# Some Safety Warnings

1. **With the power supply open and connected to AC mains (house) power, there is the possibility that you may contact AC mains (house) voltage.**

   **Depending on the scenario, that can be fatal.**

2. **The large aluminium electrolytic filter capacitors are considered hazardous, because of the combination of high voltage and high charge.**

   **You need to 'treat them with respect'.**

   **When you remove AC mains (house) power from the power supply, these filter capacitors take time to discharge.**

   **To gauge discharge status, a voltmeter can be used to measure the voltage on the filter capacitors.**

   **For information on manually discharging the filter capacitors, see http://www.electronicrepairguide.com/capacitor-discharge.html**

3. **In old power supplies, the thermal paste used between semiconductors and heatsinks, typically contained beryllium oxide (BeO).**

   **Beryllium oxide is known to be carcinogenic.**

# How to Convert the IBM 5150 120 VAC Power Supply to 230 VAC

## Discussion

The 120VAC version of the 5150's power supply, as-designed, is not switch or jumper-configurable for 230VAC operation.  However, there is a simple way to alter this design to work on 230VAC, provided 3.6 additional watts of AC power draw can be tolerated.  This document proves the instructions for such a modification, which can be carried out by persons appropriately skilled in electronic repairs.  Note, this is a properly-engineered change and the power supply active components see exactly the same conditions found in the original design.  As such, there is no reason to believe that reliability will suffer over that which is already present in the hardware you are modifying.  Be aware that the author takes no responsibility for any personal injuries or property damage that result from the use of this instruction.  Do not attempt this modification if you do not feel you possess the required skills.
The electrical theory and a schematic are provided at the end of this document.

## Preliminary Requirements

1. This work instruction applies only to power supply units with a PCB that has the same circuit as in the photos.
2. Any unit that has a 120 VAC fan will need that fan to be replaced with a 230 VAC part, or a 12VDC fan.
3. You will need to have good soldering and de-soldering skills using copper braid.  A solder-sucker is not suitable.
4. Clearly there are **lethal voltages** present when the unit is open and on AC power, and it is expected that you are aware of what safety precautions are appropriate.  For reference, my particular unit is safe from dangerous voltages after about 60 seconds wait after fully disconnecting from the mains.
5. This modification will take about 3 hours to complete.

## Tools Required

1. Soldering iron, solder, 1.5 or 2.0mm desoldering braid, and heat sink clip.
2. 1/4" and 3/16" sockets with drive handle and 100mm (4") extension, T15H TORX security bit and handle.
3. Small size smooth-jaw and diagonal cutter pliers, 1.5mm (1/16") drill bill and suitable drill motor.
4. Single-edge razor blade, junior hacksaw blade, magnifying glass, small lump of Blue-tack.
5. 3.5" hard disk to use as dummy load.

# Parts List

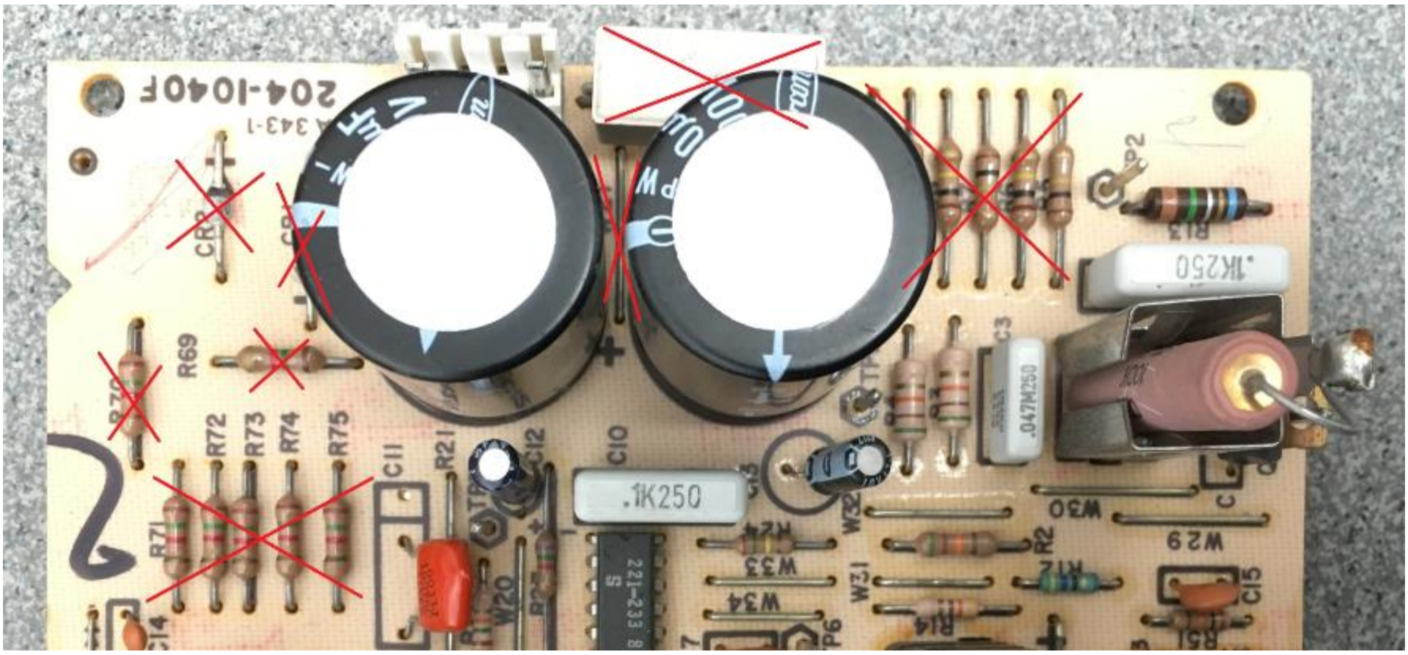
| Description | Item ref | Quantity | Part value | Jaycar p/n | Jaycar UOM |
|---|---|---|---|---|---|
| | | | | | |
| **EMI board** | | | | | |
| Resistor | R1 | 1 | 560k 1W | RR-2840 | 2 in pk |
| X2 capacitor | C1, C2 | 2 | 0.1uF 250VAC | RG-5236 | ea |
| | | | | | |
| **Main board** | | | | | |
| Relay | new part | 1 | 12V coil SPDT | SY-4066 | ea |
| Diode | CR1,CR2, 2 new | 4 | 1N5408 | ZR-1018 | ea |
| Fuse | F1 | 1 | 1T 3AG | SF-2226 | ea |
| Resistor | R1 | 1 | 10R 5W | RR-3250 | ea |
| Resistor | R71-75 | 5 | 3k9 1W | RR-2788 | 2 in pk |
| Resistor | R69, R70 w/mod | 2 | 10k 1W | RR-2798 | 2 in pk |
| Resistor | R8 w/mod | 1 | 33k 1W | RR-2810 | 2 in pk |
| Resistor | R5-7 | 3 | 100k 1W | RR-2822 | 2 in pk |
| Resistor | new part | 2 | 220k 1W | RR-2830 | 2 in pk |

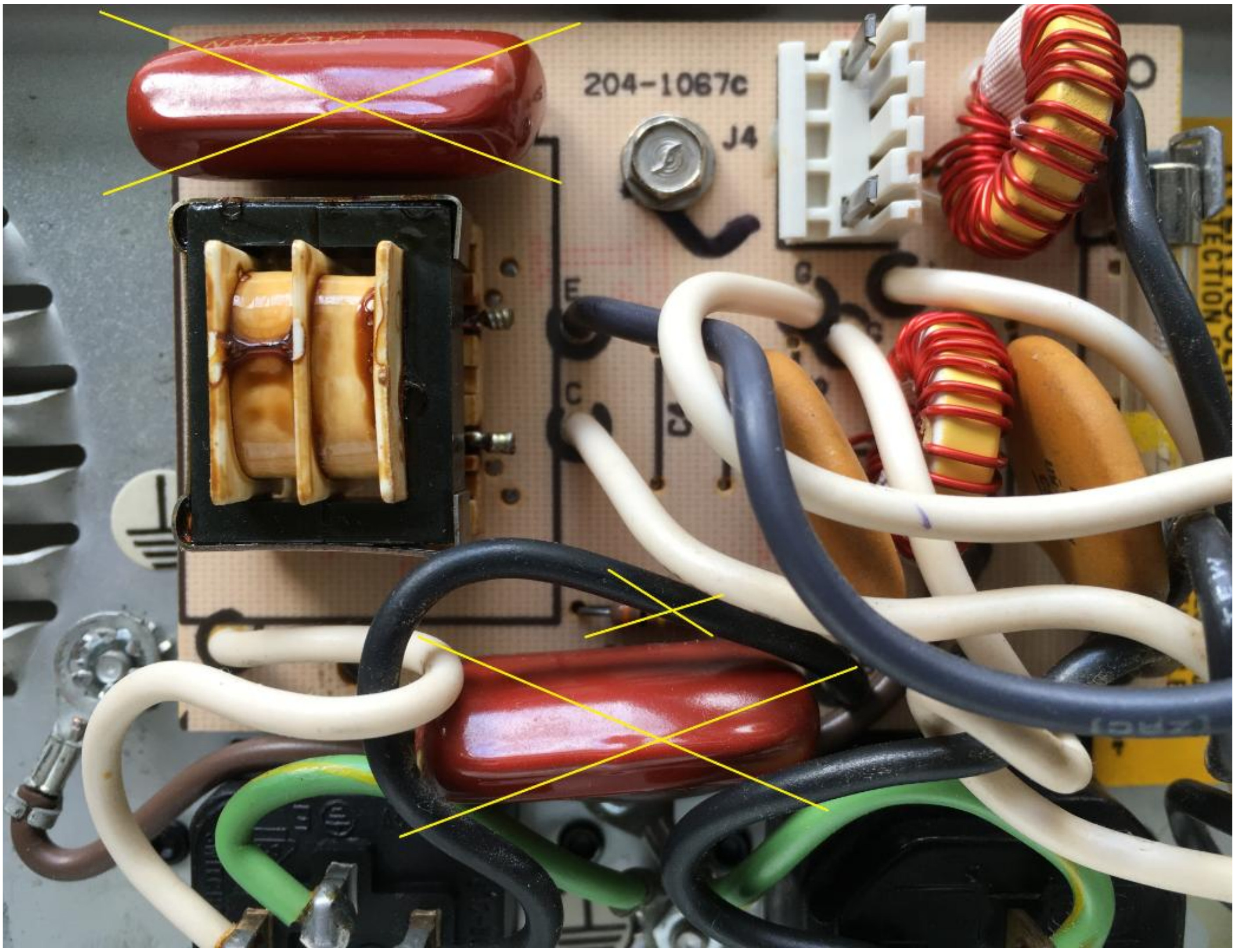
# Modification Procedure

a) Remove PS from PC and disassemble such that main PCB is removed and EMI PCB is loose on its wires.

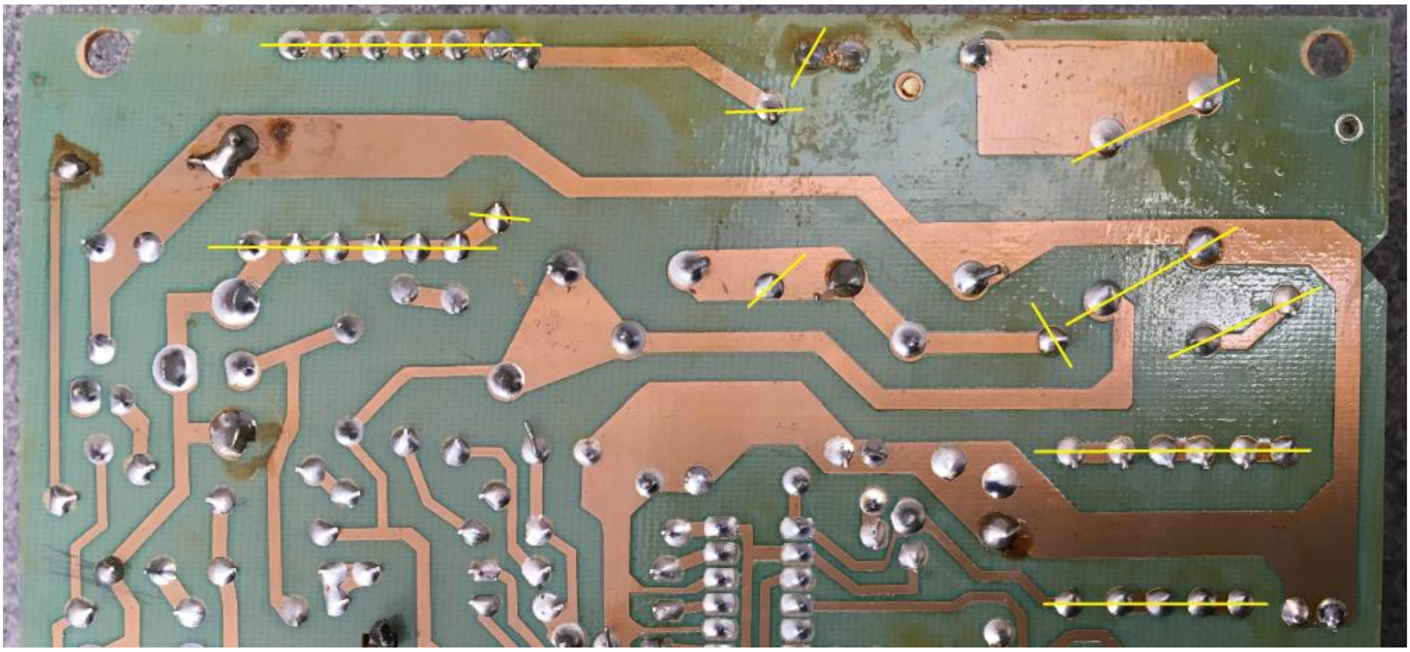
b) Double-check you have an applicable power supply by matching the main PCB with the photo below.

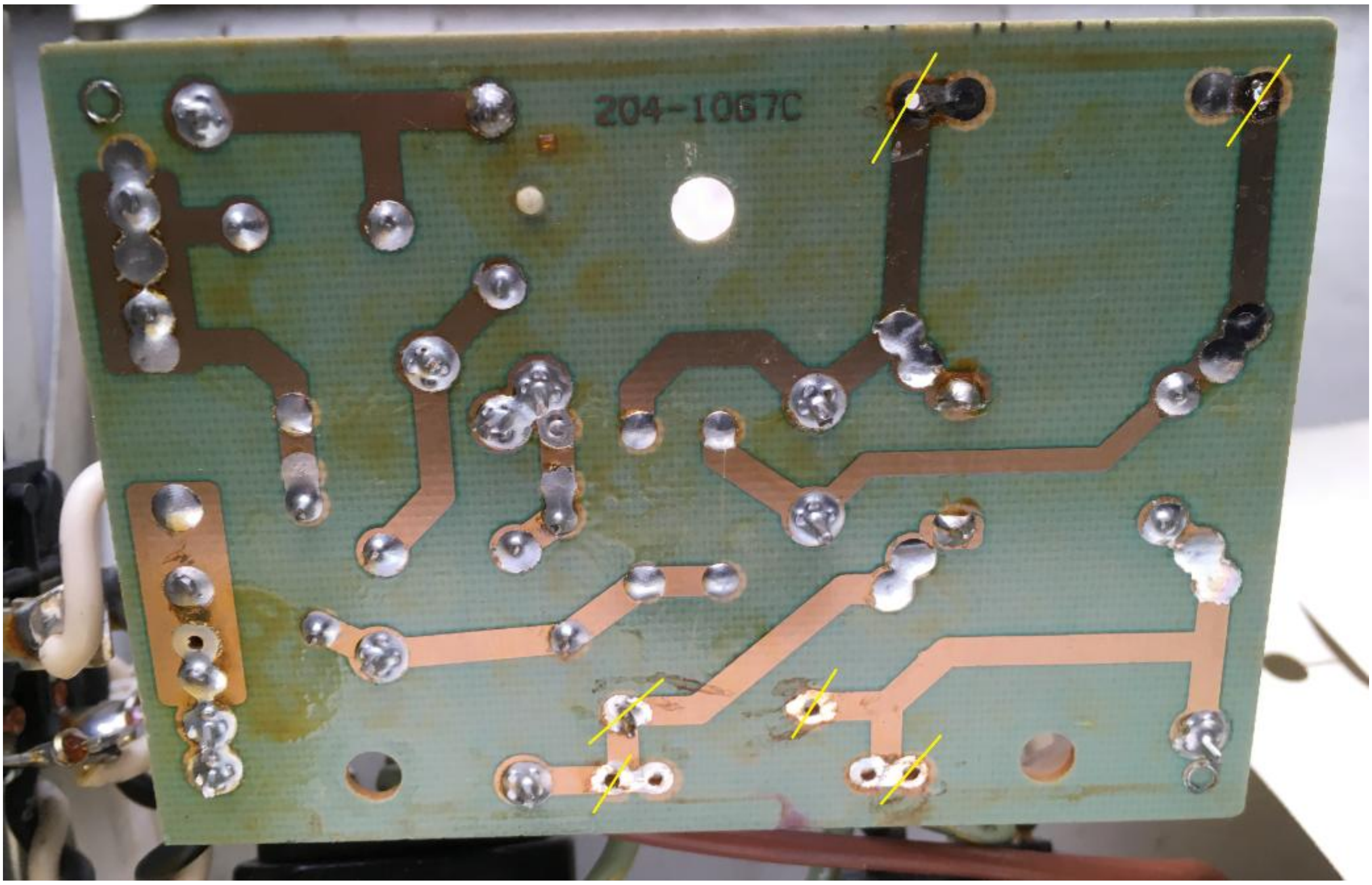
c) With small diagonal cutters carefully clip off all parts marked on the following two photos.



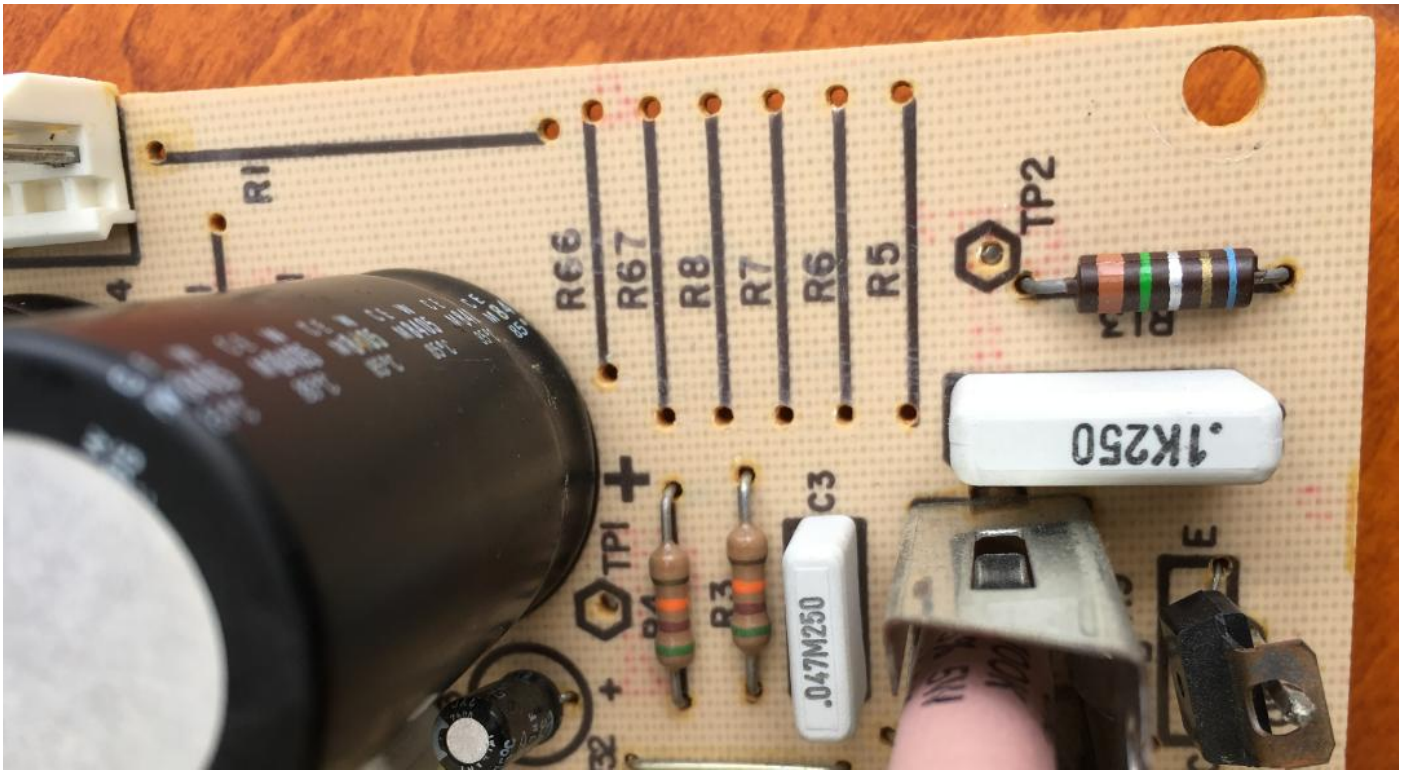



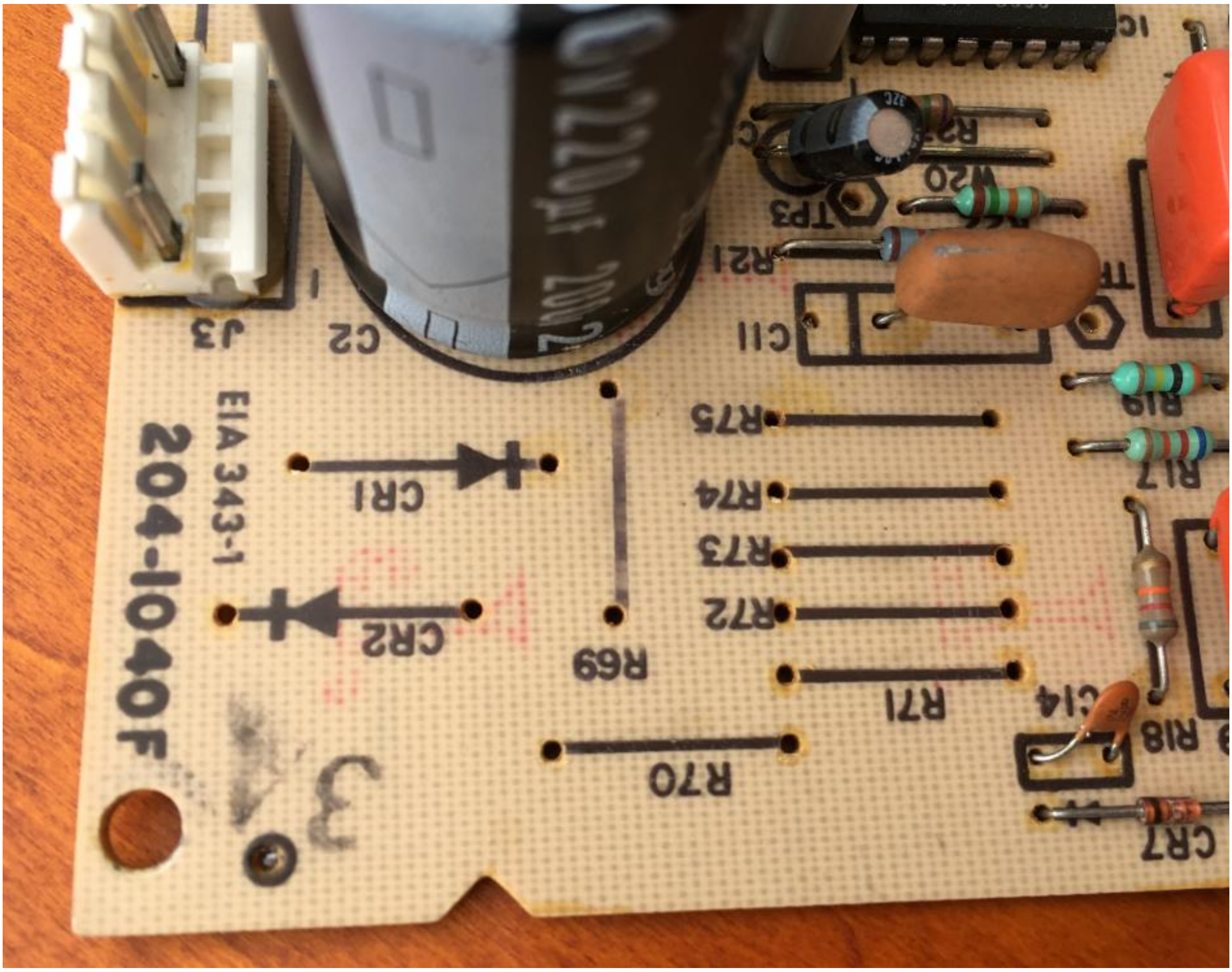
d) On both the PCB's copper side, desolder all wire nubs per following photos. **Use great care to avoid over-heating the copper.** The best technique is to lay the desoldering braid over the nub, heat the braid and drag it from nub tail towards the hole such as to push the nub out as solder is absorbed. Add new solder if you have difficulty.



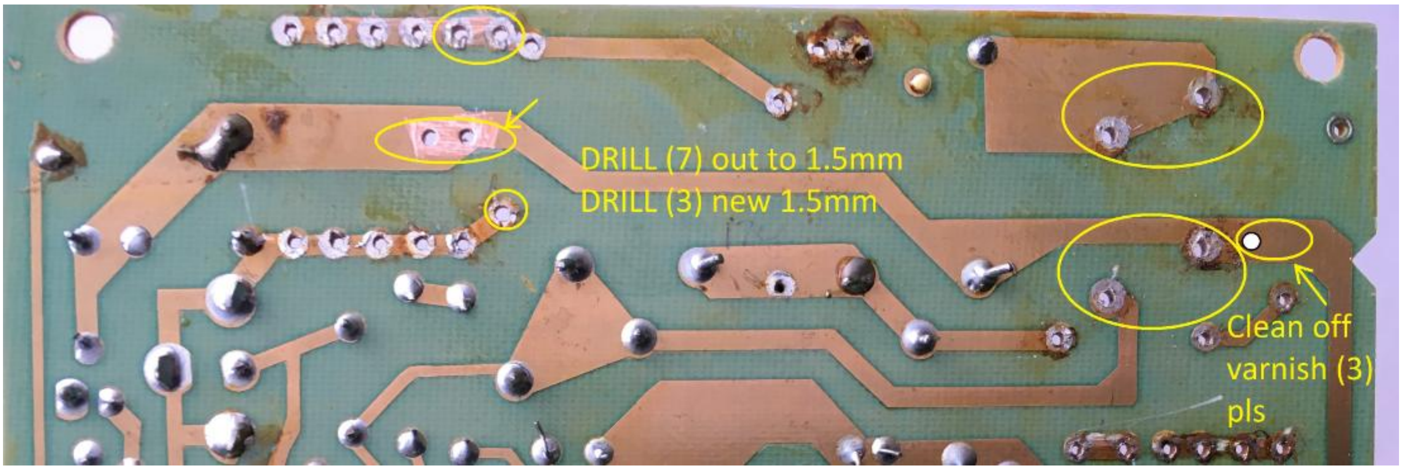


The main PCB should look like this from the component side. Your PCB may or may not have pins fitted at the many "TP" test points, e.g. TP2 below.

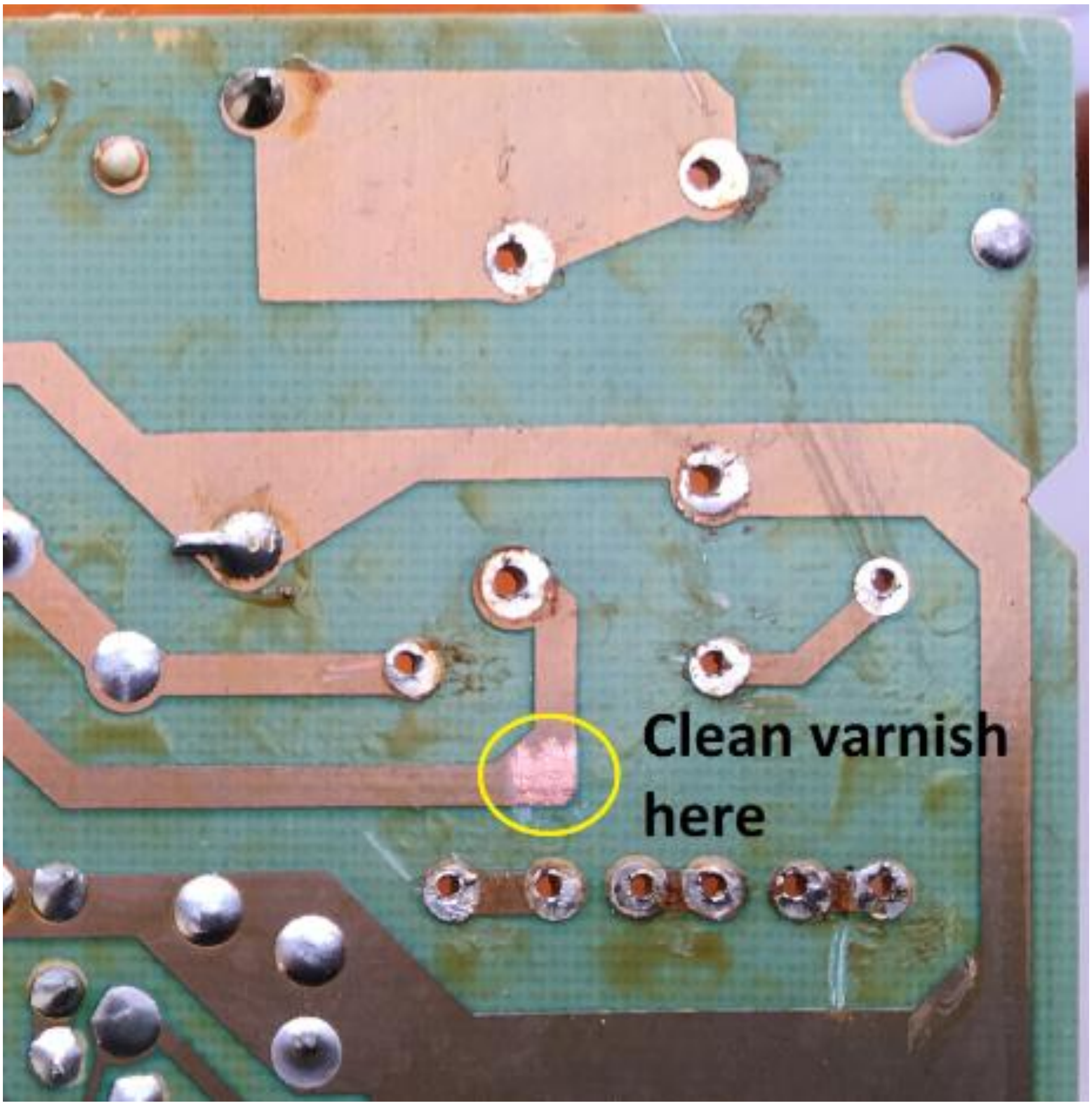


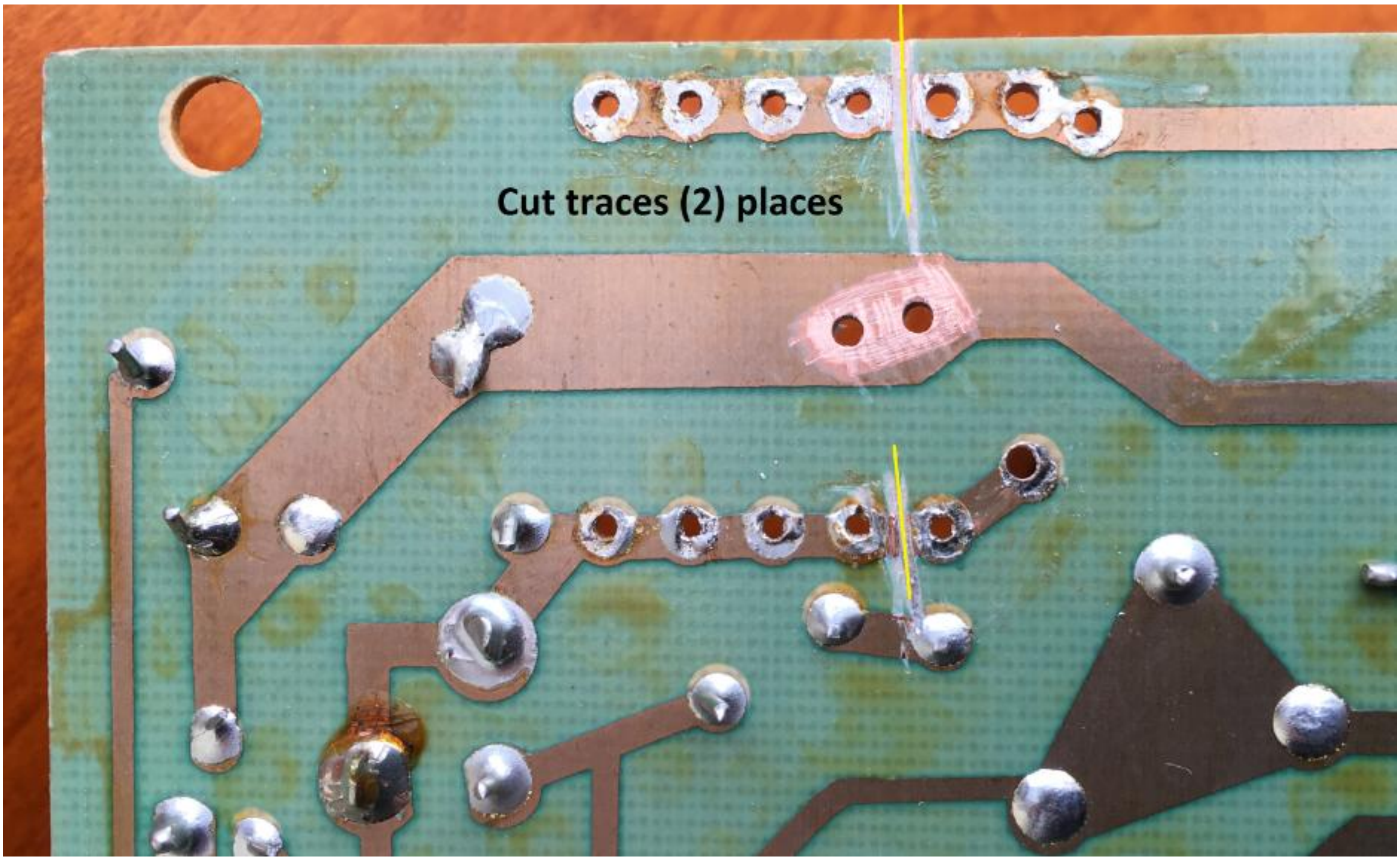
e) Drill out (7) existing and (3) new holes per the following photo using a 1.5mm drill **from the copper side.** At the (3) new holes clean away the varnish around the hole as shown.



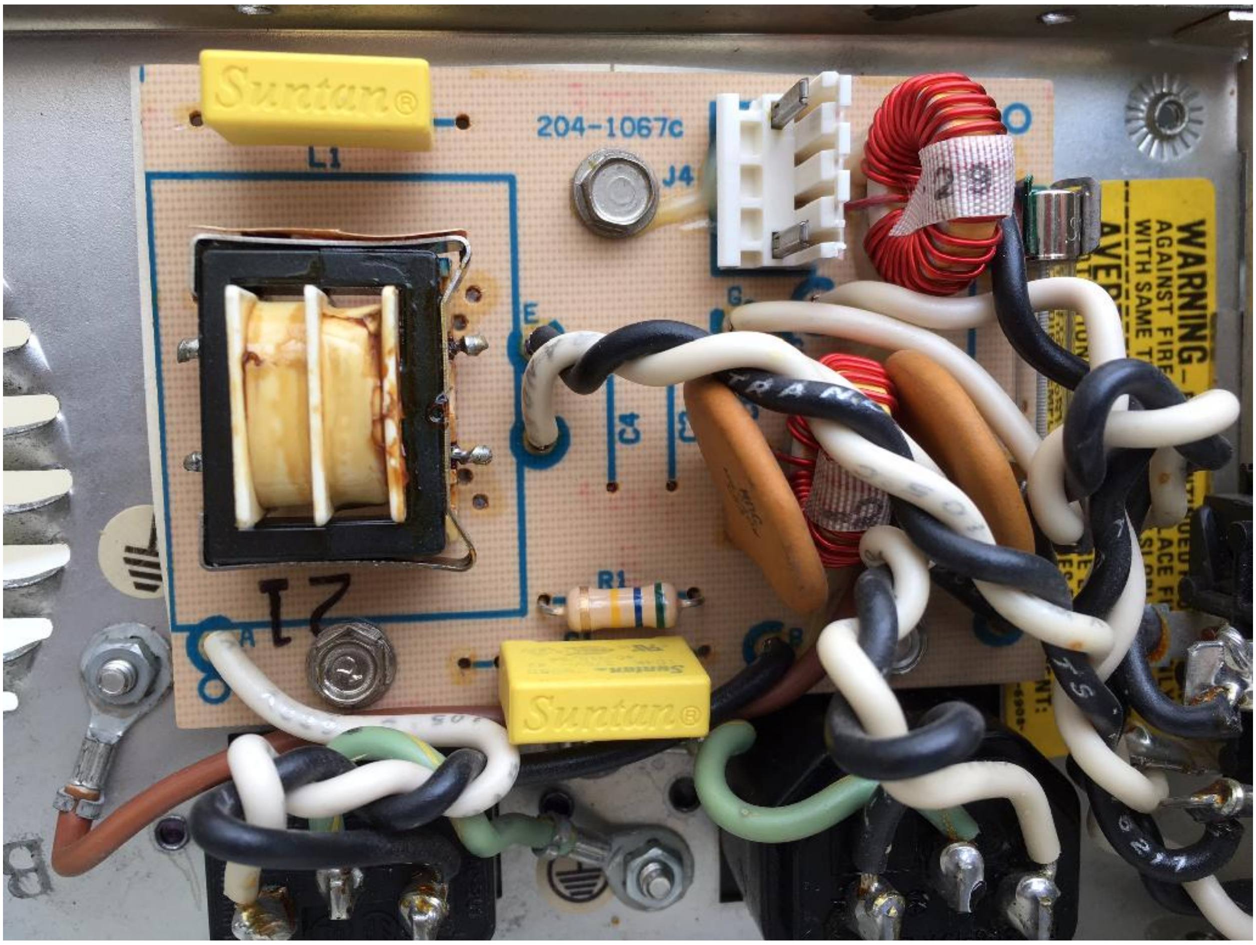

f) Clean away the varnish in one additional location as shown in the next photo.



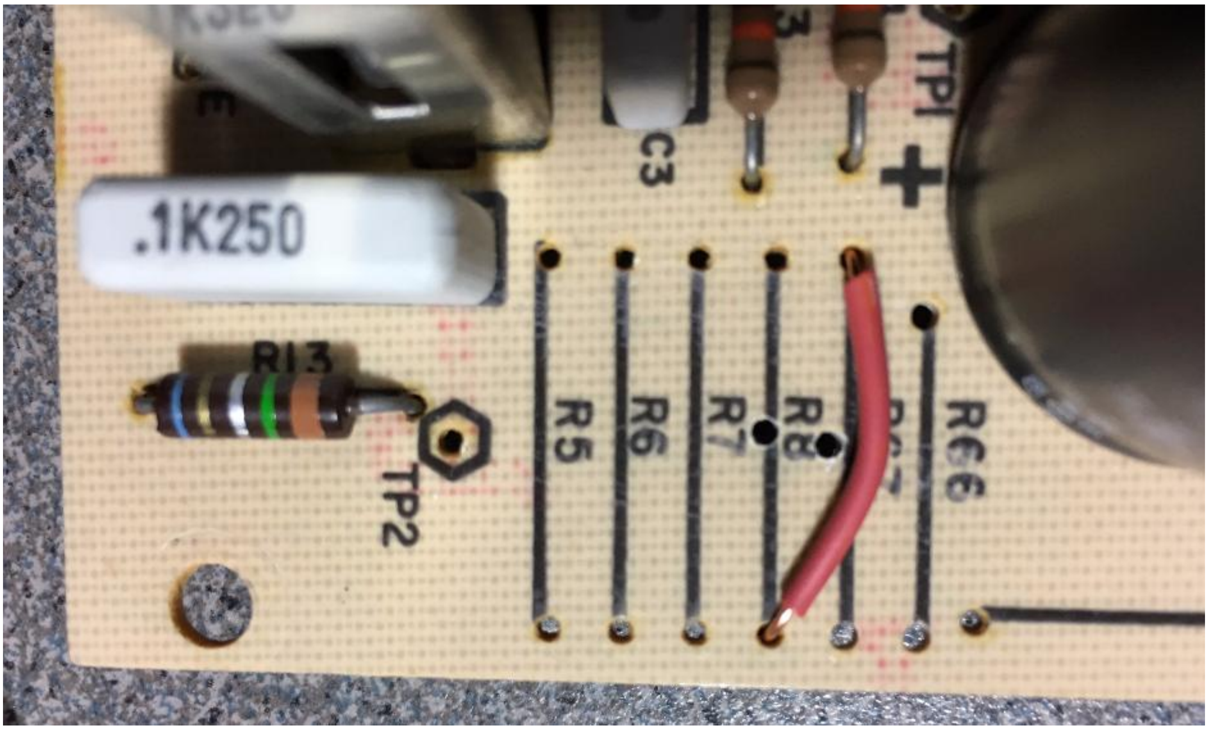
g) Cut (2) traces where shown with a junior hacksaw blade.  Inspect with magnifying glass to ensure the trace is fully cut.

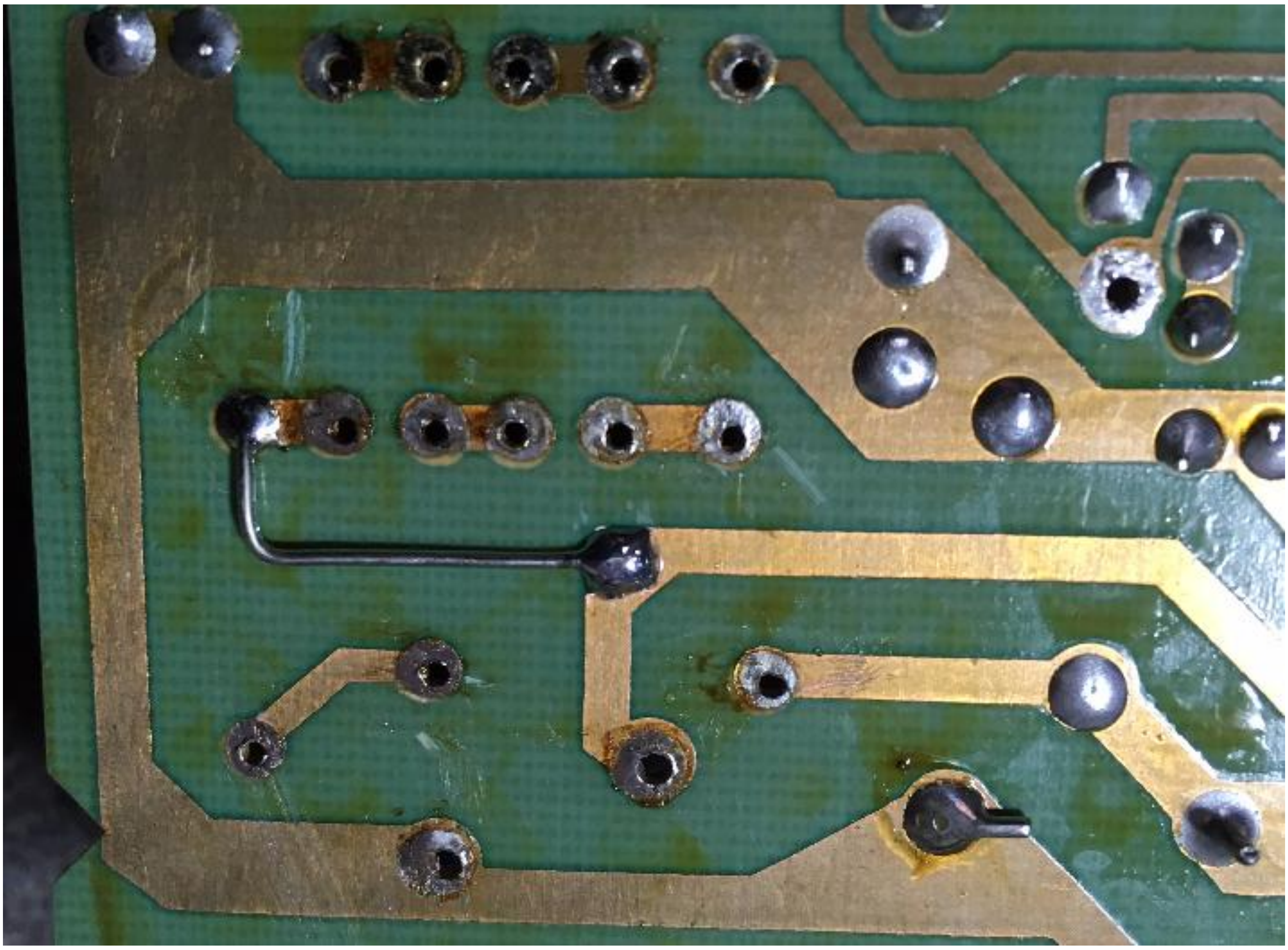



h) On the EMI Filter PCB install the 560k resistor at R1 and (2) 0.1uF caps (yellow) using the narrow pair of holes (at your option.)  Trim the leads before soldering to minimise heat required.  Install the 3AG T1 fuse as well and reattach the PCB to the case using the cardboard insulator behind the PCB (if equipped, or if not, make your own.)



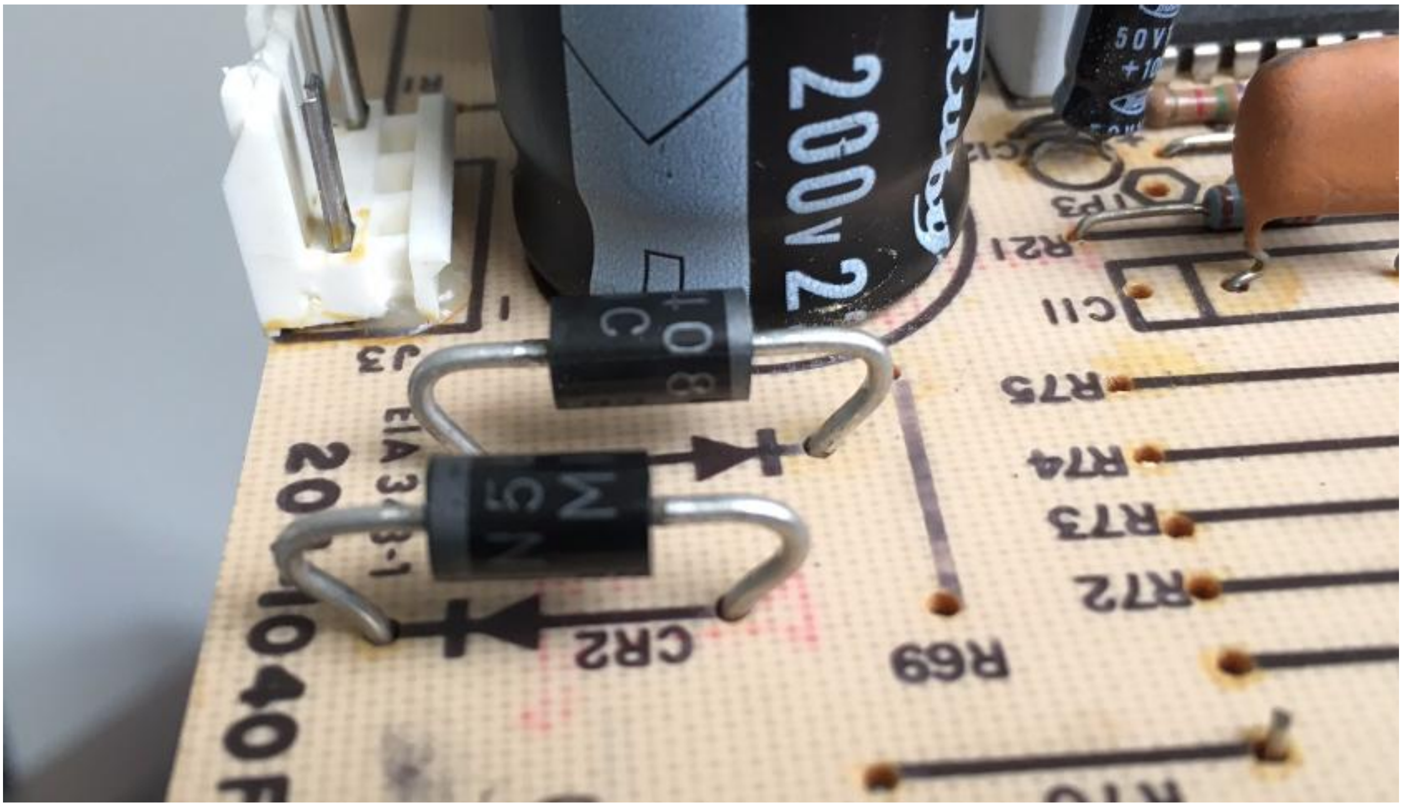
i) On the main PCB solder in (2) jumpers as shown, one is insulated with heat shrink.



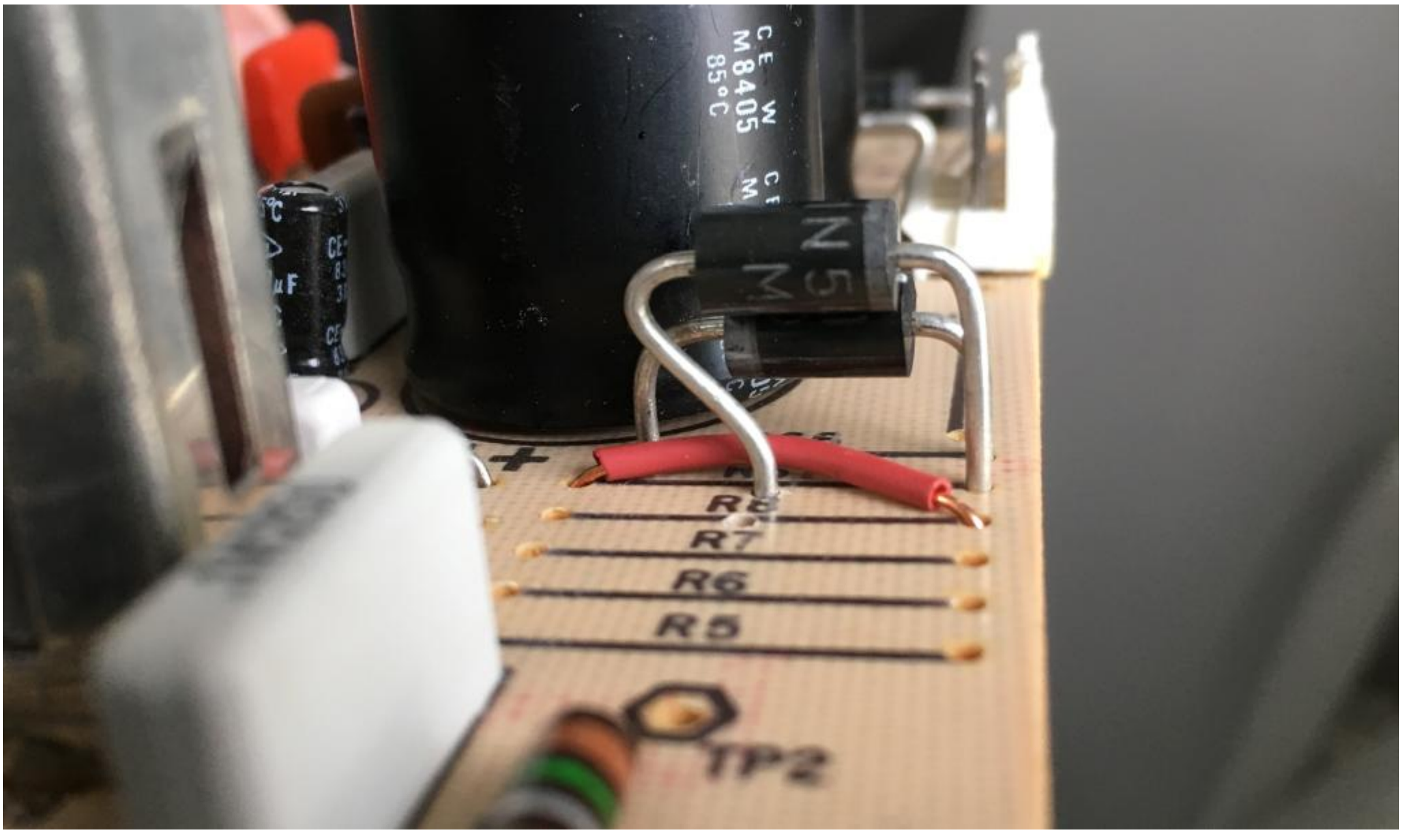



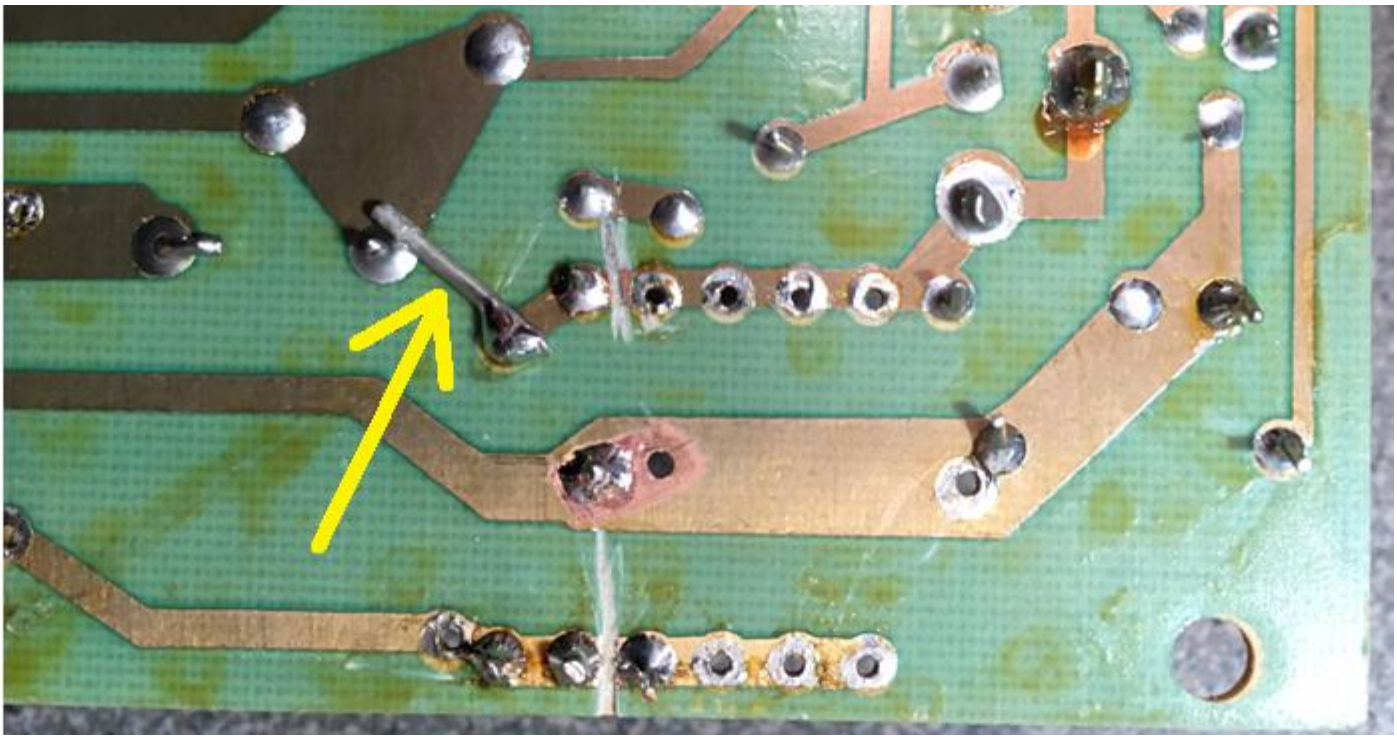
j) Prepare the (4) diodes. Bend legs as shown in the next photo using smooth jaw pliers. Be careful not to stress the plastic casing when you make the bends. Adjust leads such that they can be installed freely into their respective locations without further stressing the leads.

k) Solder two diodes into the existing diode locations first, noting correct polarity. Ideally cut the leads to near, or at the finished length to minimise solder heat required. Blue-tack is useful for holding parts in place while soldering. When soldering, heat the protruding lead first, then when solder melts move the soldering iron down to the pad. Resist the temptation to bend the diodes after installation as the strong leads may break the delicate PCB traces.

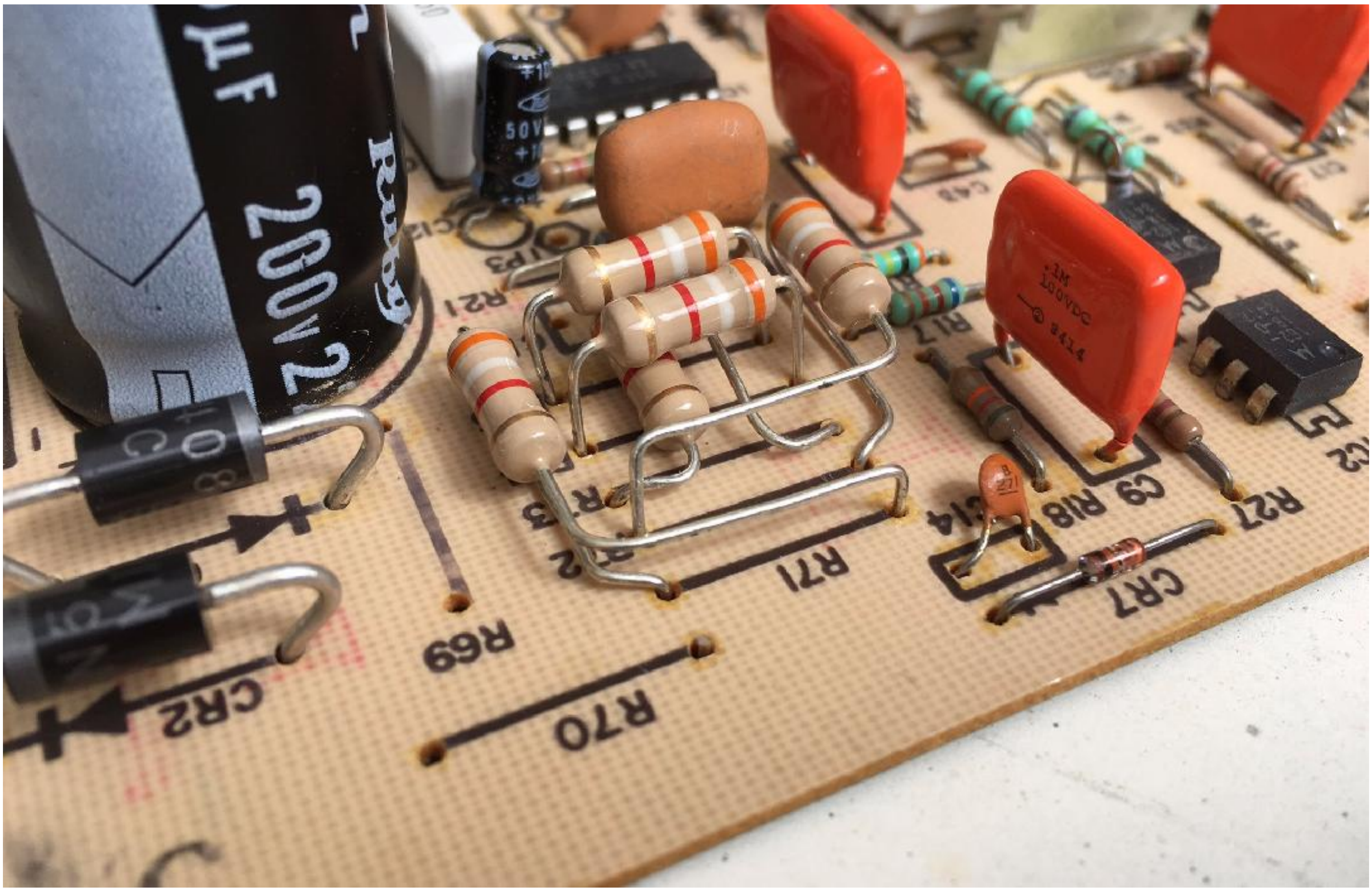


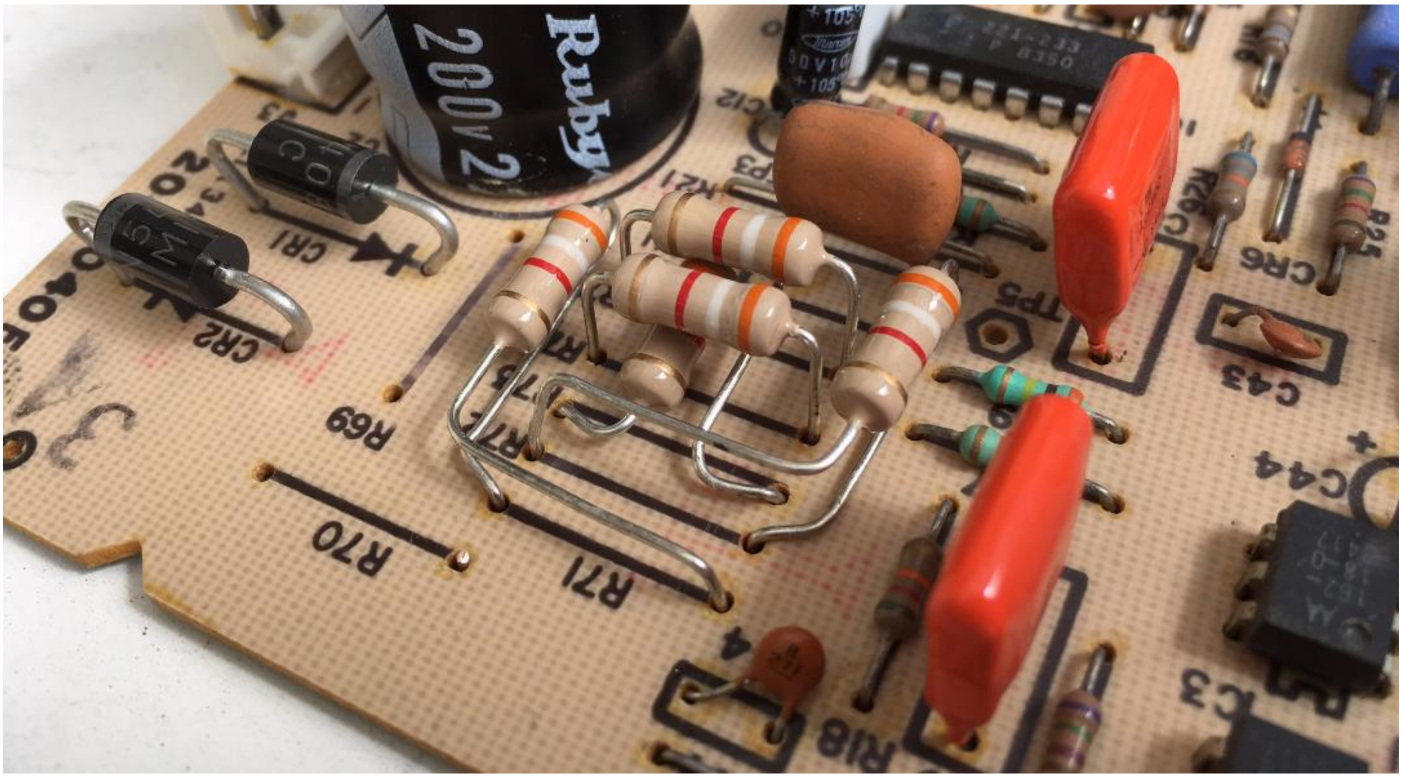
l) Install the other two diodes in the locations you created earlier being careful to **observe the polarity** shown in the photo.  On the underside use one of the lead cut-offs to create a jumper, solder both ends.

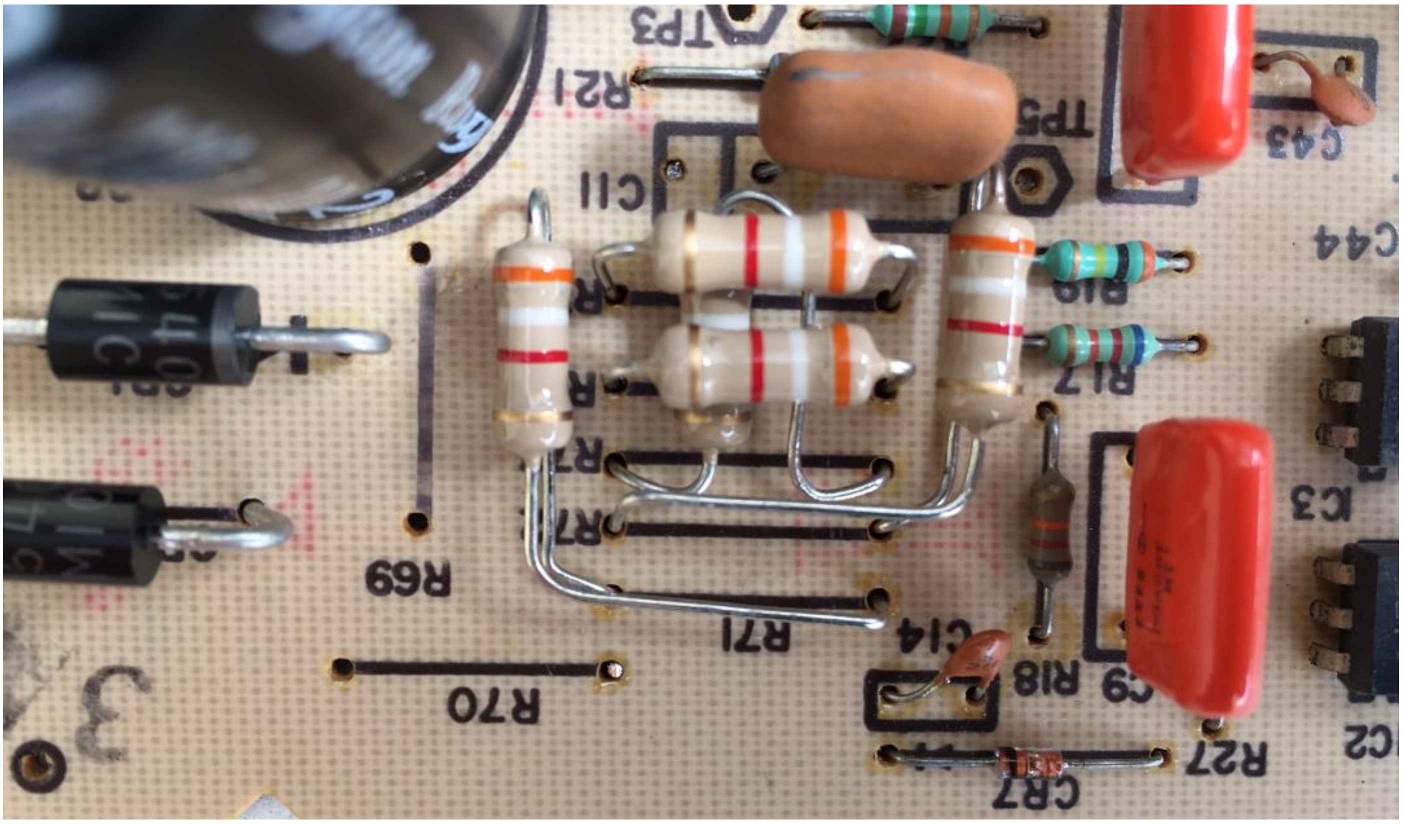


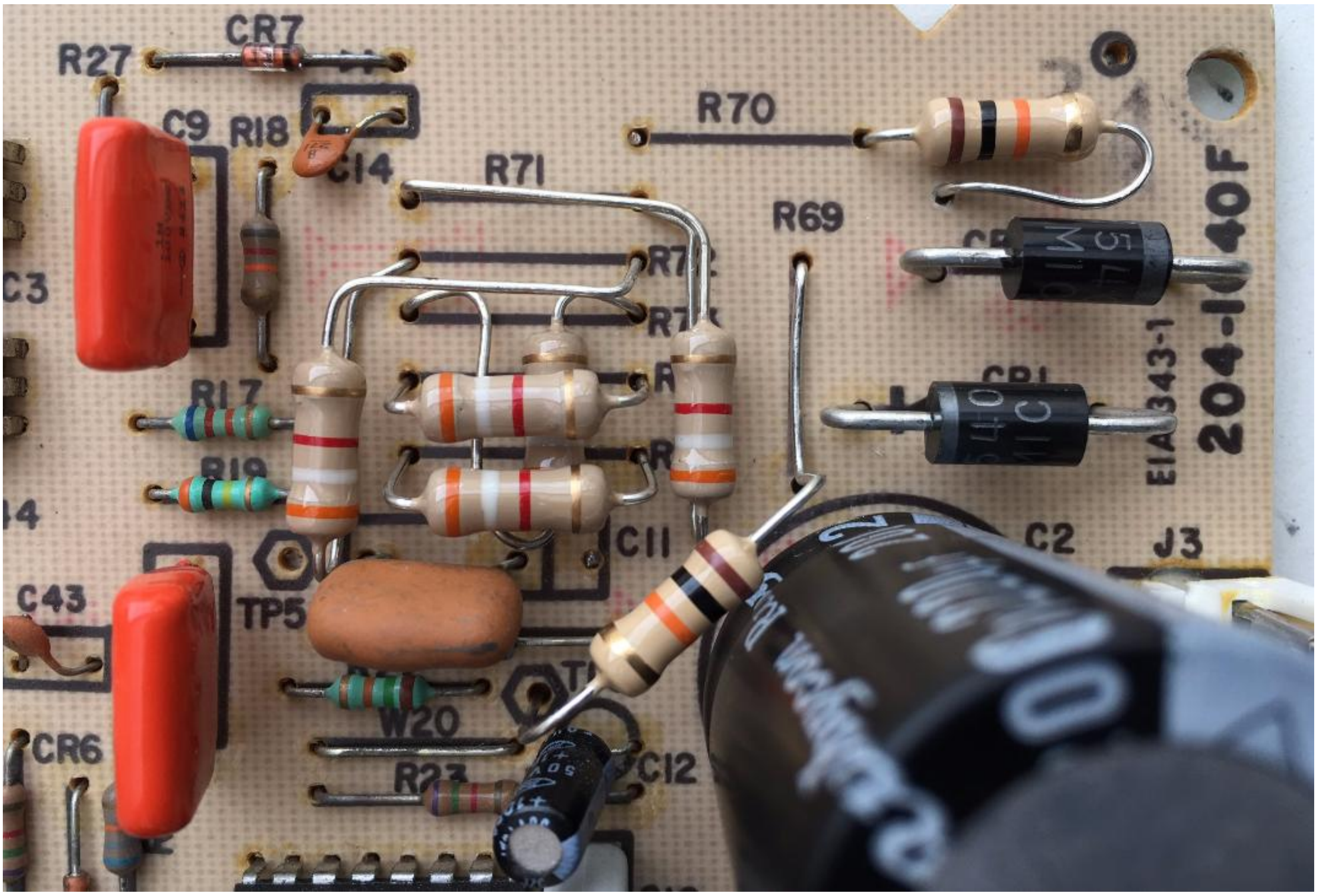
m) Prepare (5) 3K9 (1W) resistors.  These must be installed as shown in the next three photos at positions R71 to R75 to avoid clearance issues with the IEC connectors when installed and receive direct air flow from the fan.  It's a bit complicated but essential.  Start with R73, then R74, R75 above that, then R71, R72 either side.  No resistor bodies should go past the line at R73.

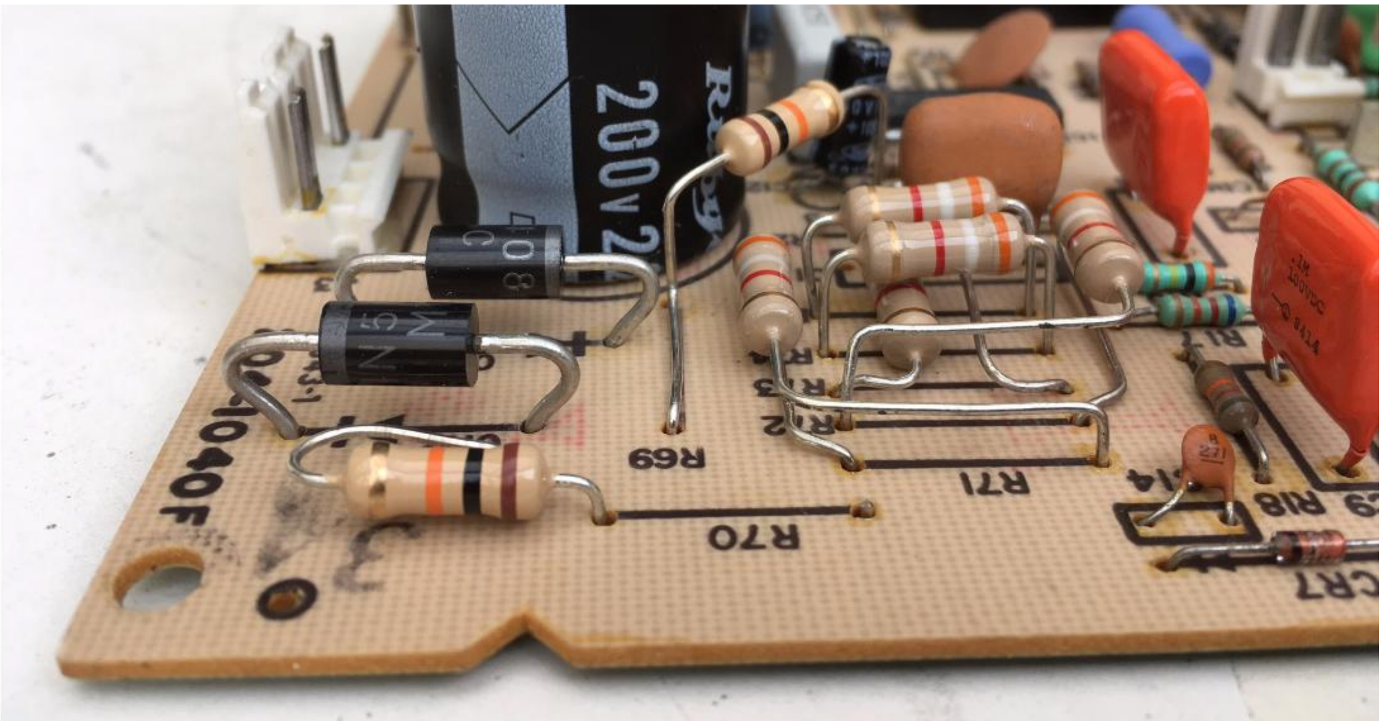





n) Install (2) 10K 1W resistors (brown-black-orange) as shown in the next two photos.  If TP3 is occupied with a pin, solder to it.

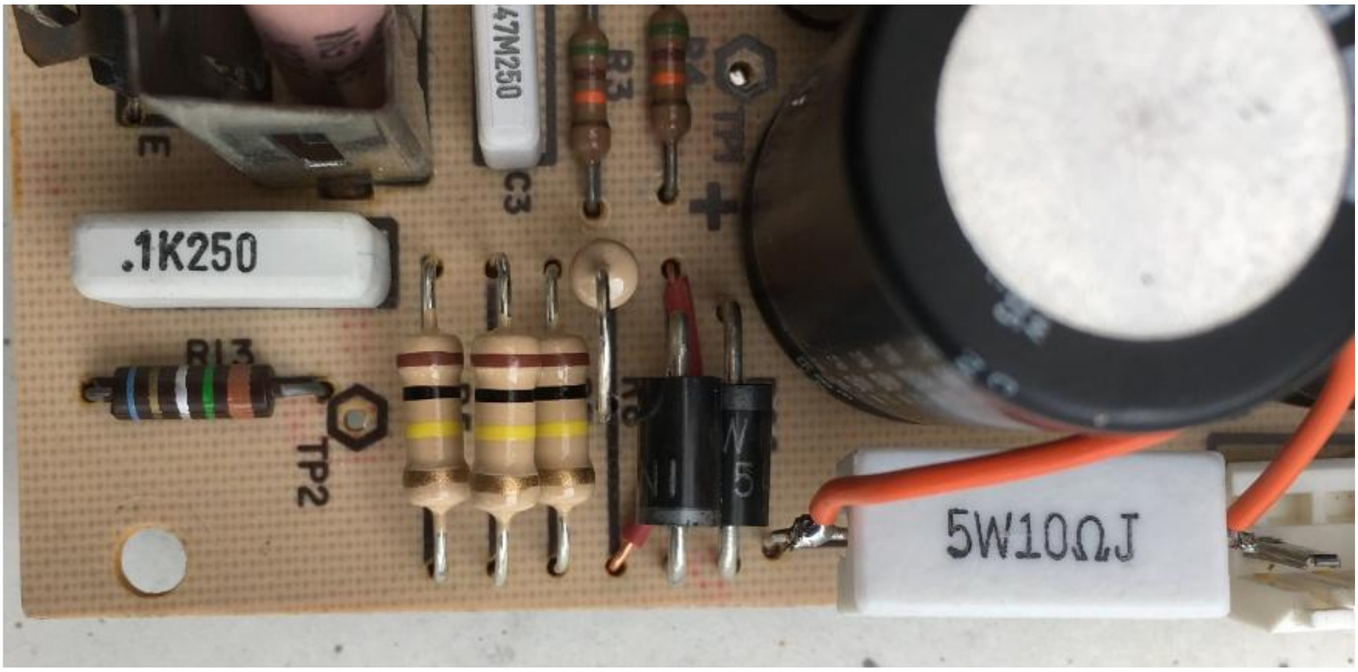


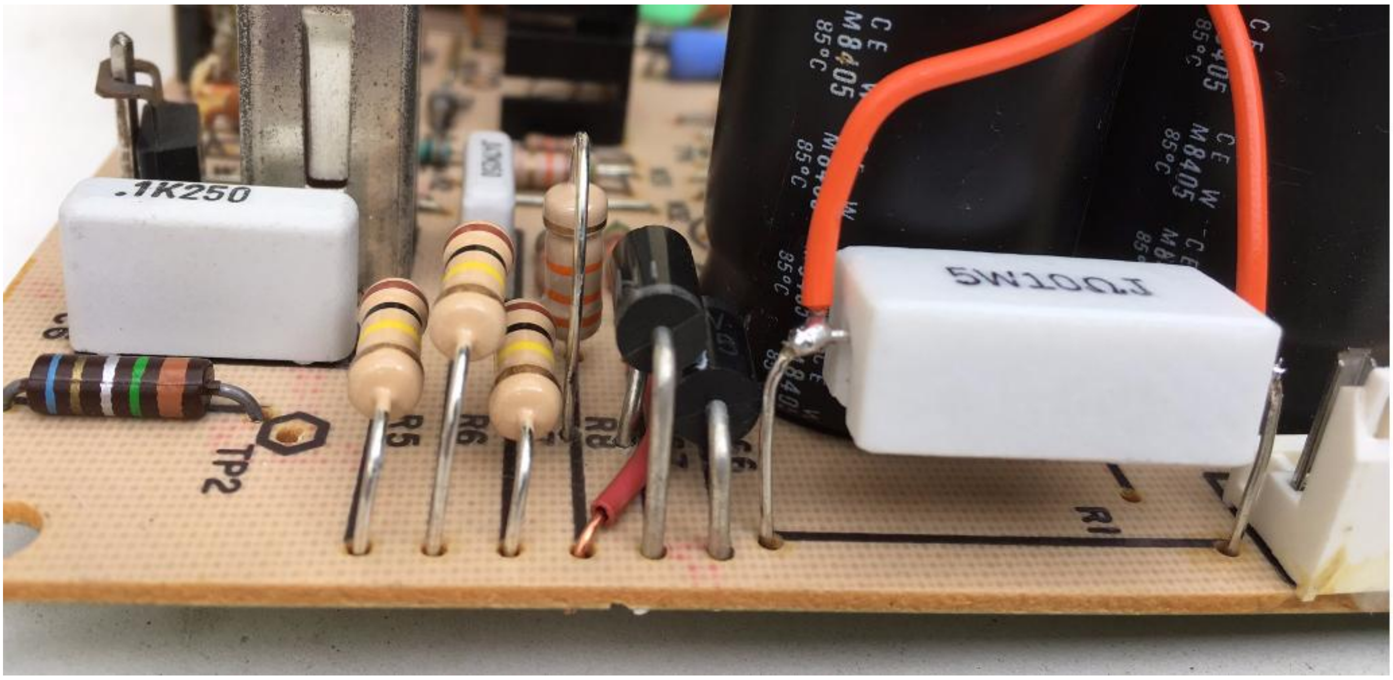


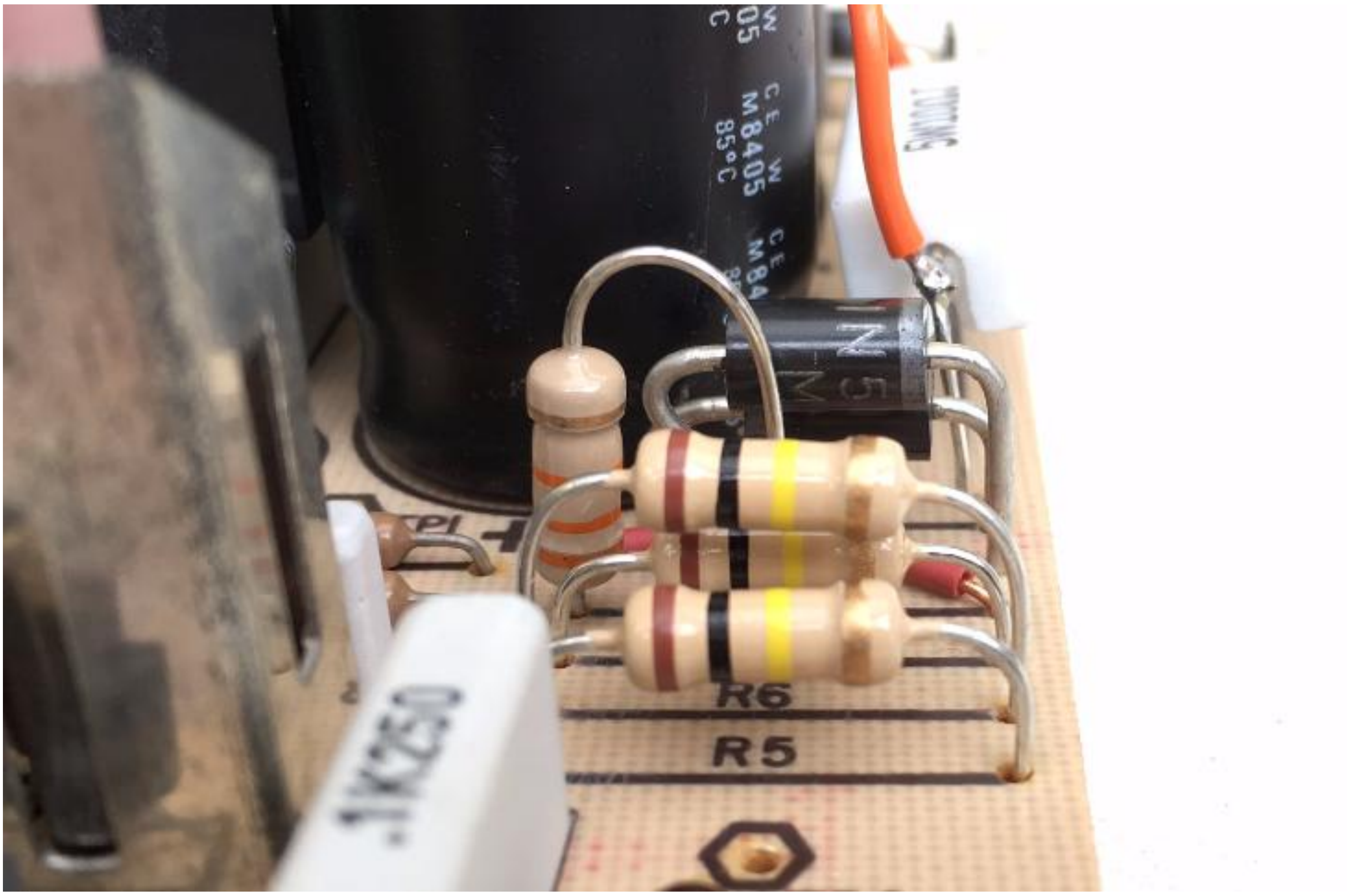

o) Solder (3) 100K 1W resistors in positions R5-R7 as shown in the next three photos.  Keep them off the PCB and clear of each other for good air flow.

p) Solder (1) 33K 1W resistor vertically as shown between one R8 hole and a new hole.

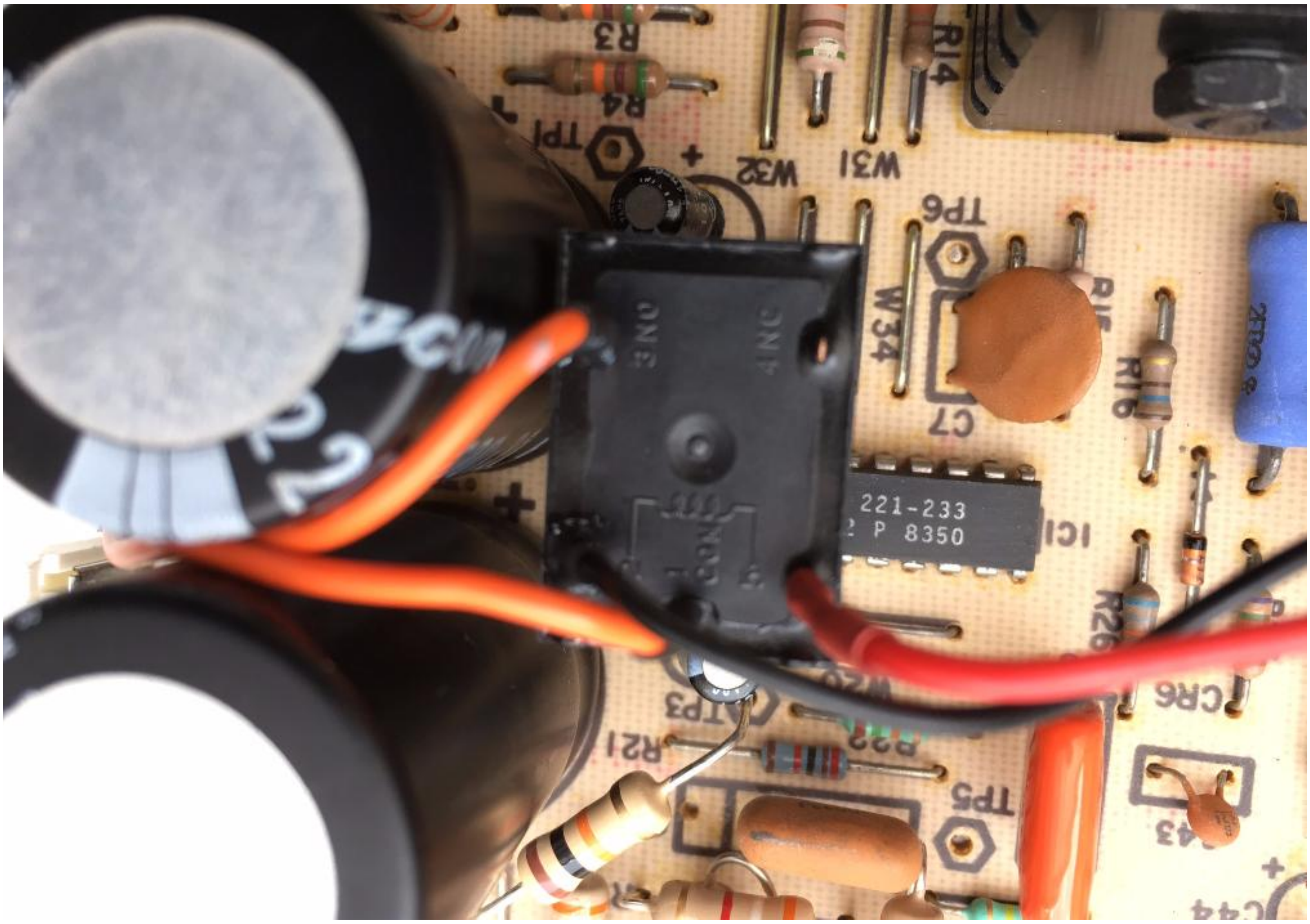
q) Solder (1) 10R 5W resistor at R1 keeping it about 8 mm off the PCB.  The orange wires will be added later.





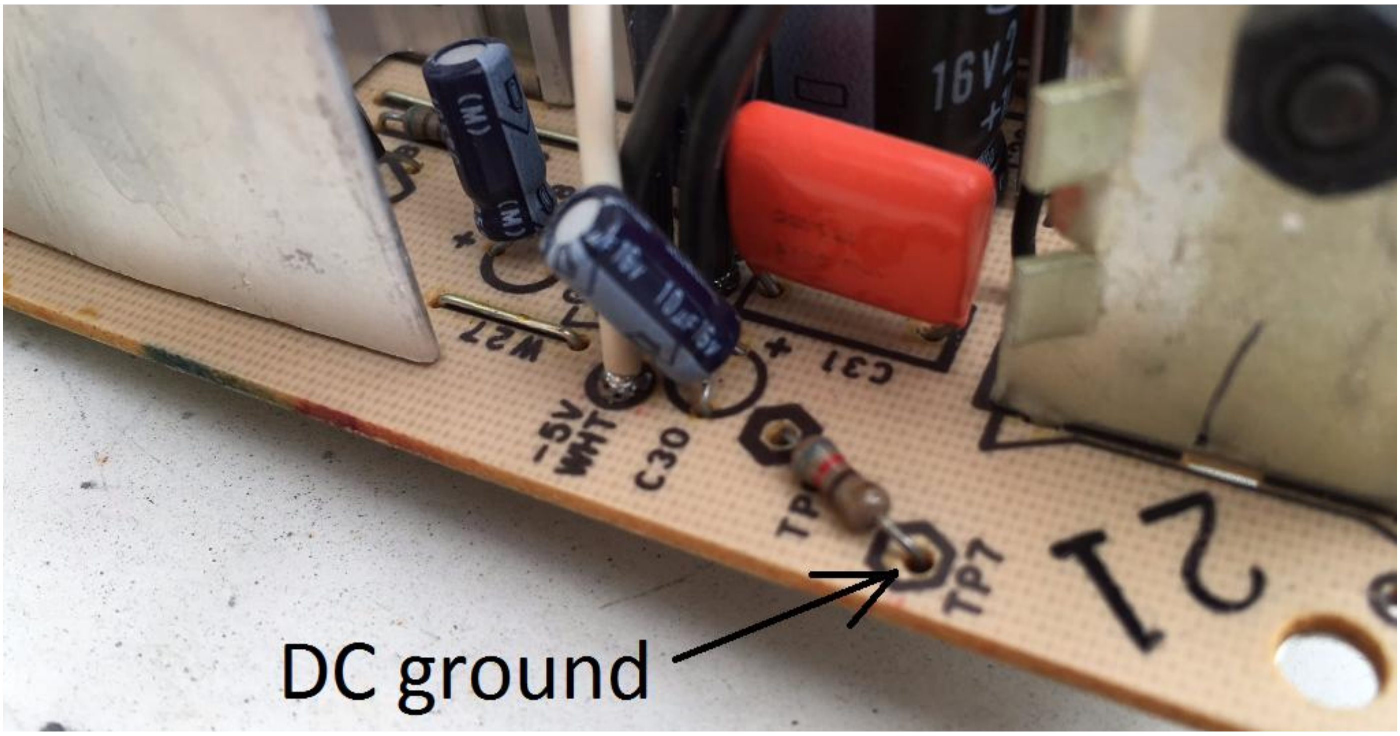

r) Solder and heat shrink (4) 120mm long insulated wires onto the relay's coil (polarity not important,) and COM and NO (normally-open) contacts.  Clip off flush the NC contact pin.  Super-glue the relay on top of the existing white capacitor.



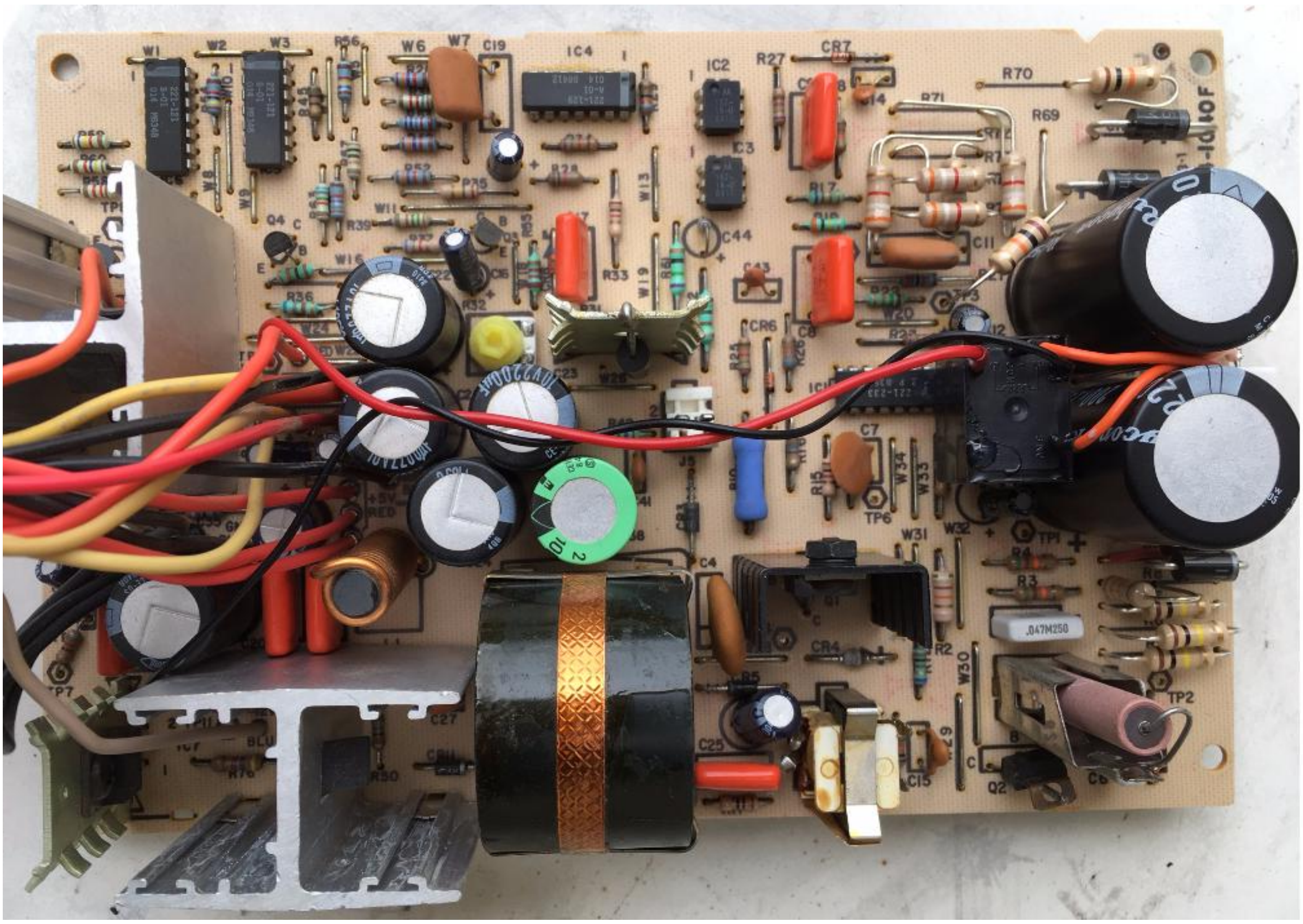

s) Wire the relay contacts across the 10 ohm resistor at R1 as seen in previous photos.

Wire the relay coil wires to the +12 VDC and ground traces on the PCB as shown next. 12 V can be found at TP10, either solder to the pin (if present) or through the hole.

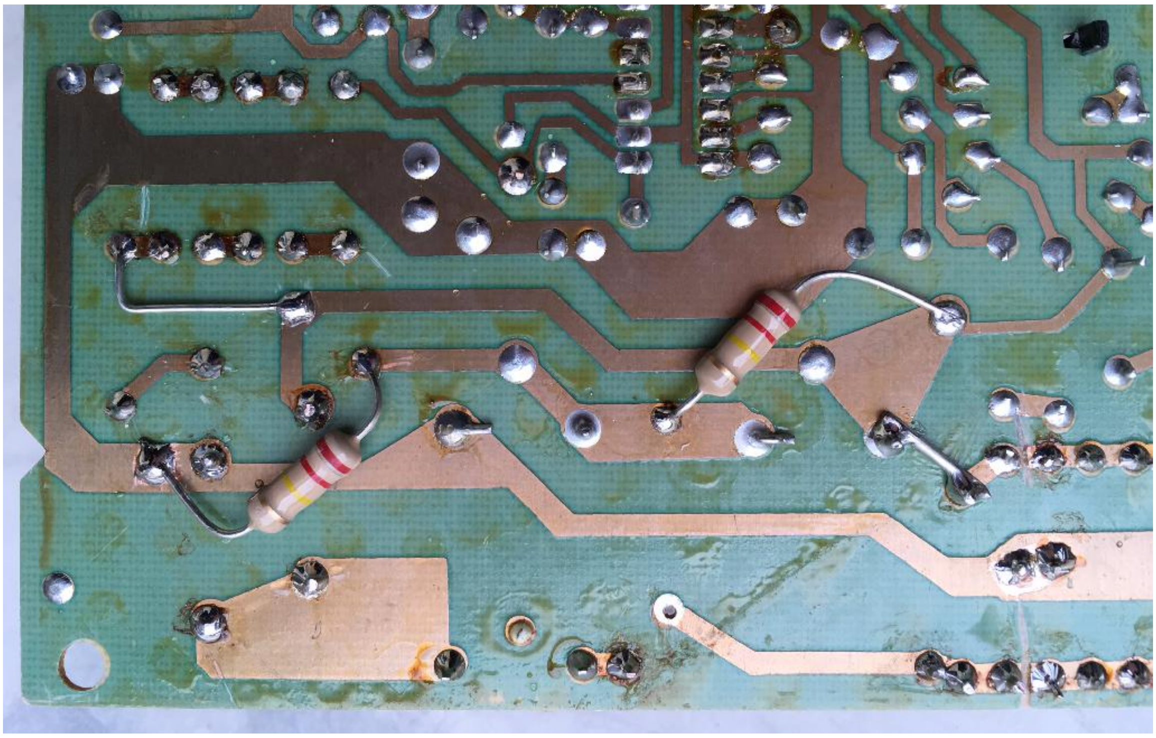
Ground can be found at TP7. If there is a pin there, solder to it. If there is a hole, use that. Otherwise run the wire through the PCB (via any nearby open rectangular hole) and solder to any part of the trace connecting to TP7.




Route these wires away from the heat sinks.



t) Install the (2) 220K 1W resistors on the underside of the main PCB.  The locations are not critical as long as the same traces are jumpered. The resistor lead (nearest the middle of the photo) can be **left sticking through the PCB** to serve as a test point for the middle capacitor voltage which is normally about 162 +/-15 VDC.



## Pre-power-up Tests (main PCB still out of casing)

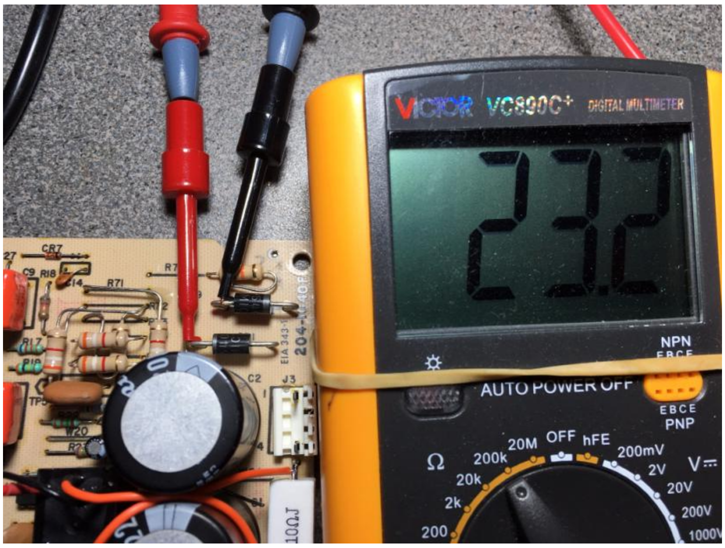
1) Visually check for (2) trace cuts, (2) copper-side jumpers and (1) component-side jumper (locations R7 and R8.)

2) Measure resistance across either IEC connector at rear of metal casing.  With power switch "off" it should be infinity.  With power switch "on" it should be 560K ohms.

3) Measure resistance across white AC-in connector on main PCB.  It should become very high and unstable, perhaps 6 M ohm.  Switch around red/black meter leads and check again.

4) Measure resistance across R1 (main PCB, white 10R 5W power resistor.)  It should be 10 +/- 1 ohms.

5) Measure resistance across the legs of CR1 and CR2 as shown in photo, noting polarity.  Verify resistance after stabilising is 23K2 +/- 200 ohms.



## Reassembly and Power-on Test

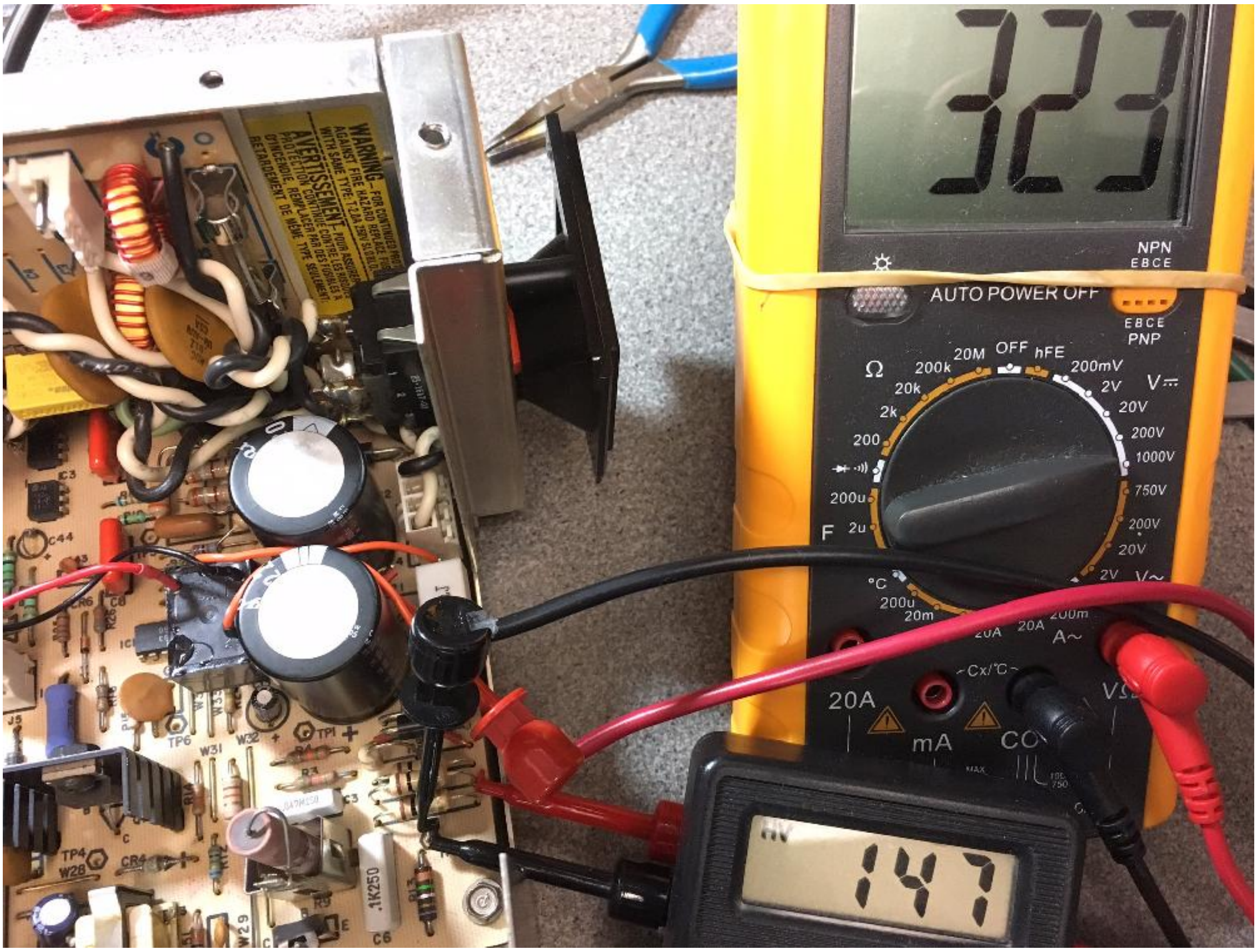
A) Install the plastic insulator under the main PCB if one was present.   If not, make one if you wish.

B) Ensure the plastic standoff is still present and reassemble the main PCB into the housing taking care that components near the IEC connectors are not damaged.  You can check by lifting away the power switch plate and peering in.  Fasten down with the original four screws.  Don't force these - if any seem to bind even slightly, try swapping with another one.

C) Attach a dummy load to the PS.  One 3.5" hard disk is suitable, or a smaller 5.25" such as the quantum bigfoot.

D) Plug the fan in and lay the cover such that the fan faces the main PCB.

E) If you wish to check the important primary voltages put 400V-rated meters here as shown.  TP2 is the primary DC supply 0V reference, any edge legs of R5-R7 is the main bus voltage (325 VDC) and the middle capacitor voltage is present on the lead left sticking through at step (t).  Do not touch your meters or test leads while the PS is powered!



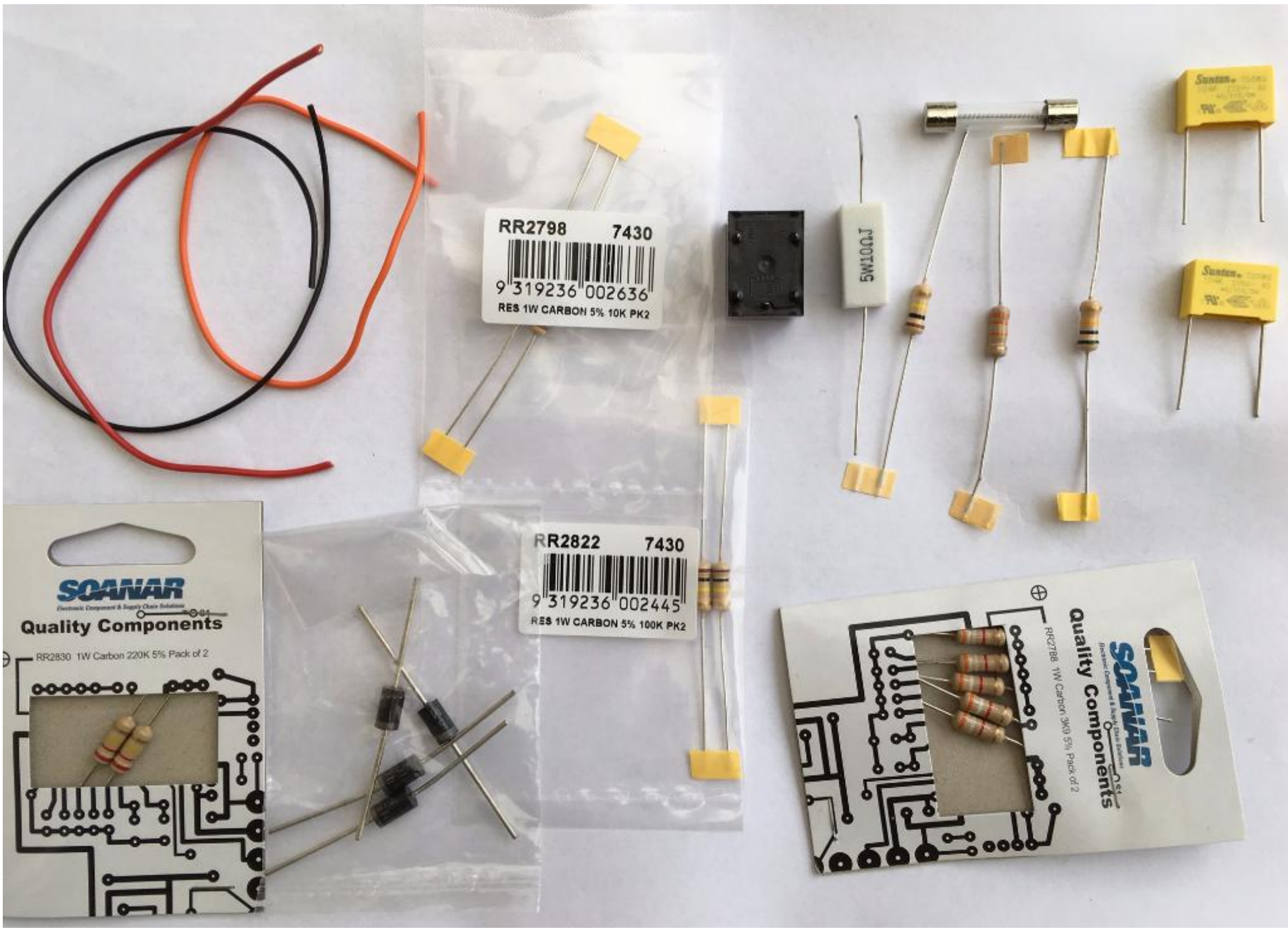

**NOTE - PLEASE BE VERY CAREFUL WHEN UNIT IS PLUGGED-IN**

F)  After power-up on 230 VAC, check 5 VDC and 12 VDC outputs at the P8/P9 connectors.  Adjust using an insulated-shaft or plastic screwdriver on the trim pot.  First set the 12 V rail (yellow wire) to 12.00 then check that 5 V rail (red wire) is between 5.00 and 5.05.  Adjust slightly if required until the 5 V rail is just in range.  The 12 V should end up within 11.9 to 12.1 V.

G) Unplug from line power, reassemble case and install all screws.  Again, if any screws bind try others as they are crudely made and fit slightly differently.  END OF WORK INSTRUCTION

## Reference Section

# Photo of Parts Required

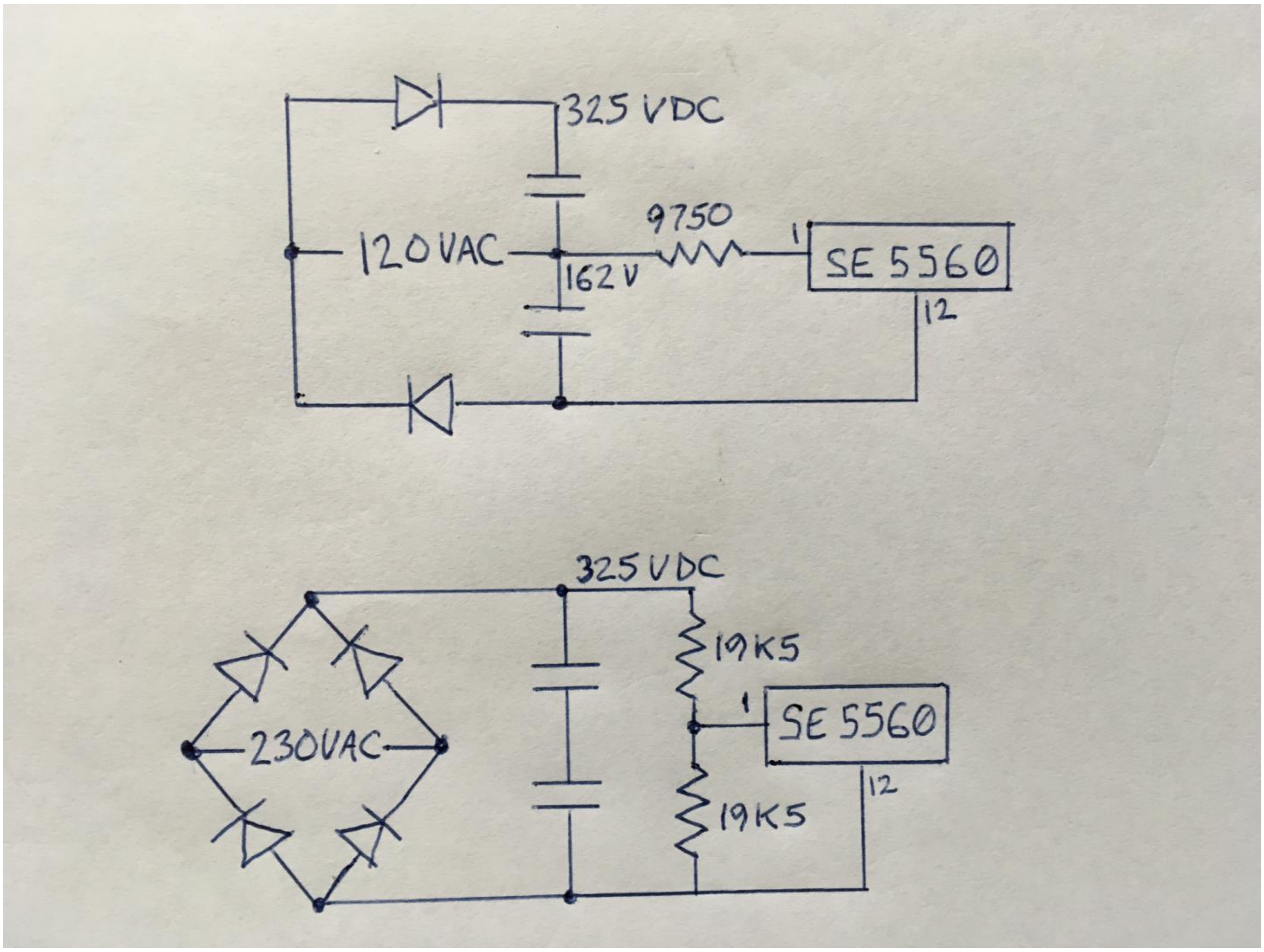



# Theory of Modifications

a) Primary bus rectification and primary auxiliary loads

Fortunately the original primary-side rectification of the 120 VAC line employed half-wave voltage-doubler mode, otherwise this modification would be more difficult. Referring to the next image below, each half wave of the 120 VAC line charges a separate 220 µF 200 V capacitor to 162 V approximately. Wired in series, these capacitors provides a 325 VDC bus with a centre tap of 162 VDC. To minimise power loss the power supply designer chose to power both the SE5560 switch-mode regulator IC and the first-stage flyback drive circuits off this centre tap, via resistor networks to reduce power and subsequently providing a "soft" source impedance. Multiple resistors were used to dissipate the lost power with 1/2 watt parts.

The regulator IC was originally supplied from the 162 V rail via (7) resistors in series with a total of 9750 ohms. The IC is designed to work in what the manufacturer calls current source mode, supplying 14mA, but for reference Sam's measures 23 V at the supply pin on the unit they tested. Heat generated by this voltage dropping network is as follows. 162 - 23 = 139 volts drop across the resistors. 139 volts divided by 9750 ohms = 14mA. $139^2$ divided by 9750 ohms = 2 watts power loss.




So first off we need to convert this primary supply to full wave, which from 230 VAC gives us the ***same*** primary bus voltage (325 V) but we lose the middle rail (162 V.) To supply the regulator IC off the 325 V bus with a simple resistor would require a value of (325 - 23) / 0.014 amp = 21K6 ohms approximately. The new power loss would be $302^2$ / 21600 = 4.2 watts. That's seems doable at first glance but another consideration is that the IC would be subjected to a potential of 325 V until the current rose from zero to the working level. Since we are unsure that this is within the maximum rating (it's not clear from the datasheet,) it's better to use a voltage divider instead, such that the IC sees 162 V at zero current, ***exactly*** as in the original design.

So, given that *any* two equal-value resistors in series can divide the 325 V bus in half, how do we choose appropriate values for this application? As it happens, a voltage divider circuit made up of two 19K5 (= 2 X 9750) ohm resistors in series (totalling 39K) across 325 V has the ***same*** source impedance as the single original 9750 ohm resistance off a 162 V bus. The impedance at the centre of the divider network is calculated as though the two divider resistors were wired in ***parallel***.

The calculation for impedance at the centre of the divider is: 1/[(1/19k5)+(1/19k5)] = 1/2 x 19k5 = 9750

As such, the regulator IC will see the same voltage drop characteristic as it did before, at any current draw, and we do not need to consider the engineering specifications pertaining to the IC.

In terms of power loss, during normal operation of the IC the upper half of the divider dissipates $(325\text{-}23)^2$ / 19K5 = 4.7 watts, while the lower half dissipates $23^2$ / 19K5 = 0.03 watts. So, two items result from that: in total we will see 2.7 watts *more* power loss with this modification over the original design, and second, using a divider over a simple series resistor costs us only 0.53 W.

In order to conservatively utilise 1W resistors, the top side is made up of (5) 3K9 in series, the bottom side (2) 10K in series. A series resistor network was chosen to best utilise the existing PCB traces in that part of the circuit.

The flyback driver circuit is treated similarly but calculated to allow the load to take the divider centre all the way to zero volts. Using a divider does not expose the first stage flyback driver transistor to any more voltage than it saw before (and that limit is unknown,) and provides the same ***source impedance*** (important, given that this circuit is highly dynamic.)

The divider is made up of two 33K resistors in series, unloaded dissipating $325^2$ / 66K = 1.6 watts, fully loaded the upper half dissipates $325^2$ / 33K = 3.2 watts, 2.4 watts over the original design, the lower half zero watts. In order to specify 1W resistors, the top side is made up of (3) 100K in parallel, the bottom side (1) 33K. A parallel resistor network was chosen to best utilise the existing PCB traces in that part of the circuit.

**So far, we are looking at 5.1 additional watts of power loss.**

The new 1 W resistors are suited to natural convection cooling despite the presence of the cooling fan. Any failure mode that concludes with the loss of 12 VDC will of-course stop the DC fan (where fitted,) therefore this is a sensible approach.

One further change: the original half-wave rectifier design used fast-recovery diodes but the reason is unclear. It may have been thought to reduce EMI noise, but at a mere 50Hz they seem pointless, so for this modification they are replaced with four 3A 1000V standard diodes. The two existing smoothing caps in series get 220K ohm balancing resistors to encourage (but not ensure) that neither are subjected to over-voltage.

<u>b) Inrush Current Limiting</u>

The existing AC inrush current limiting resistor of 2 ohms would need to be raised to 8 ohms to reduce current proportionally. Since the line voltage is doubled the resistor value must be doubled in value to maintain the same current, however, since the fuse is *also* downsized by half, we need an *additional* doubling of the resistor. The losses of an 8 ohm resistor would be excessive so I've added a relay to bypass the resistor when the 12 VDC rail appears. This is the same method used in the IBM 5160 PS and **recovers 1.5 watts previously lost**. To gain further value, I've specified a 10 ohm resistor to reduce inrush to a theoretical 23 amps. Clearly it does not need to be a 5W part now, but again I'm thinking of failure modes. Our **total power loss** for this 230V conversion stands at: 5.1 - 1.5 = **3.6 W**.

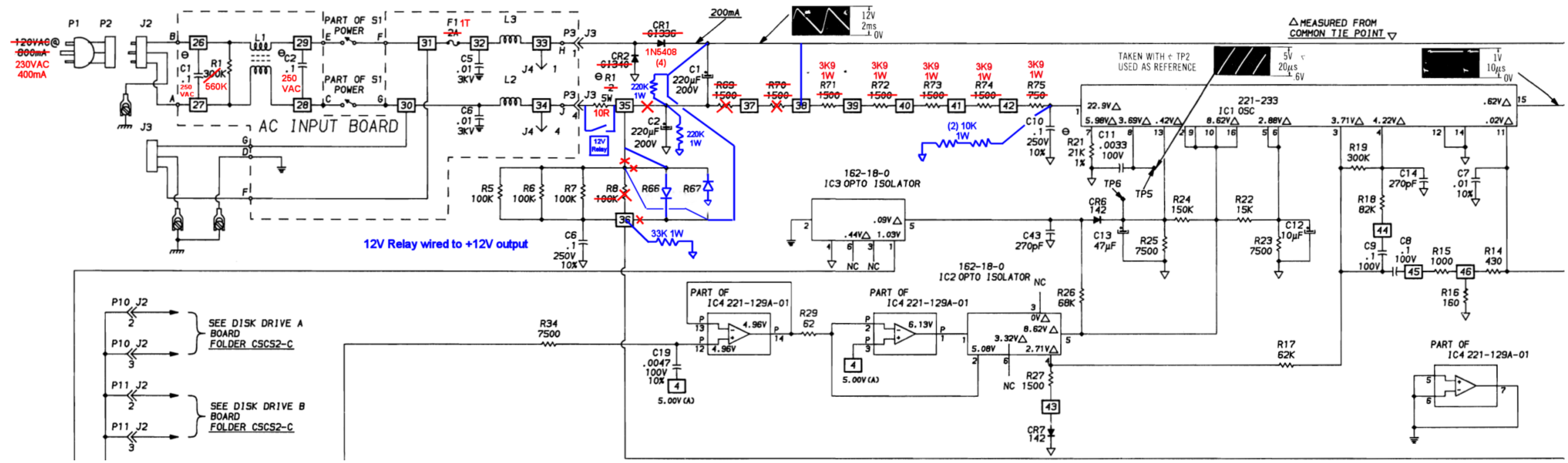
<u>c) EMI Filter Components</u>

On the EMI filter board the X2 caps are replaced with 250 VAC agency-approved parts as the original parts were only rated to 120 VAC. The X2 capacitor bleed resistor size is doubled, as it should be, and the T2-amp 3AG fuse is now T1-amp. The dielectric rating of the common-mode inductor is unknown but it helps that the current is now half what it was before.

*** END OF THEORY ***

<u>Revision history</u>

1.0 Initial release, 8 March 2017, P.A., New Zealand



AC INPUT BOARD
PART OF S1 POWER
230VAC 400mA
1N5408 (4)
200mA
3K9 1W
12V Relay wired to +12V output
221-233 IC1 OSC
TAKEN WITH ⏚ TP2 USED AS REFERENCE
△ MEASURED FROM COMMON TIE POINT
162-18-0 IC3 OPTO ISOLATOR
162-18-0 IC2 OPTO ISOLATOR
PART OF IC4 221-129A-01
SEE DISK DRIVE A BOARD FOLDER C5C52-C
SEE DISK DRIVE B BOARD FOLDER C5C52-C