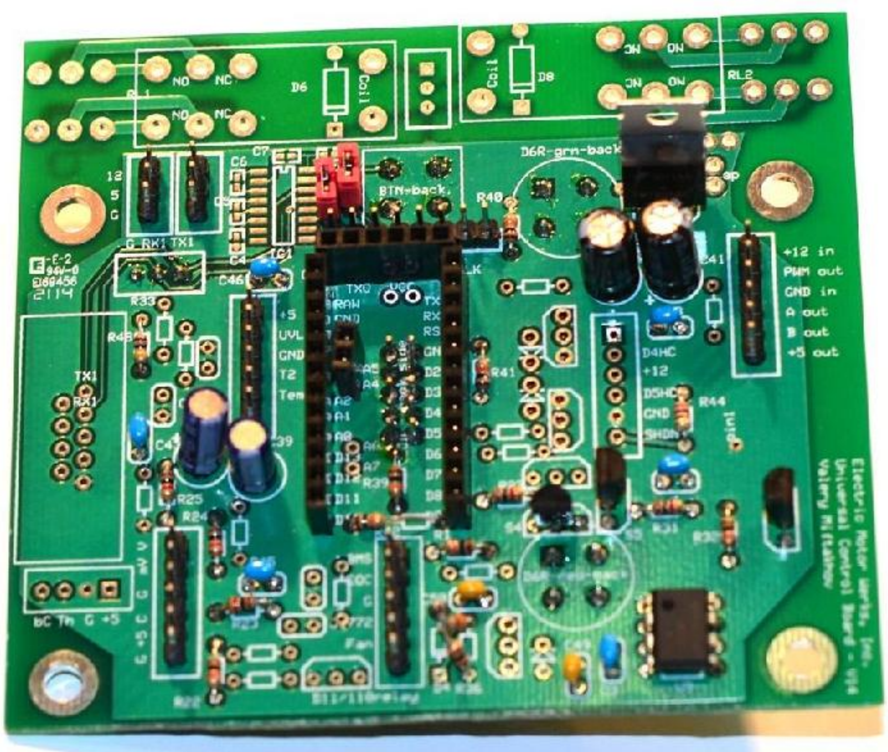
8. Using non-conductive glue, attach a piece of isolating pad to the back of the LCD board (or just place a piece of electrical tape or duct tape on the back of LCD board)

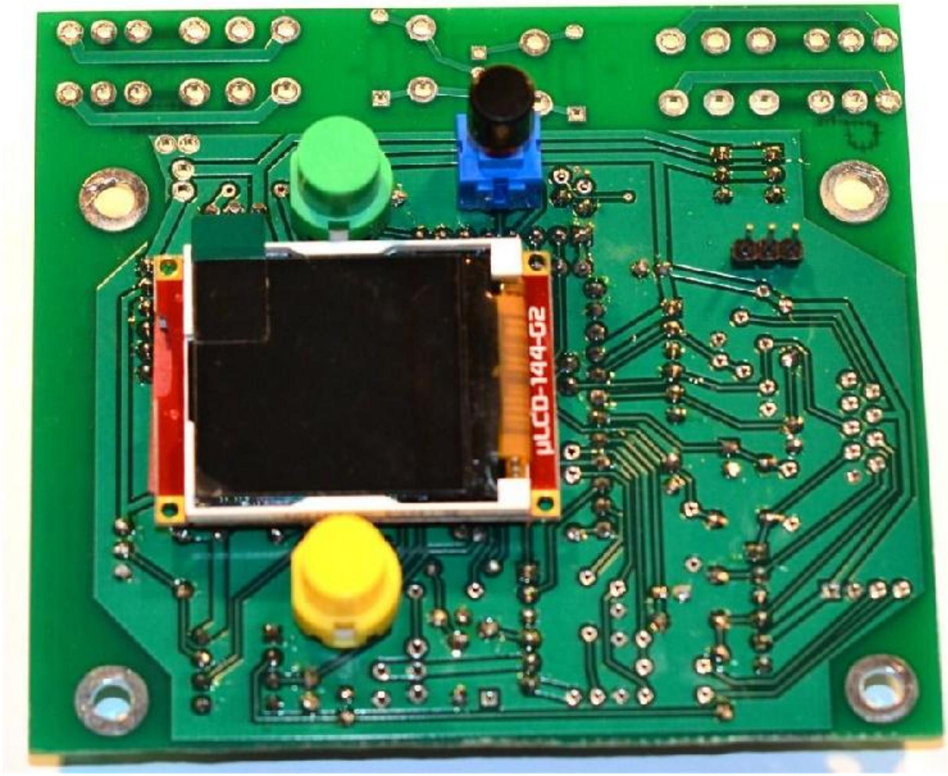
9. FROM THE BACK SIDE of the control board
    a. Place the resulting isolated LCD board through the pins in the middle of Arduino mounting area.
    b. [optional - only if you plan to control your charger via Serial UART interface) Place 3-pin header into TXI, RXI, G
    c. Solder from the top of the board

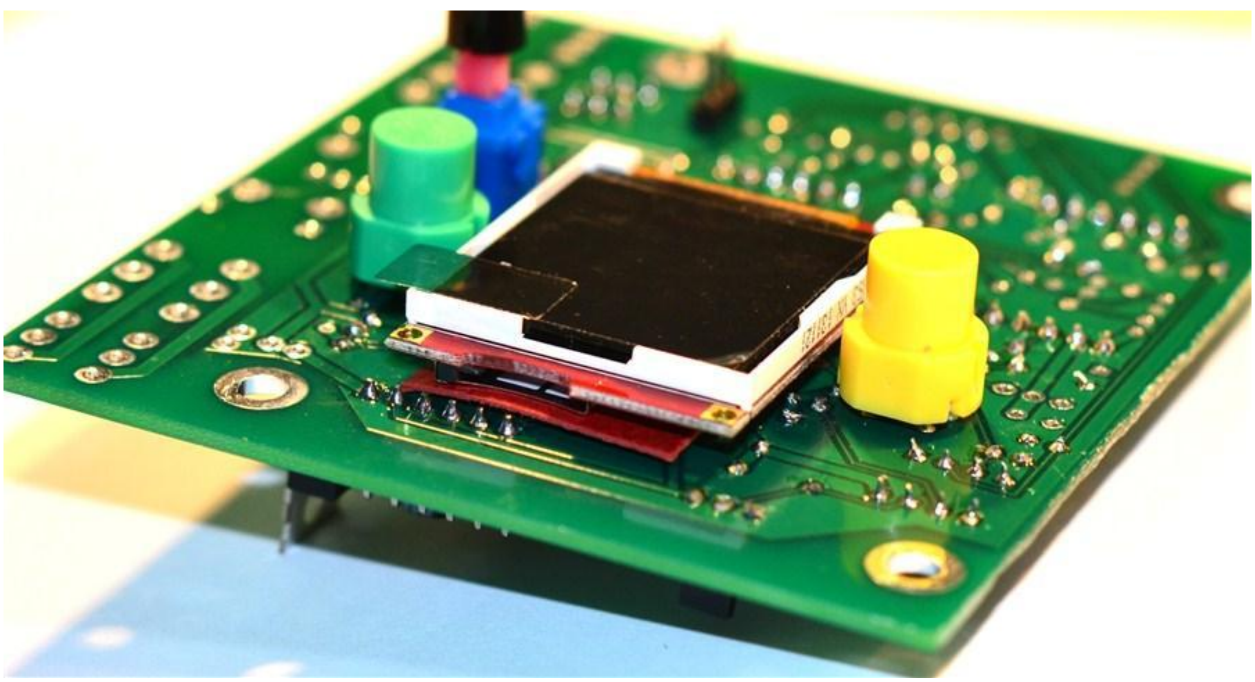
10. Reinsert Arduino board into the socket

11. Install vertical jumpers on each 2-pin jumper header

12. Here’s how the board should look after you are done:




µLCD-144-G2

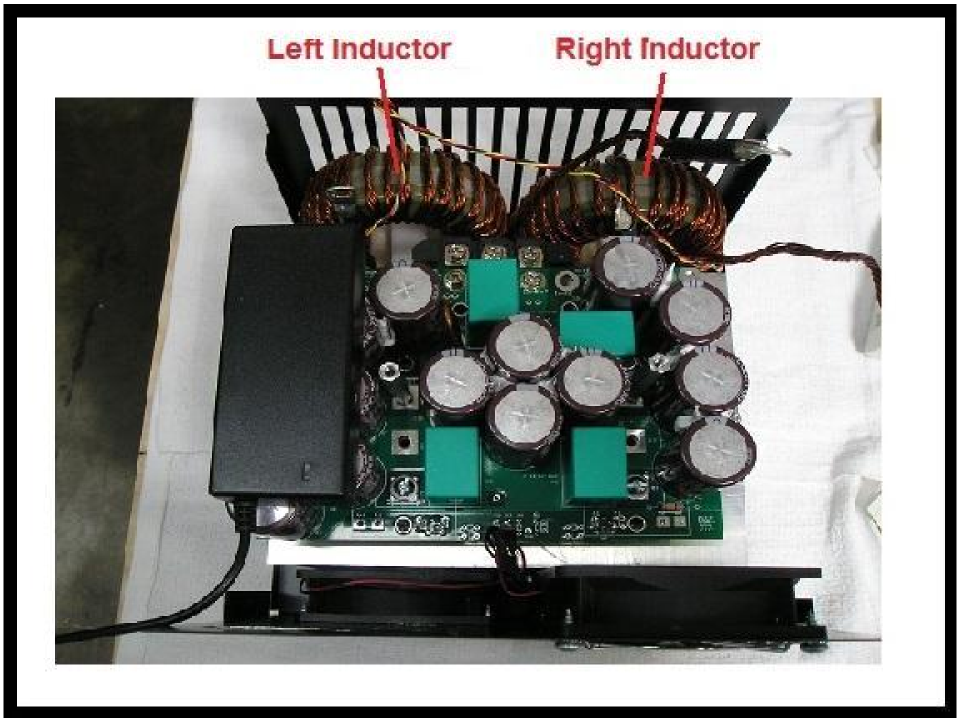


Part 6. Final Assembly

1. **First Steps**
    a. If you haven't completed the steps at Part 2. Marking up the Heatsink, please complete those steps first

2. **Prepare the enclosure & fans:**
    a. For standard (non-machined) enclosure, refer to the photos of the machined enclosure to get the idea for how to fabricate
    b. For the machined enclosure:
        i. Orient the bottom half of the enclosure with round fan openings towards you
        ii. Use supplied fans and grills to mount fans (will need x12 #10 ½" sheet metal screws or similar). Note that these parts are supplied only if you have ordered an optional enclosure from us
        iii. Prepare fan connection:
            1. Solder together fan leads in parallel (positive to positive, negative to negative)
            2. Solder a 2-pin female header to the resulting 2 sets of 3 paralleled wires (use 3 pin header with the middle position left vacant)
            3. 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

2. Prepare and Mount the inductors:
    a. Picture of inductor's installed location in chassis

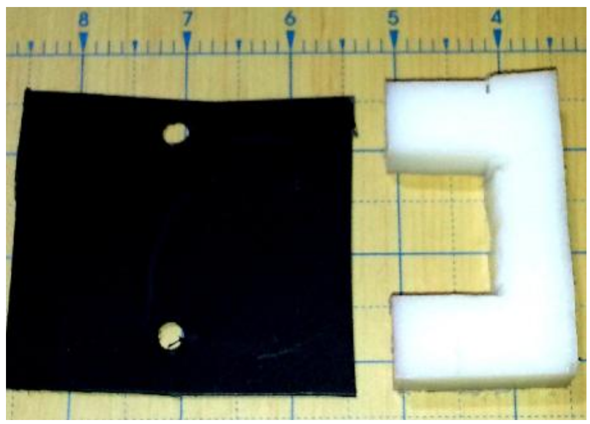


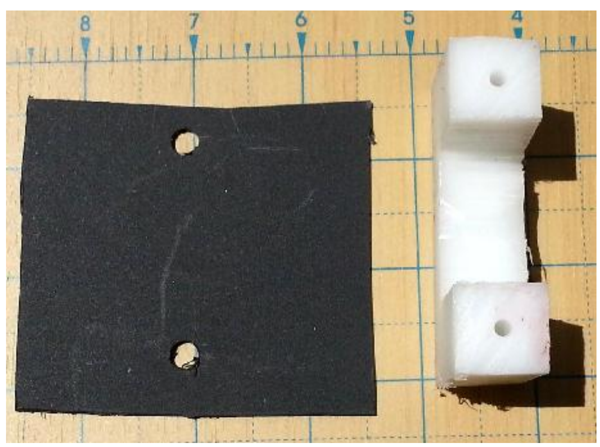
b. Prepare the inductors:
   i. Make sure that the inductors have their leads exiting from one side
   ii. Make sure that there is a ¾” clearance between the two ends of the single-layer winding. This clearance will be used by the mounting bracket

b. Prepare mounting brackets:
   i. Using ¾”-1” thick HDPE or similar material, cut a C-shape as shown below to allow the inductor core to come through; Dimensions: 2 ¾” – 3” long by 1 ½” wide material with 1 ½” cut length ¾” into material from the side
   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 insulating 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)

c. Mount the inductors:
   i. Use ¾”-1” thick HDPE or similar material
      1. Cut a C-shape as shown below to allow the inductor core to come through, Dimensions: 2 ¾” – 3” long by 1 ½” wide material with 1 ½” cut length ¾” into material from the side
      2. 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
      3. Cut a square pad from some insulating material (1/8” rubber is great here)

4. Drill out holes for the screws in the isolating material (matching holes you just drilled in the C-shape)
5. Mount the inductors:
    a. Use #10 sheet metal screws or similar, fender washers
    b. 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
    c. Normally, both inductors are identical so it does not matter which one goes where
    d. 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 a thicker wire bundle
    e. Inductor Mounting C-Bracket Reference Shots:

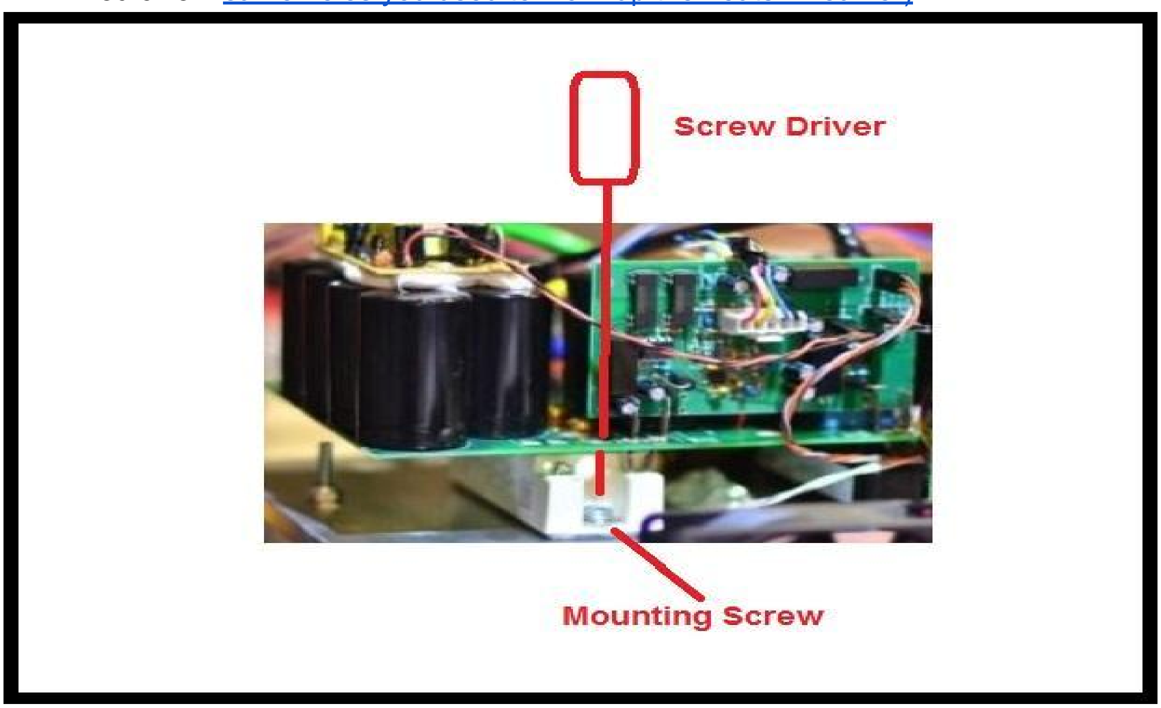




3. **Mount the control board & heatsink:**
    a. Use pre-drilled location above the fans on the left part of the enclosure
    b. Use 4x 1-1½" 8-32 machine screws, washers, and lock washers or lock nuts
        i. Use ⅜" nylon spacers or similar between the control PCB and the enclosure
    c. Orient PCB with LCD facing outside, center buttons in their respective holes
    d. Fasten all screws

e. Place the heatsink inside the enclosure:
    i. Flush to the fans (up to 1/16" spacing is fine)
    v. Using 3.5"-long #10 machine screw, #10 fender washers, and #10 locknuts, mount the sink to the enclosure
f. Connect thermistor to the control board:
    i. Use GND and Temp pins of the 5–pin connector below the Arduino board (looking at the mounted control board from the component side)

4. **Mount the power board to the heatsink:**
    a. Pre-set 6x ¾"-long #10 machine screws into IGBT / bridge mounting holes to make it easier to mount. Use lock washers and flat washers
    b. Lower the power board onto the heatsink, aligning mounting holes
    c. If output diode is used:
        i. Align the bare copper lead from the output diode so it can be fastened to the output pad on the power board
        ii. Fasten the diode lead to the output pad using #8-10 machine screw. Use a lock washer / lock nut
    d. 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)



    e. Fasten all screws!

5. **Connect inductors to the power board**
    a. Crimp AWG6 lug onto each lead of both inductors

b. Connect the left-side inductor (‘PFC’):
    i. Screw-mount one lead of the inductor to the Ind1 pad on the power board
    ii. Screw-mount the other lead to the top terminal of the left-side IGBT (marked C2 E1). 1.5”-2” brass / aluminum standoffs help clear the caps but not required
c. Connect the right-side inductor (‘output’ or ‘buck’):
    i. Screw-mount one lead to the Ind2 pad on the power board
    ii. Screw-mount the other lead to the top terminal of the right-side IGBT. Again, a 1.5”-2” brass / aluminum standoff help clear the caps but not required

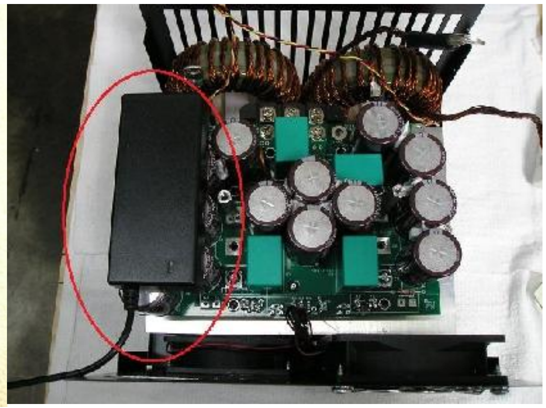
6. **Wire main charger power inputs**
    a. AC input wires are bolted directly to the integrated bridge module terminals
    b. In the single-phase application, only 2 input AC terminals are used. You can also use the same bridge with 3-phase input. However, the charger will NOT have Power Factor of 1.0 when connected to a 3-phase supply.
    c. Cut x2 (or x3 if 3-phase input is used) 12” of black AWG6 wiring, crimp a ring lug on one end of each wire and attach all the wires to the IDBM. These will be input AC wires
    d. Secure the terminals of the bridge with silicone to prevent movement of the bridge with respect to the power board and prevent un-fastening of any of the terminals

7. **Wire main charger power outputs:**
    a. Negative output:
        i. 12-18” AWG 6 black wire – connect to E2 of the output IGBT (right side IGBT)
        ii. Very helpful (but not required) to use a 1.5-2” brass / aluminum standoff here to elevate the mounting point off the board so the wire clears the caps easier
        iii. Use lock washers on the screws
    b. Positive output:
        i. If NOT using an output relay:
            1. Using 12” AWG 6 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)
            2. Use lock washers / lock nuts
        ii. If USING an output relay:
            1. Relay selection guidelines

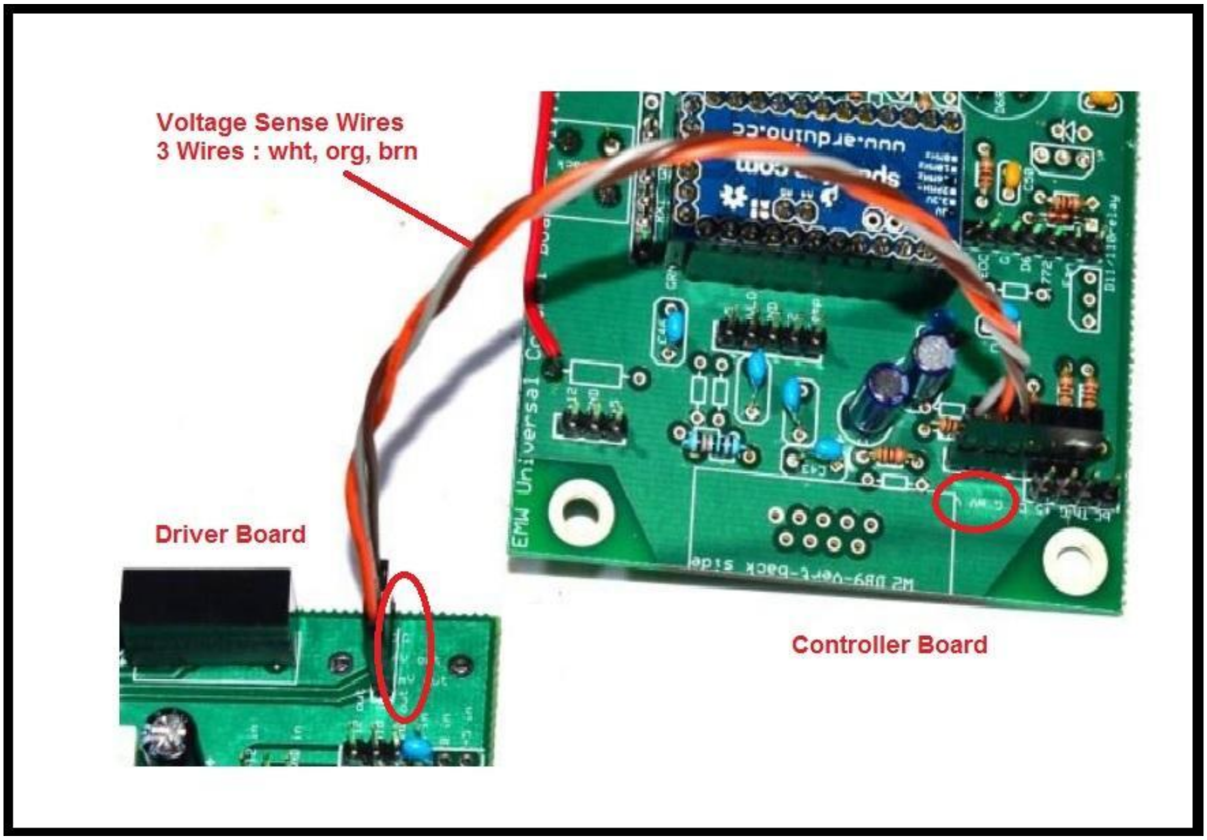
        a. Use relay rated for at least 240VAC, 60A (some 2-3-pole relays rated for 30A can be used if you parallel the poles together)
        b. Since you are not planning to break connection under current, you are ok with the AC relay
        c. You will need a 90-100A relay if you have opted for a 100A output configuration.
    2. Wire the coil connection of the relay to the control board (refer to the firmware for the right pin allocation - generally you need to connect one end of the relay to positive supply rail - 5 or 12V, and another - to D4HC)
    3. Connect the 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
    4. Wire output pad of the charger to the input of the relay (note that many DC relays / contactors will have defined polarity)
    5. Connect a 12" AWG 6 red wire to the output of the relay – this will be the positive output of the charger

7. **Route all power wires (input AC and output DC)**
    a. For enclosure versions shipped before September 2014: 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
    b. For enclosure versions shipped in September 2014 or later: there are a number of large openings above the cooling slots that can be used to pass wires.

8. **Connect 12V adapter to the driver board:**
    a. Cut the output wire of the AC adapter, leaving ~8-12" on output DC side
    b. Solder a 3-pin 0.1"-pitch female header to the output DC wires (use header positions 1 and 3). This will later be connected to the power input pins on the driver board
    c. Using silicone sealant, 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
    d. Set aside for 30 min to let sealant set
    e. Some pics:



9. **Prepare inter-PCB Connection Harnesses**
    a. Prepare signal connection harnesses – you will need a total of 4 required harnesses and 1 optional. Twist all wires tightly to improve noise immunity!
    b. Voltage sensing
        i. If using [EMW Stocked Part](), skip DIY Voltage sensing harness assembly instructions.
        ii. DIY Voltage sensing harness:
            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)
                a. V, mV, G
            3. 3-pin header will mate to the 3-pin set on top right of the driver board
                a. Note : pin 1 is on right; pin seq on board lft to rgt is 3,2,1
                b. G, V, mV
            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…)
                a. V <-> V : wht
                b. mV <-> mV : org
                c. G <-> G : brn
            5. Here’s a pic:



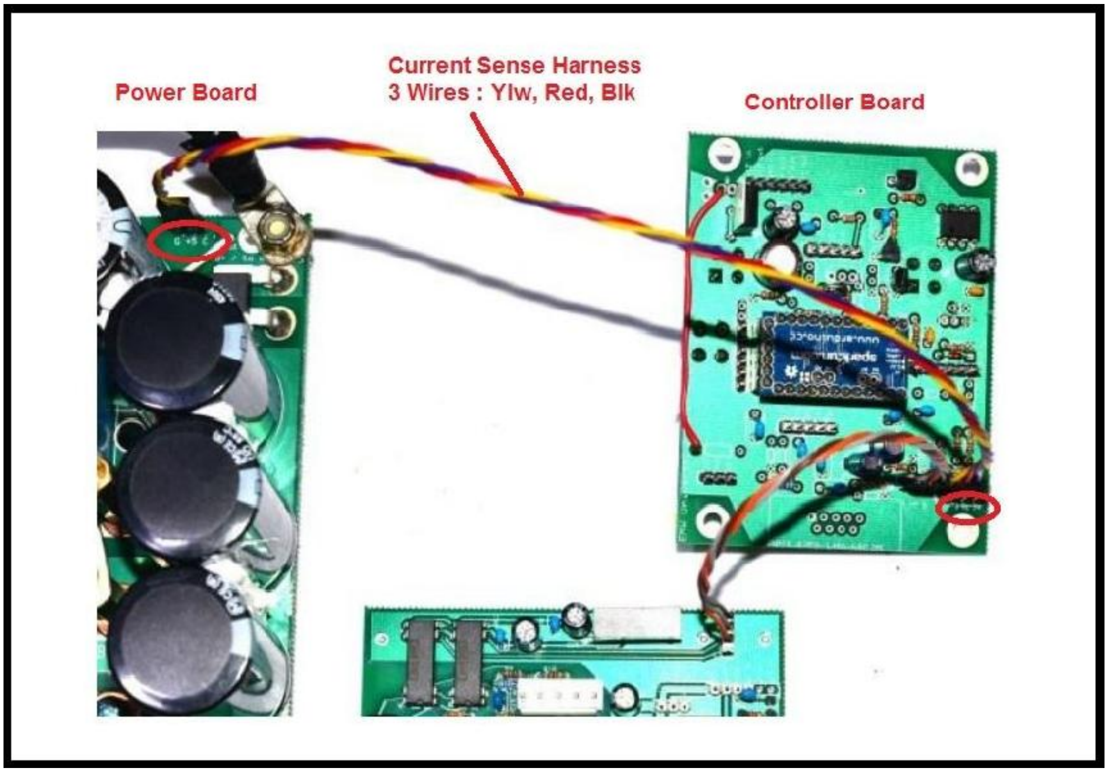
b. Current sensing:

i. If using EMotorWerks Hardware and Wiring Kit, you will be supplied with pre-crimped wiring and matching Pololu housings.

ii. Current sensing harness assembly

1. Need: x3 12” wires, x1 3-pin female header
2. 3-pin header will mate to the current sensing pins on top right of the power board
    a. G, +5, C
3. The other ends of the wires will go to the 6-pin header you used in the voltage sensing step above
    a. C, +5, G
4. Referring to the PCBs / PCB files, solder wires to the right position

    a. C <-> C : ylw
    b. +5 <-> +5 : red
    c. G <-> G : blk



c. PWM signal harness:

i. Need: 4x6" wires, 2x 6-pin headers

ii. One 6-pin header will mate to the 6-pin connector on top left of the **Control Board**. 6-pin connector signals are from pin 1, left to right.

1. +12_in, PWM_out, GND_in, A_out, B_out, +5_out

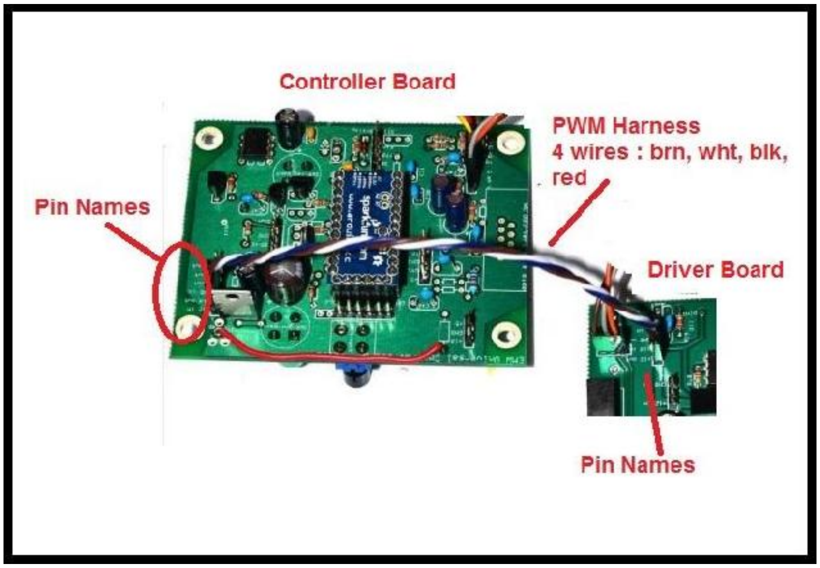
iii. Another 6-pin header will mate to the 6-pin connector on top right of the **Driver Board.** 6-pin connector signals are from pin 1, left to right.

1. +12_out, PWM_in, GND_out, A_in, B_in, +5_in.

iv. You are connecting all positions EXCEPT 'A', 'B'

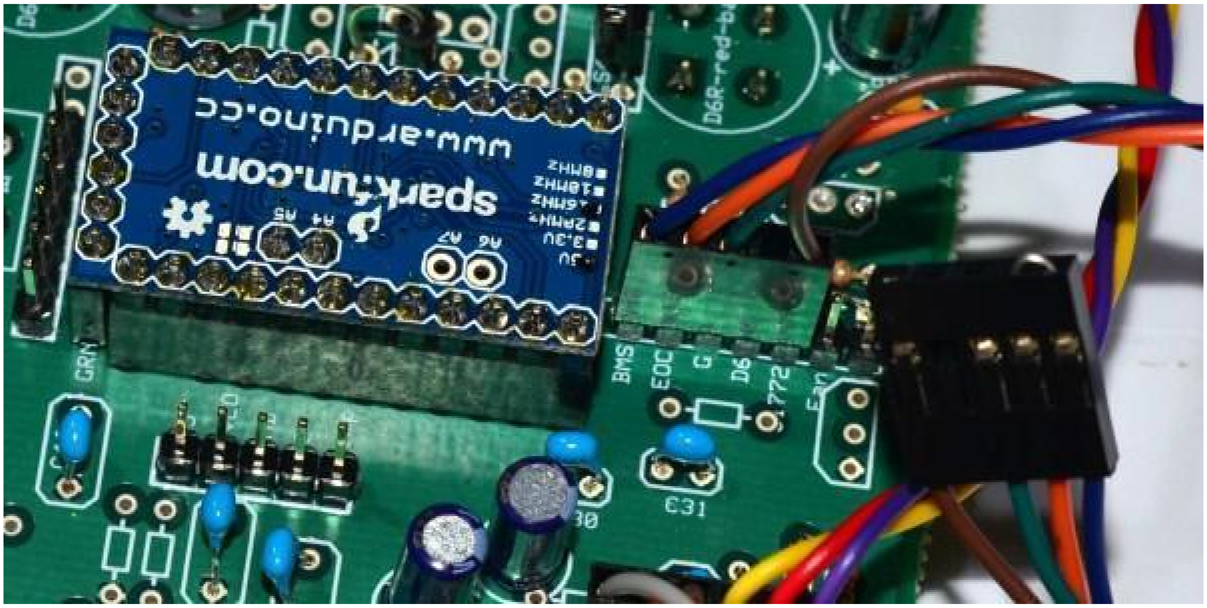
v. Connect positions from driver board to same positions on control board

1. +12_in <-> +12+_out : brn
2. PWM_out <-> PWM_in : wht
3. GND_in <-> GND_out : blk
4. +5_out <-> +5_in : red

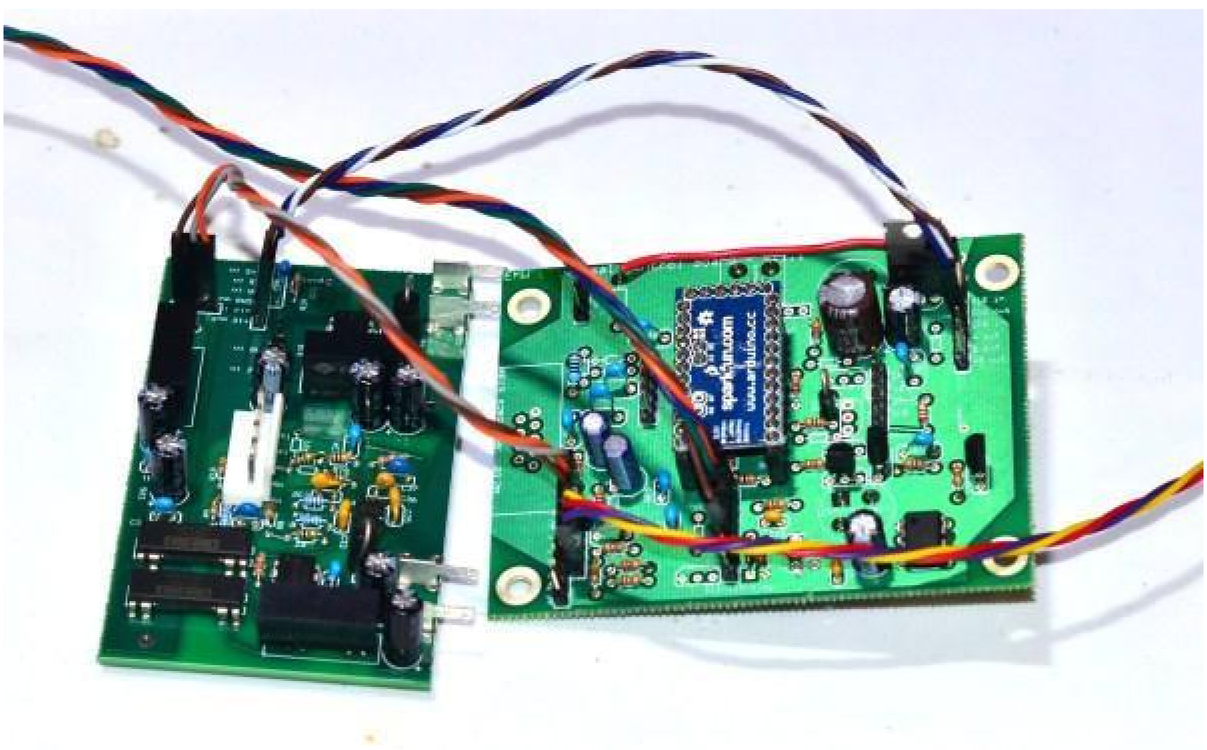


d. BMS / J1772 harness:

i. Need: x4 12” wires, x2 6-pin female headers
ii. One header will mate to the 6-pin connector on top of the Arduino in the above photo
iii. 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
iv. Connect wire between 2 female header connectors:
    1. BMS -> pin 1 : blu
    2. EOC -> pin 2 : org
    3. G -> pin 3 : grn
    4. J1772 -> pin 4 : brn
    5. pin 5 : no connect (NC)
    6. pin 6 : NC
v. 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



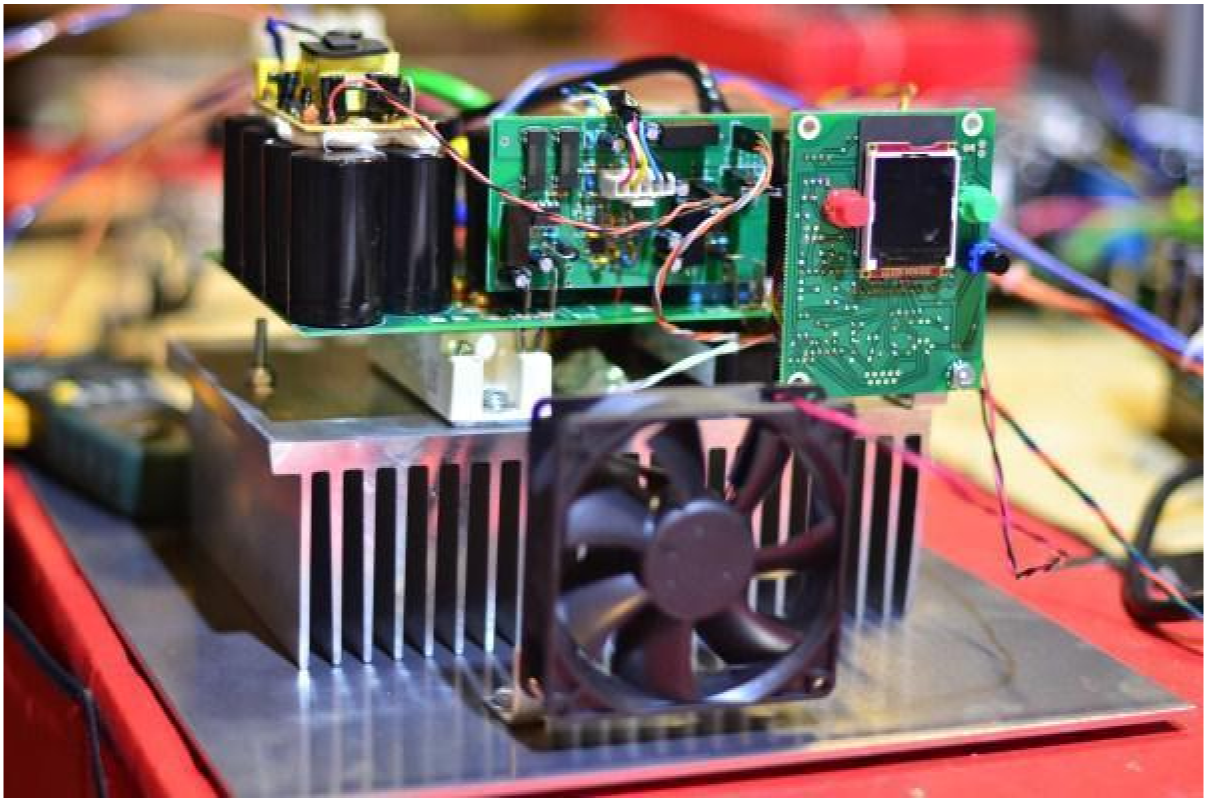
And all connections together:



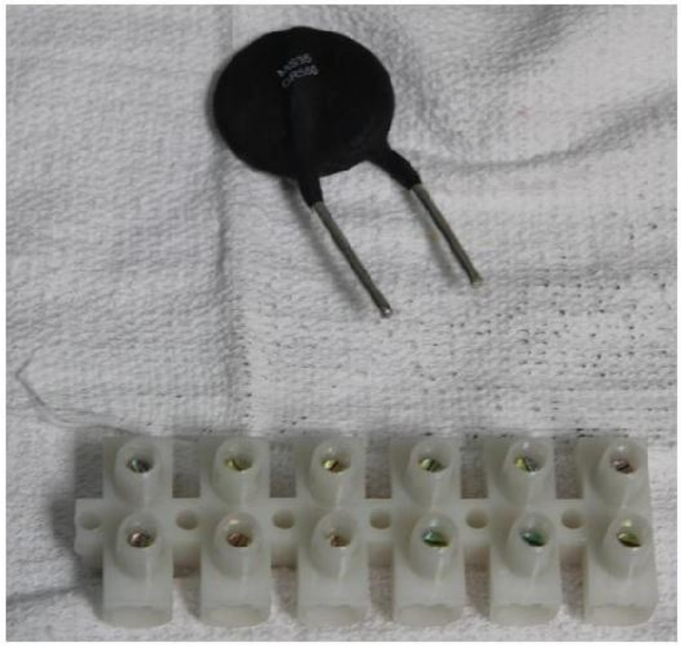
e. [OPTIONAL] programming dongle:
    i. Need: 6x 12” wires, 2x 6-pin female headers
    ii. One 6-pin header will mate to the bottom 6 pins on center left side of the control board (left of the Arduino board)
    iii. The other header will be brought outside of the charger (through one of the venting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

7. Connect the boards:

a. Connect Power Board Harness (Create and Solder a 6-conductor driver board harness:) to Driver Board
b. 12V DC supply output to driver board
c. Voltage sensing harness (driver to control)
d. PWM harness (driver to control)
e. Insert the driver board into the power board
    i. You may need to bend the male pins slightly for fit
    ii. 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
f. Current sensing harness (control to power)
g. Attach BMS / J1772 harness to control board. The other end with not be connected at this point.
h. Once you’re done, it will look something like this (this is a photo of one of our test rigs here where we test the boards before they go into the fully assembled units or to the customers who order these a-la-carte):



9. Assemble a HV input/output connector:
    a. Use a supplied 4-6 position connector



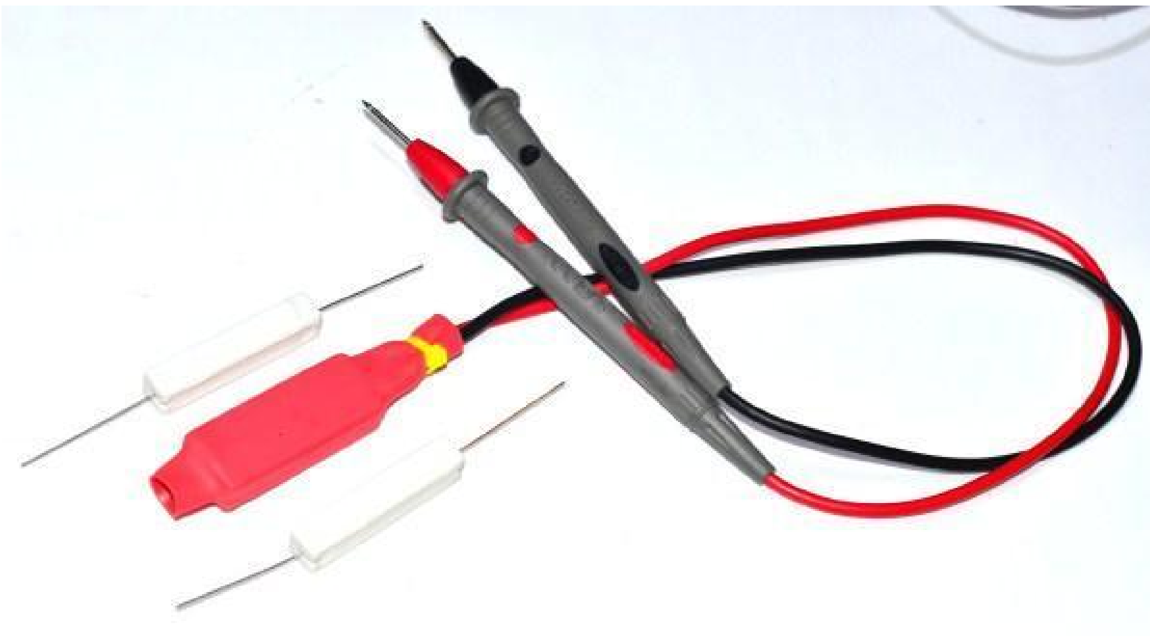
b. 2 positions to be used for input AC lines
c. 2 positions – for output DC
d. Remaining positions (if any) can be used for mounting inrush resistors
e. Using the connector as a template, drill mounting holes in the back side of the enclosure, right above the venting slots (position connector horizontally)
f. Connect the 4 power wires (that you have previously threaded through the venting slots) to the corresponding positions of the connector
g. 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!
h. Connect inrush current limiters:
    i. 2x 50A limiters are normally supplied - use 1 on each AC line
    ii. These parts heat up to 220C (450F) at max current.
        1. Use caution in placing – nothing flammable should be close
        2. Leads are copper so they transmit heat very well – take that into account when mounting
        3. We suggest extending the leads with 4-6” of the AWG 6-8 wire and then connecting one side to the input/output charger connector and the other side to your input AC lines
        4. 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
    5. 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:
    a. 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
    b. Check for loose bolts / connections
    c. [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)
    d. 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. To dissipate PFC stage voltage, tap into C1 and E2 terminals. To dissipate output voltage, tap between E2 of the output IGBT (right-hand side) and output pad of the charger.

Firmware

1. Upload the firmware:
    a. Connect the supplied FTDI cable to your PC
    b. Make sure your PC recognizes the cable and assigns it a separate COM port
    c. Download firmware from E Motor Werks site
        i. Current link https://drive.google.com/file/d/0B8zi8eqgqJWoaERHMV9qazY2UGc/edit?usp=sharing)
        ii. Unzip firmware into a separate folder. Remember its location
    d. Install Arduino-0022 package (simply select your installation folder and unzip the archive there)

e. Copy libraries (current file with libraries is [120113 (SPE LCD).zip]) into the 'libraries' folder in your installation folder
f. Be sure your downloaded code is configured correctly for your version of the charger kit
    iv. Launch arduino.exe from your Arduino installation folder
    v. IGNORE notification to update to the latest Arduino version. The firmware will NOT compile in later version of the Arduino IDE
    vi. File->Open Sketch, browse to the folder containing firmware, select .pde file there, click Open
    vii. File->Tools->Board, select 'Arduino Pro or Pro Mini (5V, 16MHz) w / ATmega328' as the board type
    viii. Check the "#define …" statements in the first 2 pages of the code (marked as '--------- MAIN SWITCHES -----------')
        1. Uncomment '#define PFC' if this is a PFC unit
        2. Uncomment '#define A7520_V' if your driver board uses a 7520 chip for battery voltage sensing
        3. You should normally have '#define OUTC_SENSOR Allegro_100U'
        4. If using a custom 100A output inductor, uncomment '#define MCC100A'
        5. Make sure '#define PFCdirect', '#define UV12', '#define NiXX', '#define buck_Ecore', '#define A7520_mV' are all COMMENTED OUT

\

v. Upload the code to the charger:
    1. Depress the programming button on control board to disconnect power from the LCD (to enable programming)
    2. Picture of programming & control buttons

    3. Connect the FTDI board to either…

   a. …the programming header on the control board (on top of Arduino board)
   b. OR to the programming dongle you made earlier (a simple 12” 6-pin female-to-female 0.1” pitch harness)
   c. In Arduino IDE, click Compile icon, confirm error-less compilation
   d. Click Upload icon, confirm error-less upload

2. **Test the charger.** This is the testing sequence we use at Electric Motor Werks on all units. Generally, if any step fails, DO NOT proceed until you fix whatever is preventing it from passing. **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… **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 E Motor Werks distribution)
   a. Prep the props:
      i. 2x regular household lamps connected in series – this will be a dummy test load for the charger
      ii. ~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)

   b. Test Logic circuits:
      i. Make sure programming button is pressed in (small button to the right of the LCD, below the green (or black) button
      ii. 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)
      iii. Fans should turn on
      iv. The screen should go live
      v. If you see the 5-second count-down on the LCD, press any control button
      vi. On the screen, set up the charger:
         1. Select LiFePo4
         2. Select 3.5 V CV cutoff (‘350’ setting in the CV menu)
         3. Select 30 cells
         4. Select capacity (e.g. 100 amp-hours)
         5. The charger should ask you to short the output:
            a. 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

            i. You need to recalibrate your unit whenever you change any major switches in the code affecting voltage sense (#define A7520_V)
        b. If it does ask to short the output:
            i. MEASURE the output before shorting
            ii. If it reads more than a few volts, connect your dummy lamp load to the output. Measure again in a few seconds
            iii. If it still reads more than a few volts, something is wrong. Time to debug...
            iv. Assuming reading is zero, short the output, press the green button
    6. The charger should now ask to connect the battery to calibrate itself:
        a. In this first calibration, use 30-60V battery when requested to connect battery, ideally through a DC circuit breaker (you can use a higher-voltage battery but in that case you run a higher risk of blowing something up if you made any mistakes in assembly. For example, we had an incorrectly installed elcap which literally exploded when we plugged a 300V battery to the output… You can also use lower-voltage battery but then charger might not be able to detect it)
            i. Watch polarity!
            ii. Use a clamp meter to measure the current through one of the wires connected to the batteries. Current should be less than a couple of Amps
            iii. 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)
        b. 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
        c. If the LCD does not change at all when you connect the battery, something is wrong in the sensing circuit. Time to debug…
vii. 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) Make sure to save the configuration by passing battery setting confirmation (Press Yes on confirmation screen)

viii. 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%.
ix. The output voltage reading on LCD should be very close to your battery voltage. The current reading should be zero (or 1A max)
x. Check the heatsink temperature readout. Should be close to ambient
xi. Disconnect AC from the adapter

c. **Test power circuits**.
   i. If you have assembled a liquid version of the kit, you will need to connect the water cooling circuit to your cold plate.
      1. The cold plate we use has bare 3/8" OD copper tubing so you will need to use some interface hardware in order to connect your flexible tubing to the charger
      2. A good option is to use one of the "sharkbite.com" connectors:
         a. "U110LF" part number fits great and is available from any Home Depot store (in the US) for ~$4-5 apiece. You can also Google it and order online from any of the 10s of stores that will show up. We did find that Home Depot usually has the best pricing for these (!)
         b. We like it because it's a fully reversible connector so if you ever need to disconnect your cooling system / remove the charger from your car, it's just a matter of seconds
         c. The above fitting will have a male ½" MNPT end that you would need to couple with something like "½" FNPT to ½" barb fitting". Then you can fit your flex tubing over the barb (do not forget to clamp with some hose clamps)
      3. Another option is to get a couple of "½" barb to ½" barb" fittings from any hardware store:
         a. The inner diameter of those is ~3/8" and fits the cold plate's tubing perfectly
         b. You would fit the barb fitting on top of the pipes and solder it using a torch and plumbing solder
         c. This option, while a little bit cheaper, is more involving and more prone to errors (bad joints etc). It also makes multiple connects / disconnects harder
   ii. 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!):
      1. Connect your 30-60V battery to AC inputs of the charger:
         a. Be ready to quickly disconnect if you see / hear / feel anything out of the ordinary (e.g. hissing / crackling sounds etc)
         b. Wait for 10-15 seconds

2. Measure input voltage on the IGBT (bottom and middle terminals). Should be very close to your input voltage
3. Measure input current if possible (clamp meter is handy here). Should be very close to zero
4. Connect your dummy lamp load to the charger. Measure output voltage. It should be close to zero

iii. Limited test of Power + Logic stages:
1. Connect AC adapter to the main power lines in the charger:
    a. Solder AC adapter’s AC input to the small AC wires that you earlier had attached to the power board (that should by now be firmly attached to the cap block with the sealant)
2. Connect 120VAC (regular household power) to the input lines of the charger – ideally through a protected power strip
    a. Be ready to quickly disconnect if you see / hear / feel anything out of the ordinary (e.g. hissing / crackling sounds etc)
    b. The PFC stage will turn on, potentially causing dimming the lights for 1-2 seconds. This is normal.
3. Let the charger time out through the calibration routine (5-second timeout)
4. Set the Power level:
    a. If you get a ‘reverse polarity’ warning on the LCD screen, just override as directed on the screen
    b. If the charger stops to ask for input power, set input and output power to 10A
    c. 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
    d. During the power setting routine, note the stated input voltage. It should read close to 120V
5. Check PFC operation:
    a. 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...
6. Measure the output of the charger. Should be close to zero (remember that you still have the lamps connected to the output)
7. Check the input voltage measurement - displayed on the LCD as ‘I: xxx V’. Should read 120V. If not, follow the procedure below to calibrate the firmware
    a. Note that the DC input voltage may NOT read correctly and that should not be used as the reason for going through this calibration process.

        b. Carefully measure the voltage between mV and G pins of one of the driver board connectors. Record the voltage - let's call it V_120
        c. Disconnect 120V and connect 240V input to the charger
        d. Repeat the measurements of mV / G. Record as V_240
        e. In firmware, circa line 1518 (for V14_4 firmware, your line number may vary slightly), change '3' in 'if(Aref/ 1024.*outmV_ADC < 3) return 240;' to a midpoint between your recorded V_120 and V_240.
        f. Save and re-upload. Power charger up again and confirm you see the right AC value displayed on the screen
    8. Start the charger:
        a. Using the control buttons, navigate through the menu to start the charger
        b. Confirm run.
    9. This time, the duty should ramp up from zero to ~30-35% before charger goes into the CV stage
    10. In the CV stage, watch the duty to go from zero to ~30-40% and stabilize there. Lamp should be reasonably brightly lit at this point
    11. 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 calibrate the circuit later)
    12. Let the charger run for 10 min. Check the heatsink temperature readout on the screen. Should be close to ambient
    13. Disconnect AC input & lamps

iv. Full test of the Charger on resistive load:
    1. Connect the 5kW load to the output
    2. Connect 240VAC to the input
    3. Confirm your charger reads 240V input voltage (only for V13 driver boards or higher)
        a. Wait to time out through the first 5-second timeout
        b. The charger should now show 240V input voltage
        c. If the input voltage reading is still 120V, you need to calibrate a sensing circuit based on PC817 IC by editing this line in code: " if(peakV>2.) return 240;" - this will be around line 1411 in V12 firmware
            i. easiest way is to change '2.' until you get correct detection of BOTH 120V and 240V
                1. best to change that value to the middle of the current range you are exploring
                2. For example, if the current value is 2., change it to 1. (mid point between 0 and 2). If the charger now reads 240V on both 120V and 240V input, change the threshold value

        to 1.5 (midpoint between 1 and 2V).
      3. Repeat until you get reliable detection.
    ii. Another, more precise way is to measure voltage on mV pin of the G-V-mV connector on top right of the driver board and adjust the same line in firmware based on your readings.
      1. Measure mV voltage (relative to the G pin on the same connector) when charger is connected to 110V. This is your V1
      2. DO same for 240V connection. This is your V2
      3. Change the ‘2.’ threshold in firmware to ‘5-(V1+V2)/2’. Basically, you are moving the threshold to the midpoint of the voltage Arduino detects at these two voltage points.

4. Repeat the steps from the previous limited test, confirm normal operation
5. Power cycle 240VAC
6. Interrupt the first timeout, set the charger to 70 cells
7. Interrupt the second timeout, set max output current at 25A
8. Watch the charger ramp from zero to ~70% duty, go into CV mode, then ramp to 70% again and stabilize there
9. Observe the heatsink temperature for 5 min. Should not exceed 20 degrees above ambient
10. Disconnect AC input and 5kW load

v. Full test of the Charger on battery load:

1. Connect lamps to output. Confirm zero volts.
2. Connect 240VAC to input
3. Interrupt the first 5-second timeout, set the right number of cells for your pack
4. Press through the ‘short output’ step (as you have the lamps connected which for calibration purposes is the same as shorting the output)
5. 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:
    a. If you did not use the output diode for your build, you will get a spark on the first connection. This is normal.
    b. If your battery voltage is above 140-160V, we recommend using a power resistor (10-100R) for the first connection to pre-charge the capacitors
6. Follow the instructions on screen to calibrate the charger again for your actual battery voltage

7. Low-power run:
    a. Set power output to 10A
    b. Watch the charger ramp from zero to some duty cycle corresponding to your battery voltage (generally will be ~110% * <your battery voltage> / 380VDC)
    c. Let run for 5-10 min, watching temperature. Should be minimal rise
    d. Interrupt the charger by pressing and holding the red button. Press again to exit.
8. Medium-power run:
    a. Press and hold the green button until the charger goes back to the power setting dialog
    b. Set to output current corresponding to 6kW output (e.g., if your battery voltage is 200V, set the charger output to 30A)
    c. Start the charger
    d. Let run for 5-10 min:
        i. Watch temperature. Should see heatsink temp rise of 10-15C
        ii. Confirm output voltage rise (rate of rise depends on the SOC and capacity of your pack)
    e. Interrupt the charger by pressing and holding the red button. Press again to exit
9. Full-power run:
    a. Press and hold the green button until the charger goes back to the power setting dialog
    b. Set to output current corresponding to 12kW output (e.g., if your battery voltage is 200V, set the charger output to 60A)
        i. The charger might limit the current to lower value depending on your battery voltage (to satisfy the limits of 12kW and 70A output)
    c. Start the charger.
    d. Let run for 5-10 min:
        i. Watch temperature. Should see heatsink temp rise of 15-20C
        ii. There should not be any unusual loud noises coming out of the unit. If you hear screams, loud buzz, etc – time to debug.
    e. If possible, let the charger run through full charging cycle:
        i. Watch temperature & voltage to prevent damage to charger and/or battery

ii. 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
iii. Confirm CC to CV switch at the right CV voltage
iv. Confirm CV termination at the right output current
    1. ~5% of the capacity, can be changed via firmware edits
    2. ~1 min lag in termination is ok
v. Disconnect AC
vi. Disconnect battery

d. Troubleshooting suggestions:

i. If you have made an error in configuring settings (i.e. number of cells, voltage, etc.) on the LCD screen: Press green button through until you have confirmed last digit and then use Red button to return to setting the first digit. Note: once past the last digit, pressing the green button will confirm setting and you will be taken to the next setting (screen will change
ii. Troubleshooting the driver board: Disconnect and Use 12V power harness to power up the board. Use voltmeter to check voltage between each set of output terminals; should be -15 to -17 volts on each.

Congratulations! You now have the best charger money can buy.
And you built it yourself!
Go Electric!

Yours truly,
Valery and the entire Electric Motor Werks Power Electronics Crew

TROUBLESHOOTING GUIDE

As you can imagine, this part is never complete. Please do not expect it to have all the answers. We will add to this section as we go.

Problem:

PFC stage does not boost input voltage to the expected PFC rail voltage (usually ~370-380VDC)

Debugging approach:

There are several 'lockout' functions in IR1153 that might be disabling it the chip and therefore the PFC stage operation:

1. First check voltage on R12-C32-pin4 (of IR1153) junction. You need to measure while 110VAC is on (so pls be careful). The voltage should be above 1V.
2. Second, check voltage on R7-R14-C33-pin5/6 junction (again, while 110V is on). It should be above 1V.
3. Third, make sure your IsS and IsG are connected properly to the corresponding pads on the power board.

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 a description and we'll add it here!

MODS File

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