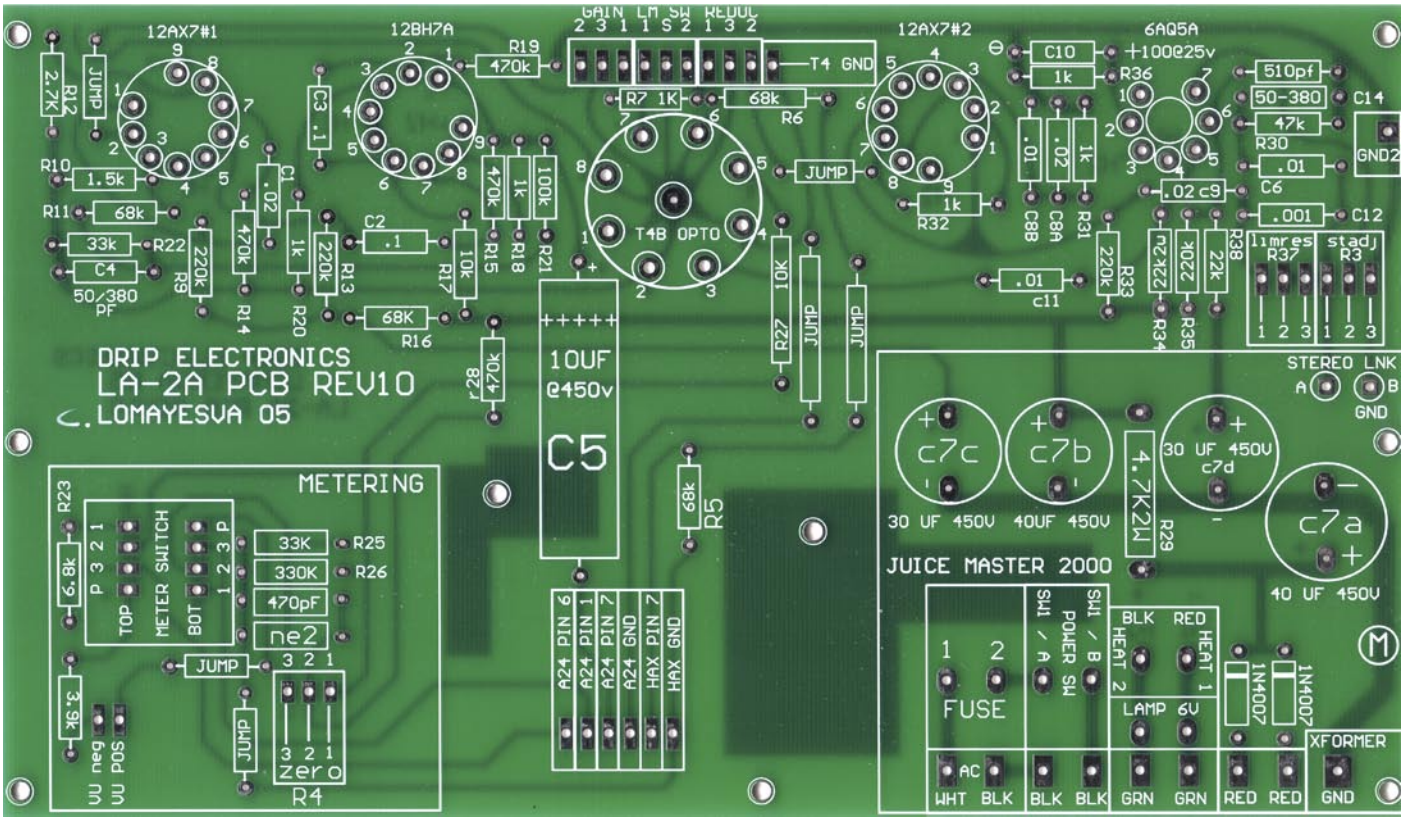
La-2a pcb build manual ver 1.0

Section one : populating the board



attention|disclaimer







this project is presented as artwork and is solely intended as such.

although this board can be assembled and built into a functioning LA-2a compressor.

due to the high voltages and possibilities of human error , drip electronics | gregory lomayesva hereby assumes no liability for injury/damage/loss which might unintentionally occur.

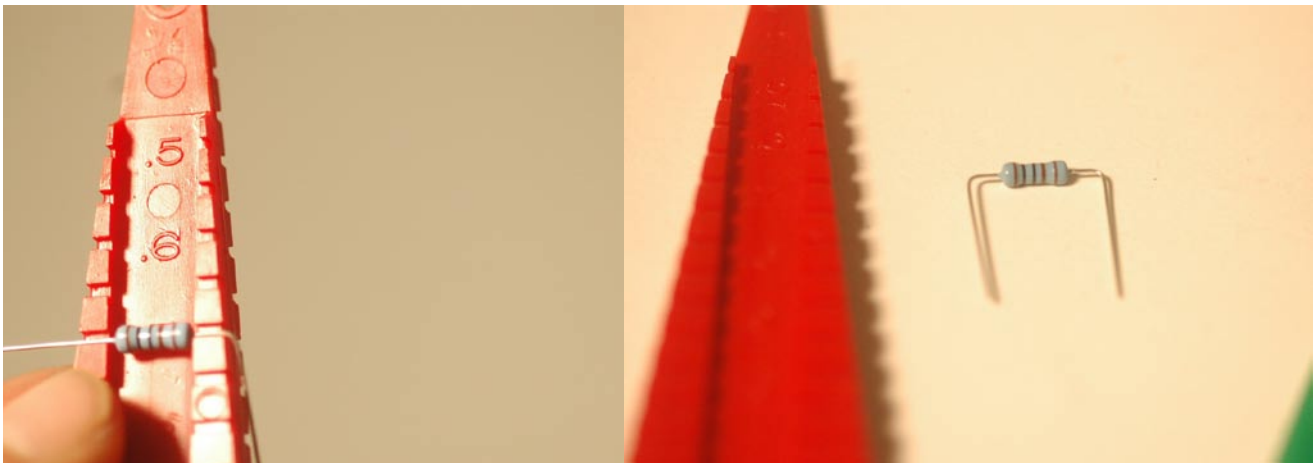
Section 1 Populating the board

Order of build :

1 place resistors, jumpers and diodes
2 place tube sockets and octal socket
3 place capacitors and variable capacitors
4 place other components

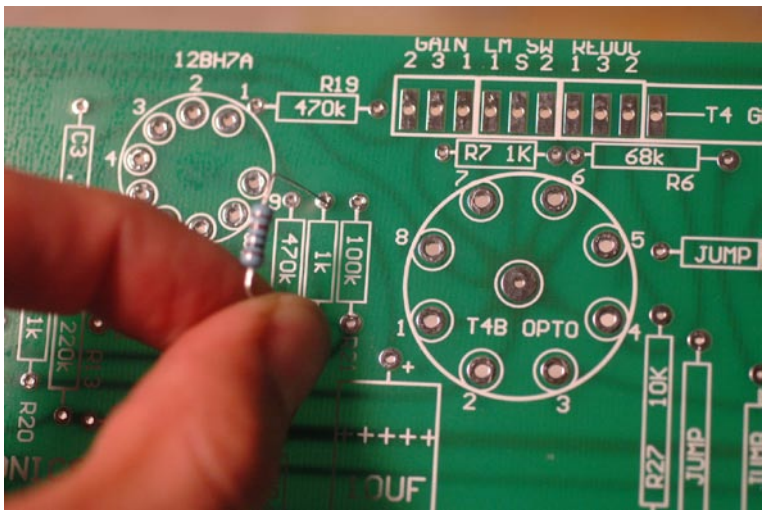
#1 placing resistors and jumpers

Here it is a good idea to use a resistor bending tool.



This makes the process much faster and keeps the resistors straight and tight, most of the resistors are to be bent At the same length.

The holes for the Resistors should be able to accommodate older style carbon resistors with larger leads as well as more modern resistors with thinner leads.





when placing a resistor , it is important to double check the value . Depending on resistor manufacturer the resistor can have several color bands around it.

Using a resistor code wheel or an on line color code calculator , it is a good idea to check the code to make sure it corresponds to the correct value.

Errors in this area can have damaging effects , so please double check each placed resistor.

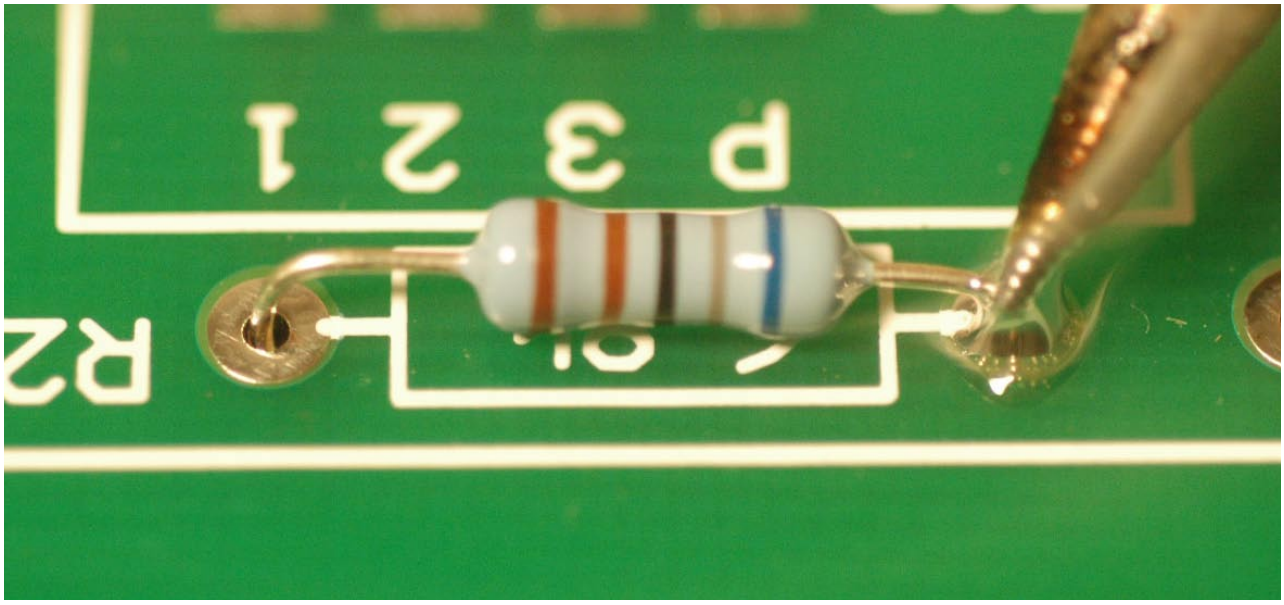
(this will save hours of trouble shooting and prevent damage to other expensive components including your self)

most all of the resistors used in this project are 1/2 watt. there are 2) 2watt resistors.

It is ok to use larger watt resistors , but in some areas the spacing will be very tight.

Each place for a resistor is marked by a square with a value in it (i.e. '220k') and a 'R' with a number next to it, this number is used to reference that particular resistor.

Bend and place the corresponding resistor in each of these box's.



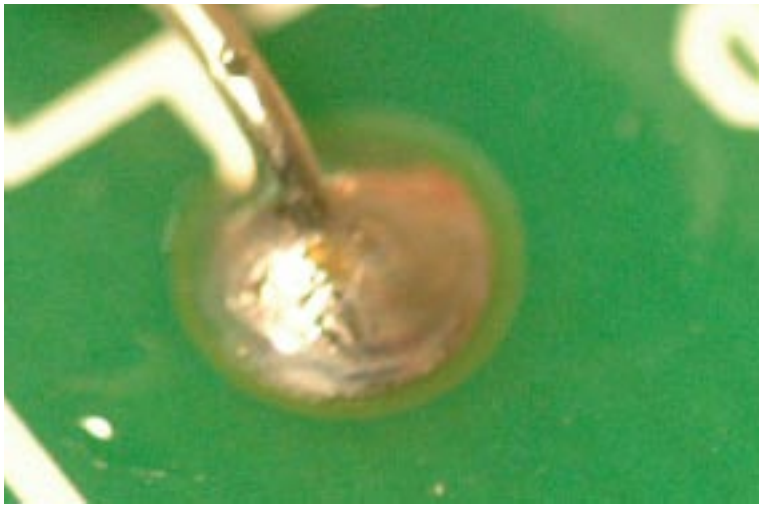
Once the resistor is placed , solder the leads to the pads. You should not use too much solder here , just enough to nicely attach it to the pad.

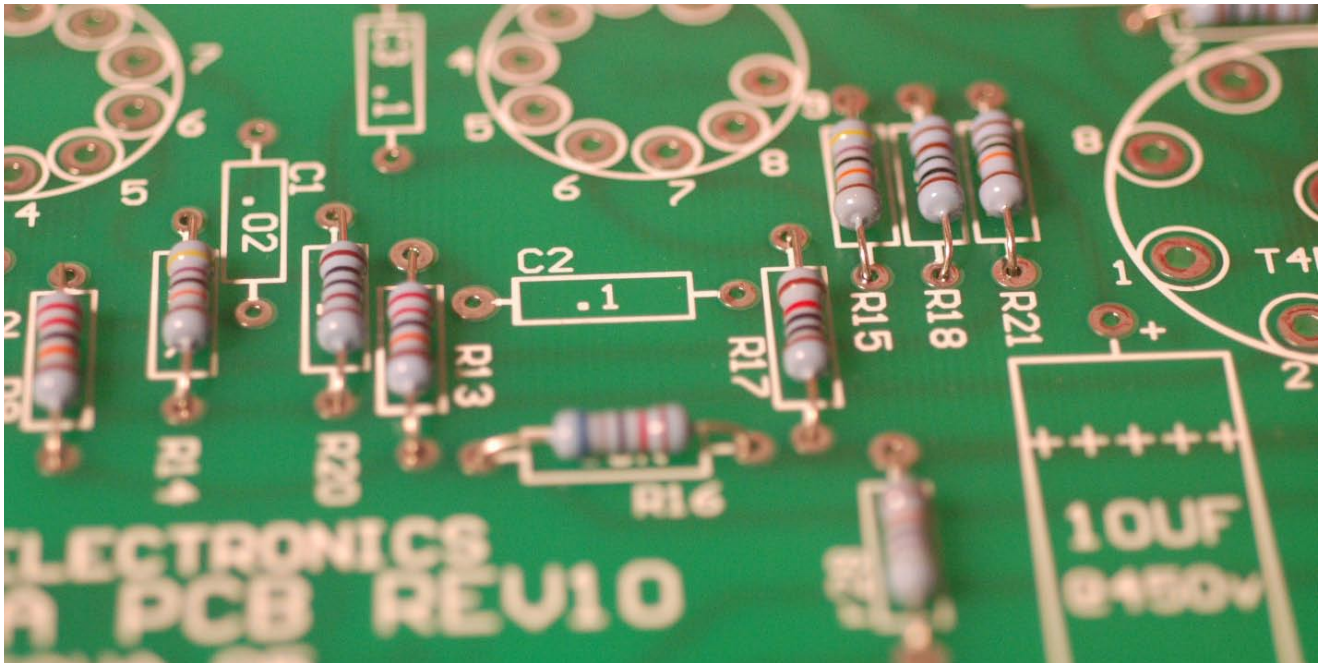
The pads are plated ‘through hole’ so you can solder on the top or on the bottom.

Soldering on top seems to be much easier and less time consuming.

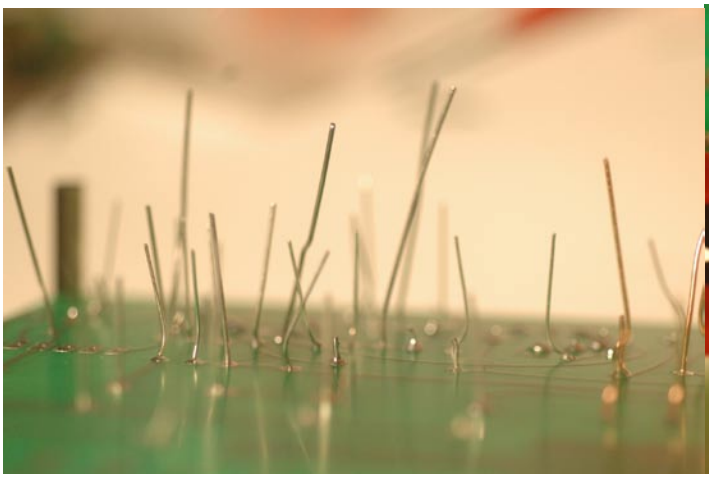
keep in mind that if you put excessive amounts of solder while heating the pad and lead, it will simply flow to the bottom side of the pad and make a big solder blob.

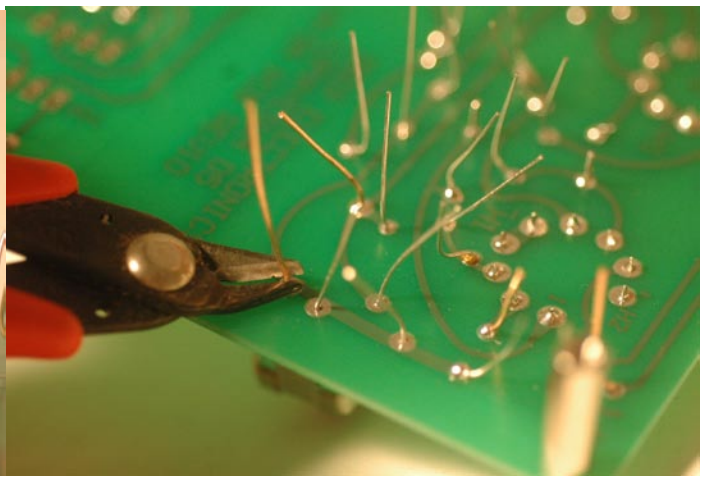
here is an example of a correctly soldered lead.





After you have placed and soldered all of the resistors, flip the board over and snip off all the excess leads coming off the bottom of the board.



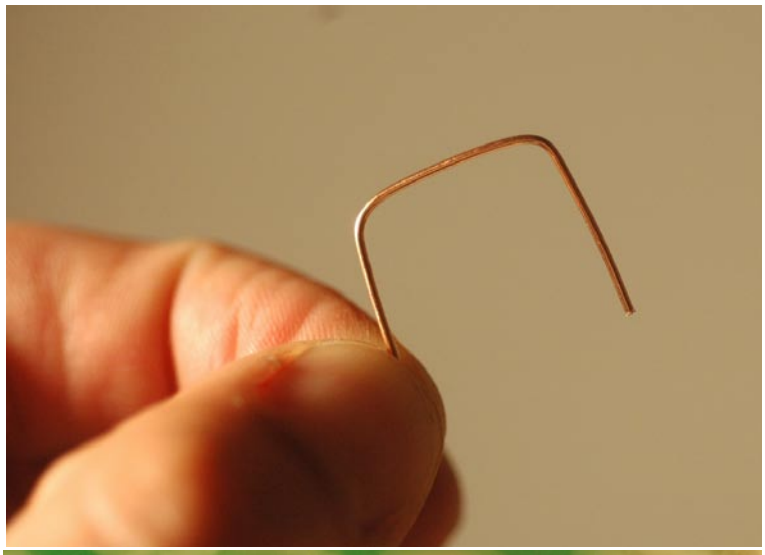


At this point , check and make sure all places for resistors have been filled.

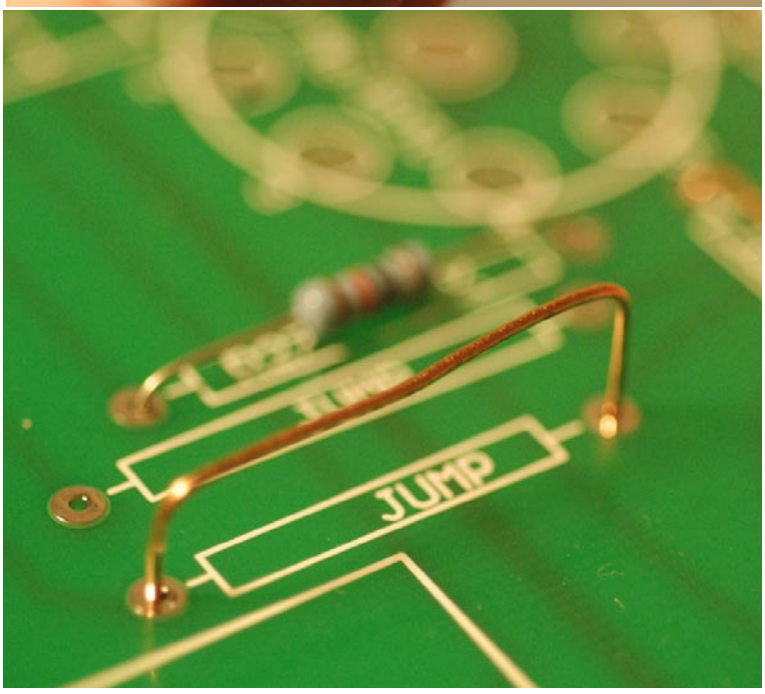
Jumpers



Simply strip a piece of copper wire and bend like you would a resistor.



Then solder in to place



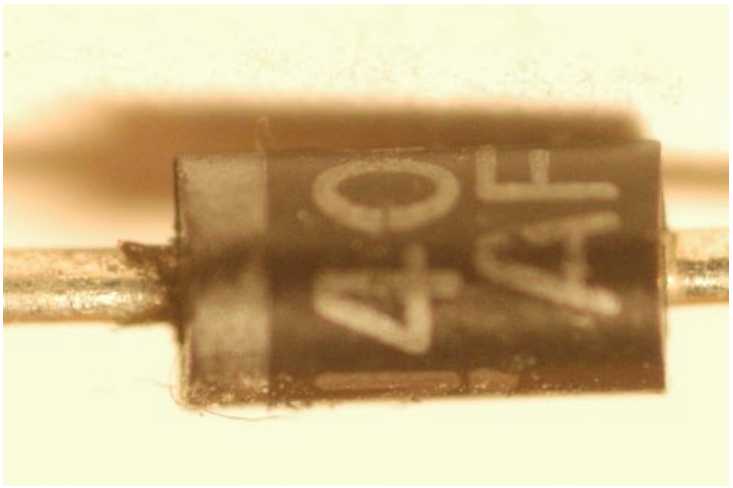
Diodes

The diodes type is 1n4007

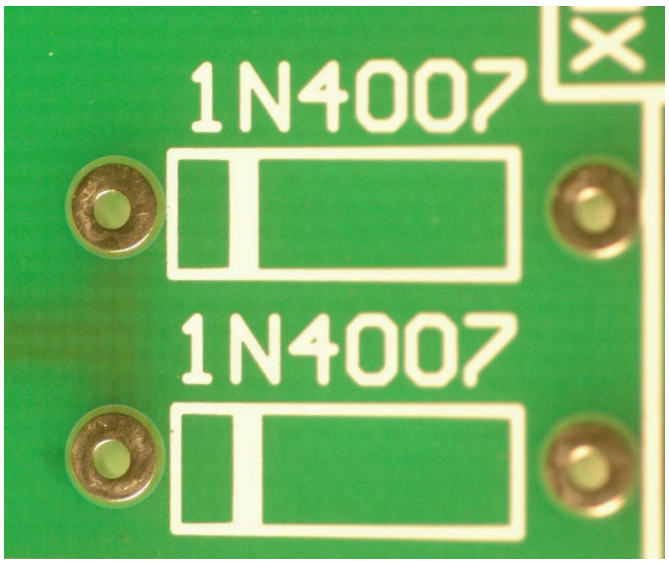
They are one way components and must be placed in the correct direction . .



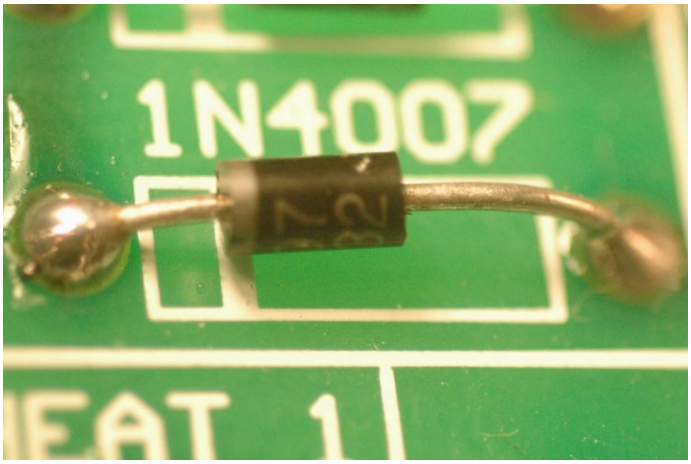
Notice the grey line on the Diode.



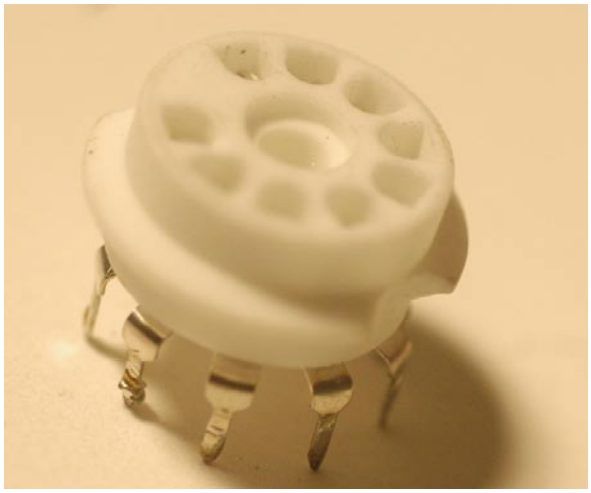
The footprint on the pcb also has a line on it similar to the diode.



Bend the diode like a resistor and solder in to place.

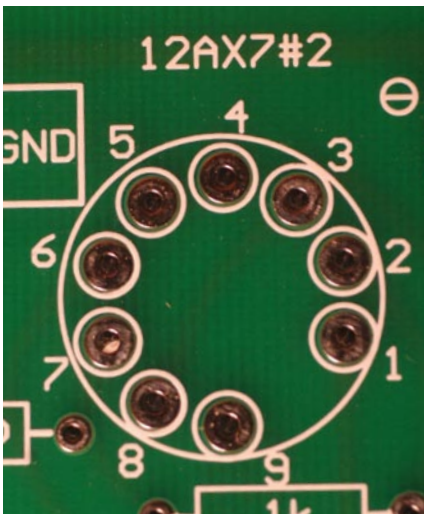


Tube sockets

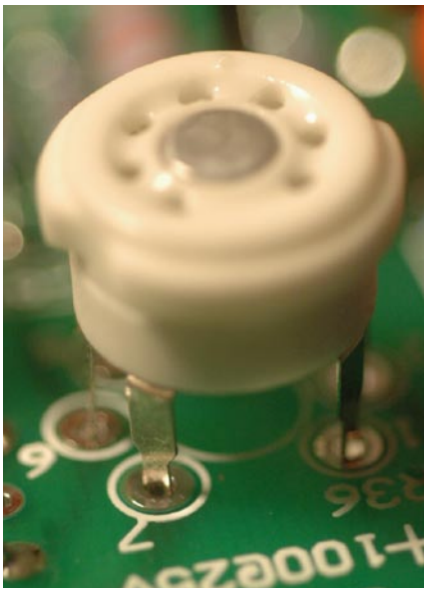


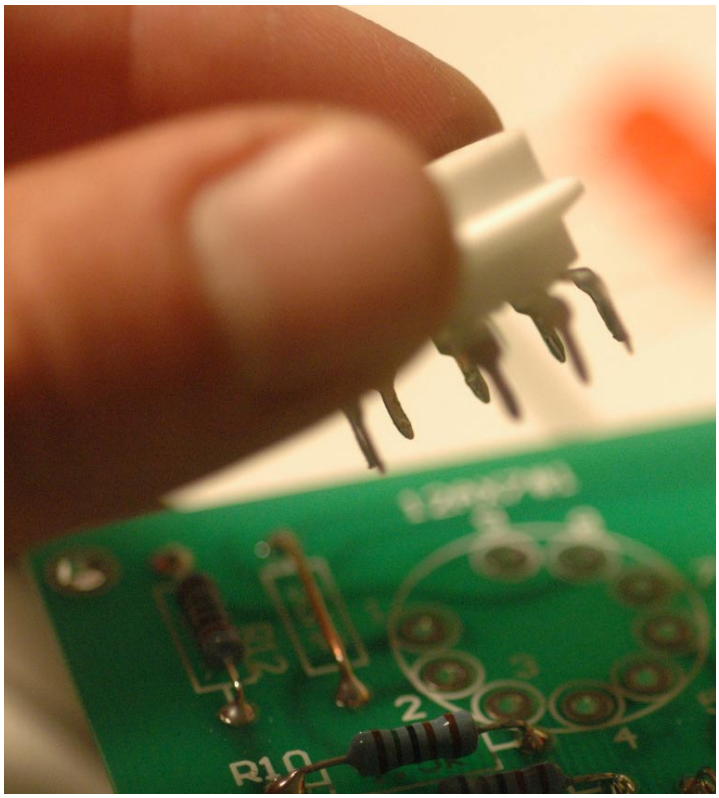
On the board you will find four places for vacuum tube sockets.
Three sockets are identical. they are the 9 pin type.
for 12ax7#1 , 12BH7A , and the 12AX7#2 vacuum tubes.
One of the sockets is a smaller 7 pin type for the 6aq5a tube.

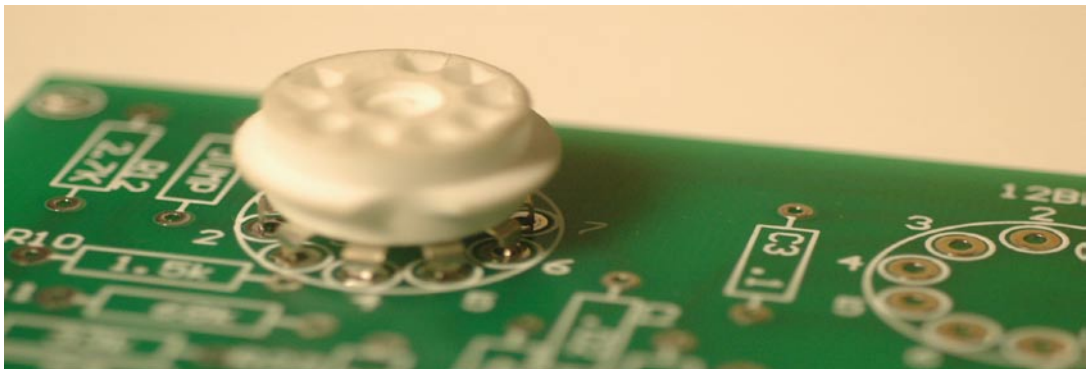
Here you will simply align the tube sockets to the holes on the PCB.
The pins of the tube socket should slide right in to the holes.
You can use a tooth pick to nudge in the pins if needed.

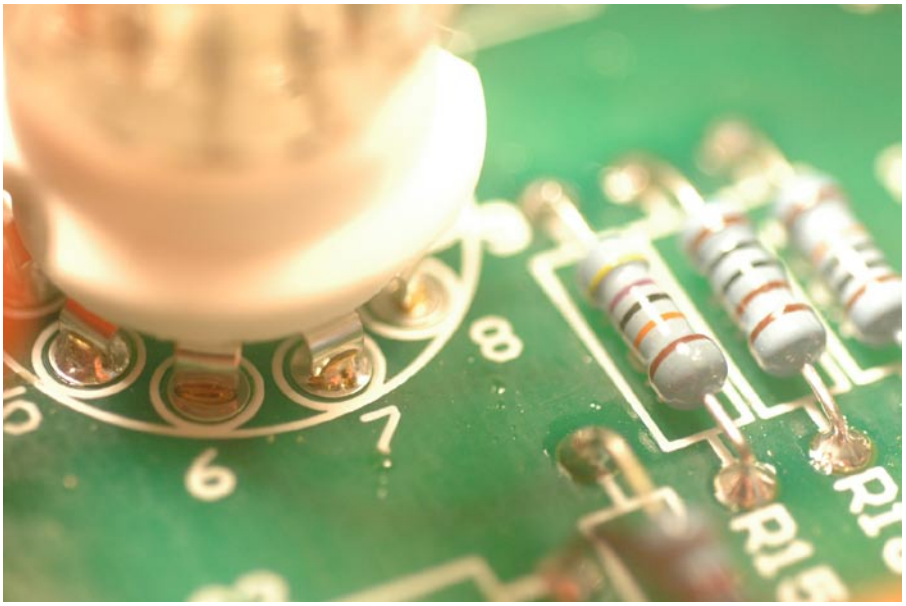


Some pins of different tube sockets brands will not compleatly go all the way through the hole .
As long as the pins are all aligned correctly and resting on their designated pad.
Put a little pressure on top of the socket , and solder from the topside of the board.
Using enough solder to make a good solid connection between the pin and the pad.




R10
2
4

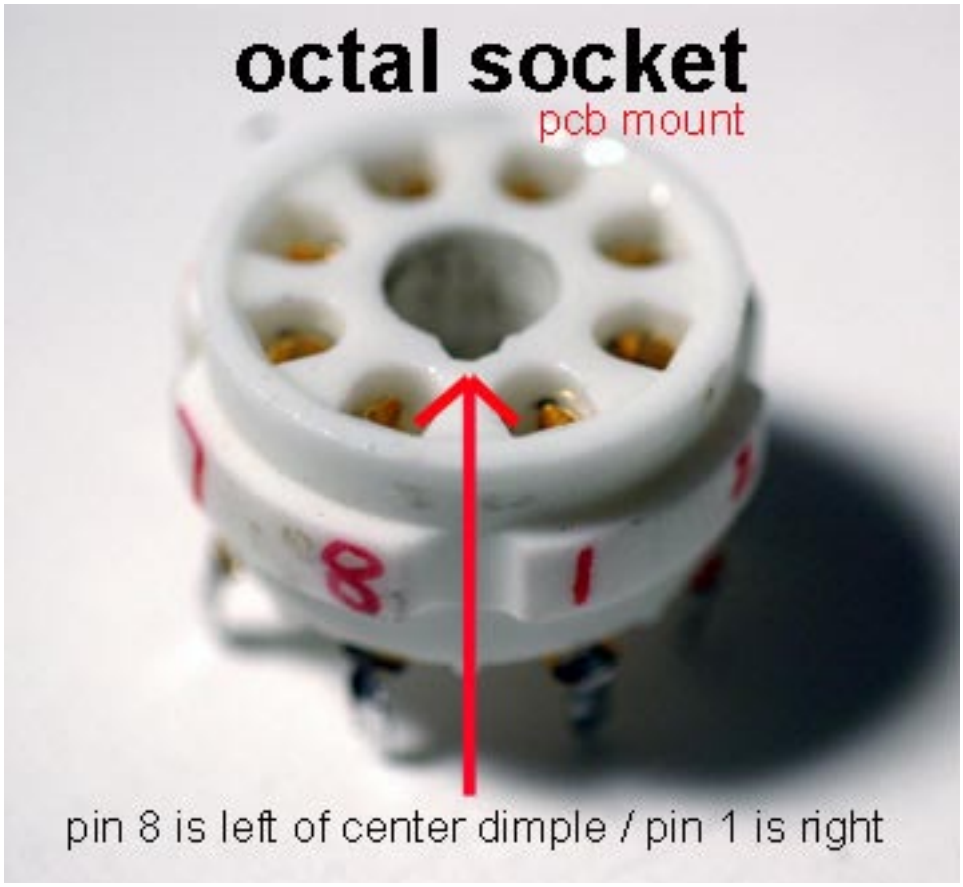

2K2
1.6K
R10
2
6


8
7
6

Octal socket

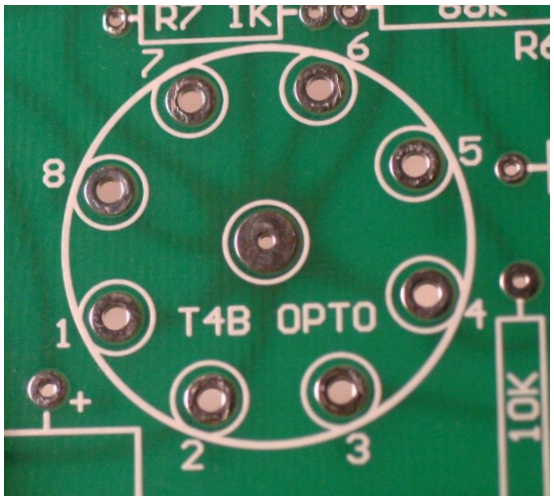
The larger 8 pin pcb mount octal socket usually comes un numbered.

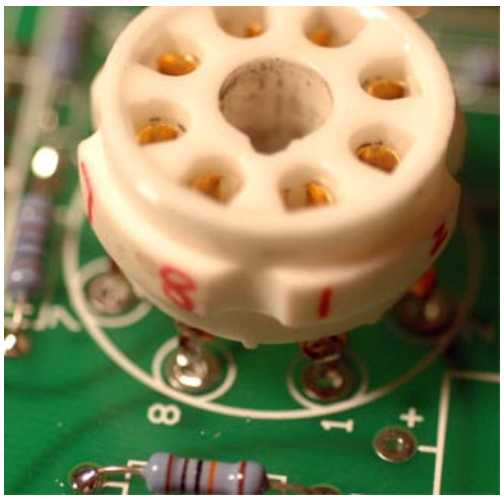
You will need to align this socket correctly or you could damage the expensive T4B optical cell.



You will need to look at the top of the socket , you will see a hole in the center of it and small dimple. Like in the illistration , pin 8 is to the left and pin 1 is to the right , use a marker and mark these two pins. Once this is done , double check your markings , then place the socket on to the pcb in the area designated T4B Opto.

Align the pins of the socket so they rest on their pads. press down and solder each pin to its pad.

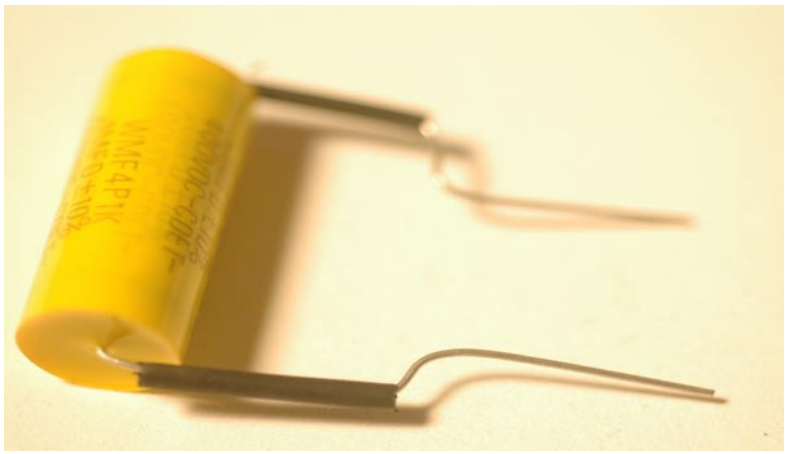




Capacitors

After you have completed the tube sockets and double checked your resistors , it is time to start placing the capacitors.
For the capacitors , you have many different choices on brands and sizes ,
Depending on what you use, space can be an issue.
in some areas it is a good idea to use some very thin heat shrink tubing on the leads , to make sure there are no potential shorts
with other components.

illustrated here is an example of using heat shrink tubing on the leads of a .1 capacitor.



electrolytics

There are 5 electrolytic capacitors that you will use on the board .



These are polarized and can only be placed in the proper position . They could potentially explode if they are placed in reverse.

Illustrated here is and example of how the negative lead is marked.

Notice the arrow with the minus sign in it .

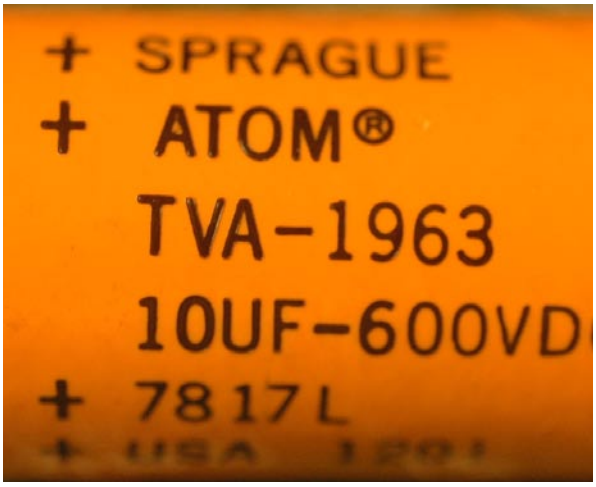
This designates the NEGATIVE lead.



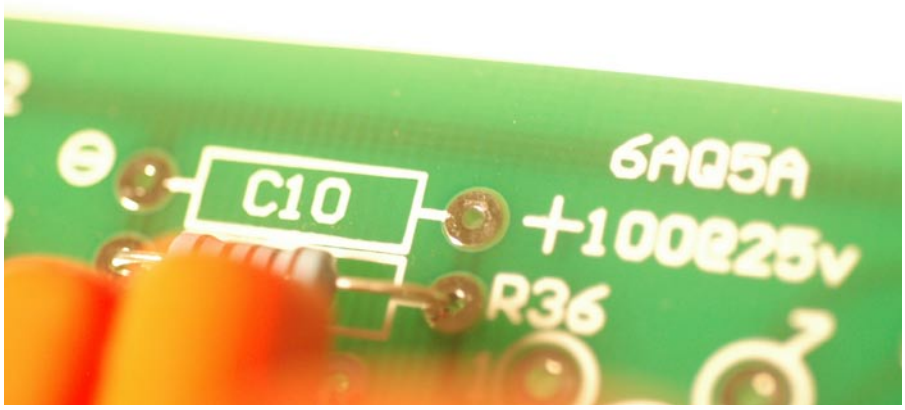
Illustrated here is an example of a larger axial leaded 10uf@600vcd (C5)

On this cap the POSITIVE side is marked with the plus signs , designating that side as positive.

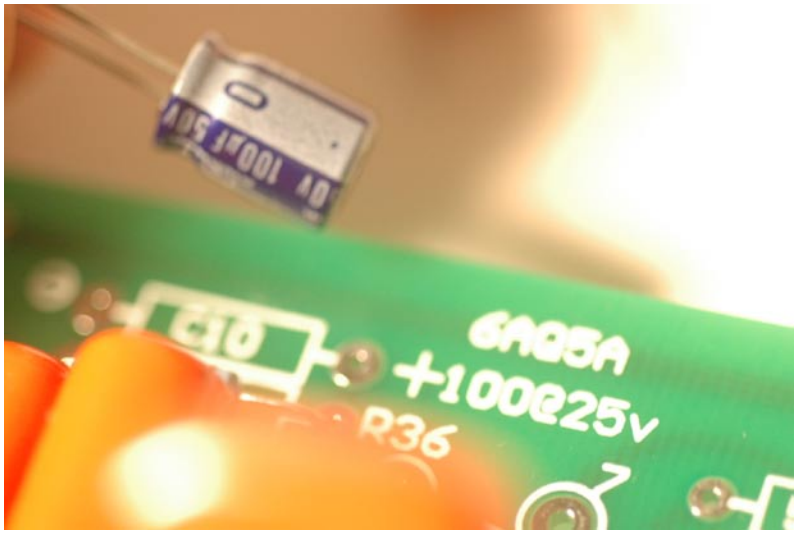
So the opposite unmarked side would be NEGATIVE.



Here it is illustrated how the pcb is marked. for capacitor c10 , There is a plus sign next to the 100uf@25v that side is positive.



Notice the negative sign on the cap. Designating that lead as negative. So you will place the the opposite lead into the pad on the pcb marked ‘+’ .



Another example of how the pcb is marked. The pad marked with the plus sign is for the positive lead.



The electrolytics are as follows:



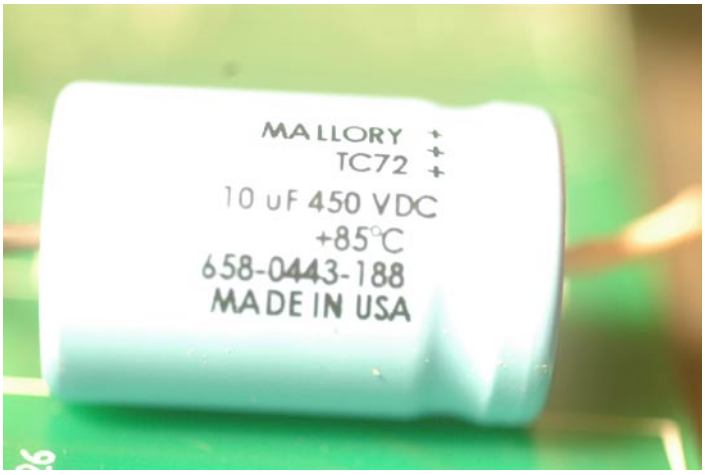
C5

This cap is a 10 uf @ 450v electrolytic. (axial leaded)

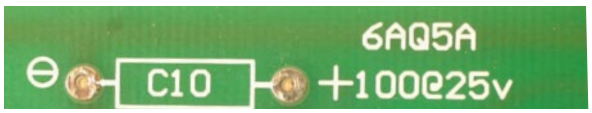
This cap can have a lot of effect on the tone of the la-2a .

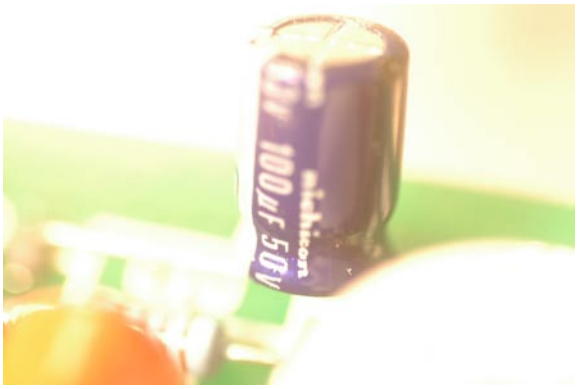
The reissue la-2a uses the mallory tc72.

sprauge atoms can be used here also.



C10 100uf@25v





In the reissue this cap is a 47uf@25v.

On one of the schematics it is a 50uf cap.

Using either seems to work just fine here.

(Radial or axial leaded work here.

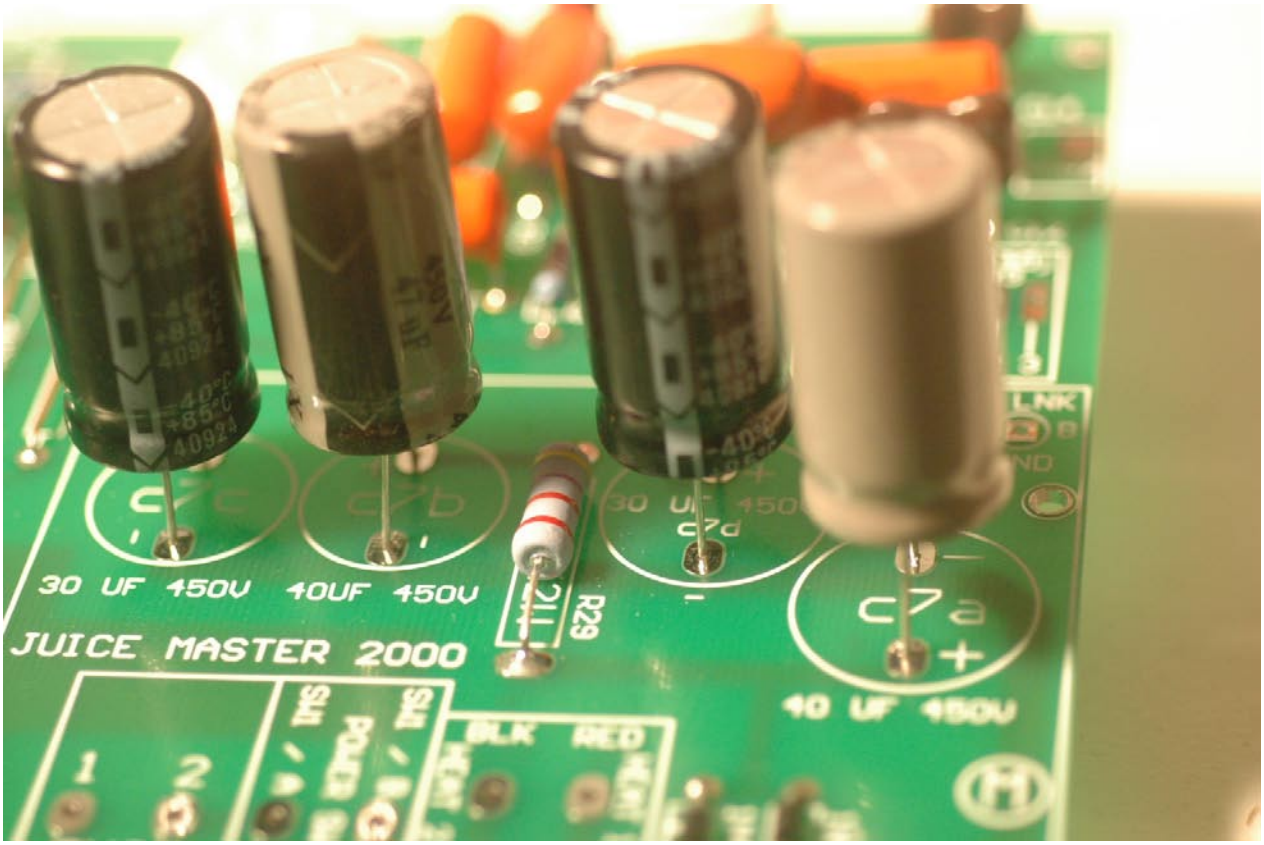
C7a/C7b 47uf@450V(radial mount)

On the schematics and drawings , these caps are 40uf@450v , 47uf work and are readily available from most sources.

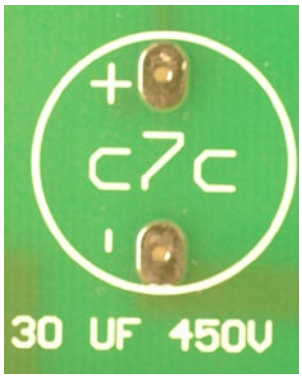




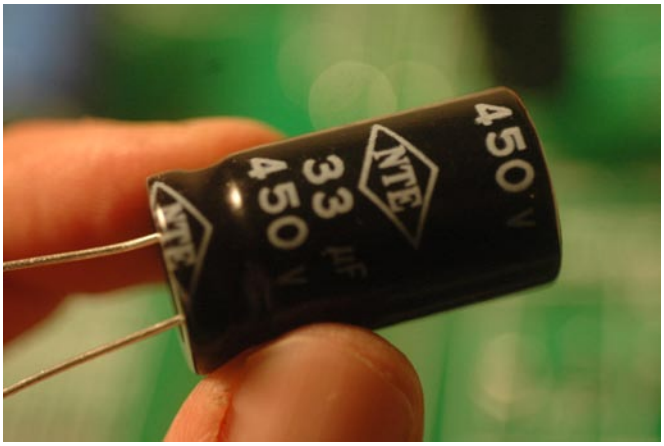
It is important to examine the position of the plus(+) symbol on the boards silk-screen , For on some it is on top and others on bottom.



c7c/c7d 33uf@450v







(Radial leaded)

On the scematics , these are 30uf@450v

33uf@450v work here.

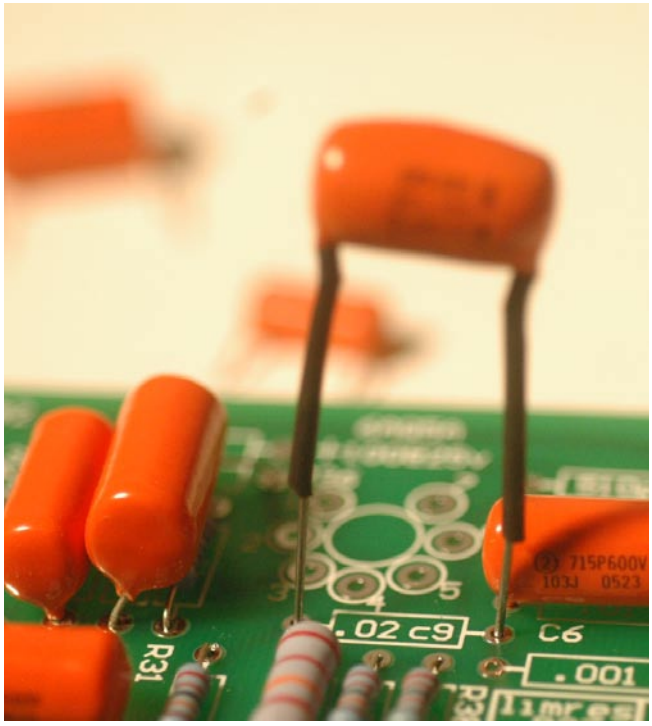
The reissue la2a use 22uf@450 here.

Please double check your electrolytics!



Non polarized capacitors

The pcb was designed with ceramics in mind , But have found that most other caps fit just fine. Depending on the type of caps you choose to use you might have to add a little bit of shrink tubing so the capacitor may be elevated or bent away from a tight set of other components.



The non polarized capacitors on the original la-2a are ceramics/mica/and polyester. There are several different brands you can substitute in for these , sprague orange drops are often used in place of the ceramics. But ceramics are very good for conserving space.





Here is a list of the capacitors and the type used on the original la-2a/re-issue



c1 = .02 (ceramic z5u)

c2 = .1 (Metallized Polyester Film) WMF4P1K

c3 = .1 (Metallized Polyester Film) WMF4P1K

c6 = .01 (ceramic z5u)

c8a = .02 (ceramic z5u)

c8b = .01 (ceramic z5u)



c9 = .02 (ceramic z5u)

c12 = .001 (mica)

c13 (not marked on board) 510pf (sits above c14) (Here you can use a 500 pf mica or ceramic)

c? Unmarked 470pf above NE-2 lamp. (Here you can use a 500pf mica or ceramic)



arco 465 variable capacitor



C4 and C14 use the arco 465 trimmer capacitors

C4 seems to be the more important place to use this,

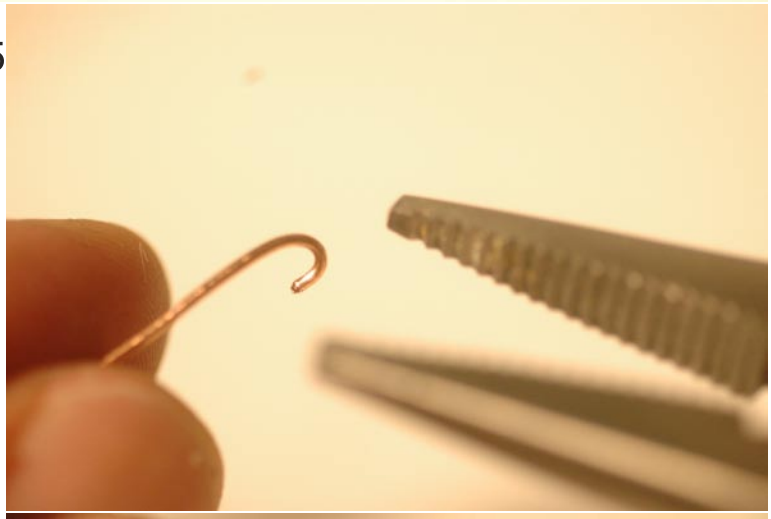
You will adjust your la-2's frequency response with this trimmer.

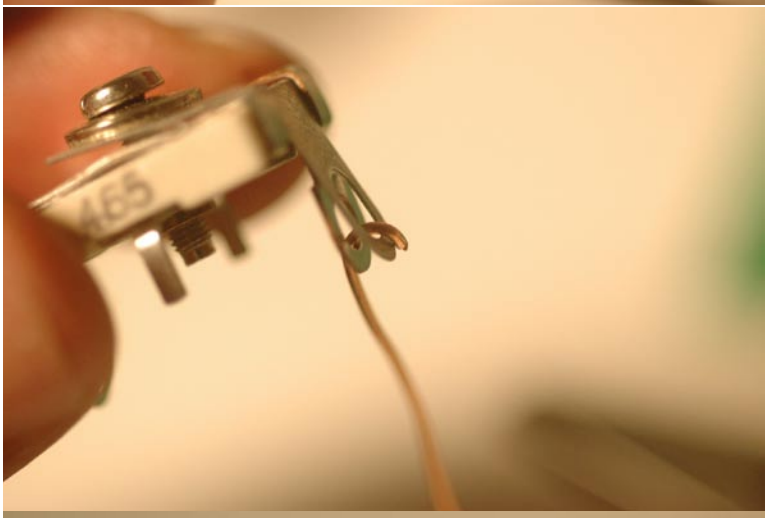
As where c14 can be a fixed value of 300pf or so.

You will need to solder a wire to each lead of the arco.

This is done by stripping a piece of wire and bending one end of it into a hook.

Then you will attach it to a lead on the trimmer , and press with pliers.

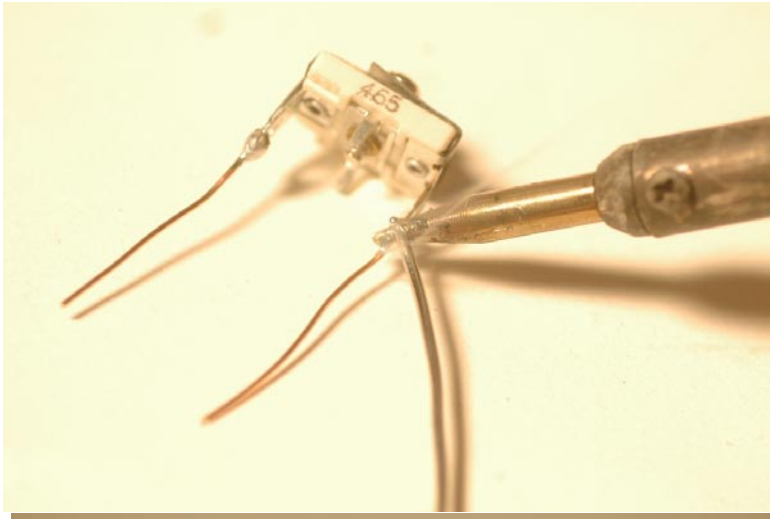




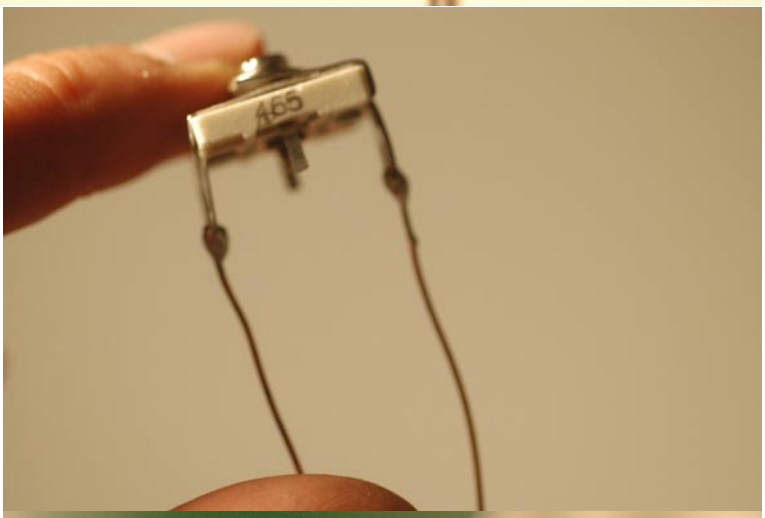


After you do this for both leads ,

Solder each lead to the wire making a nice solid connection.



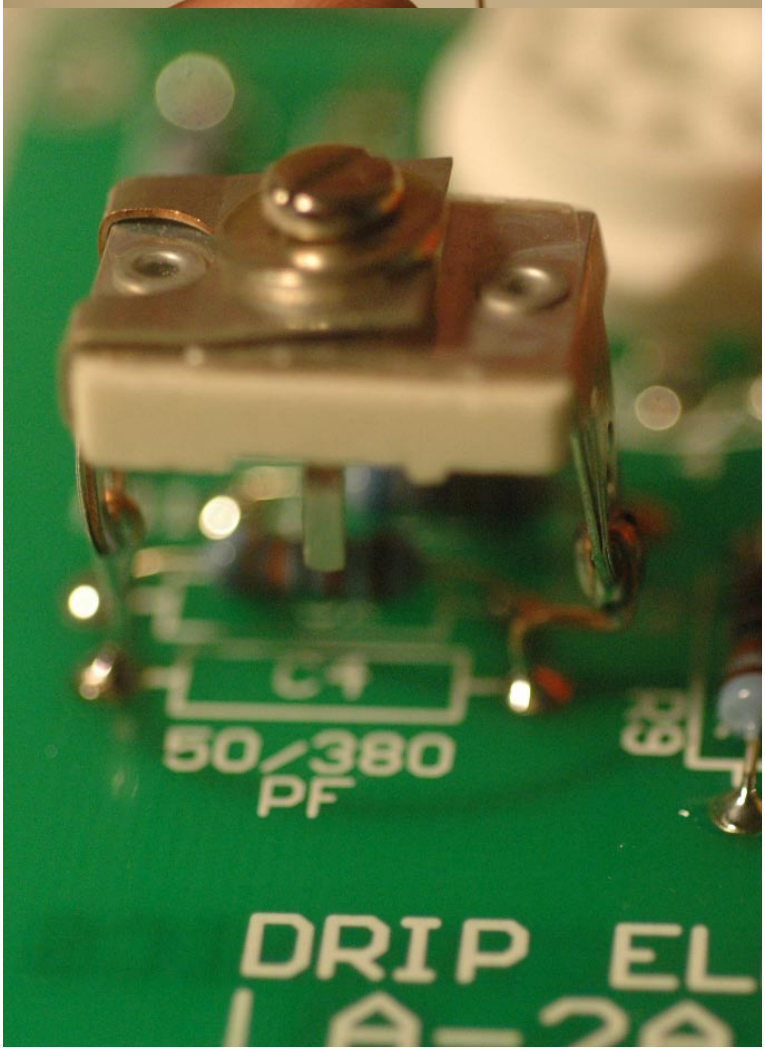
Straighten both the wires.



Then place into the footprint marked C4.

You might have to bend the leads a little bit to make it sit just right.

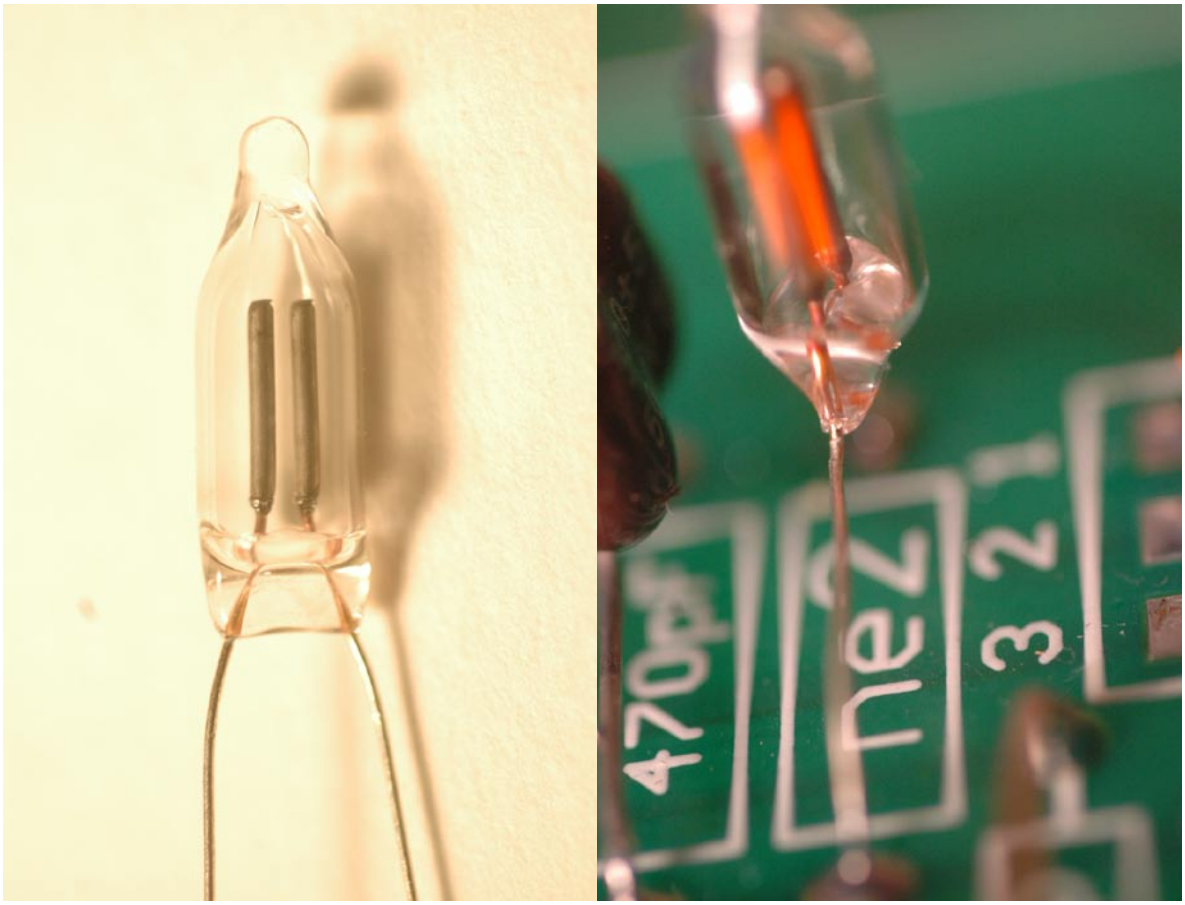
then solder.



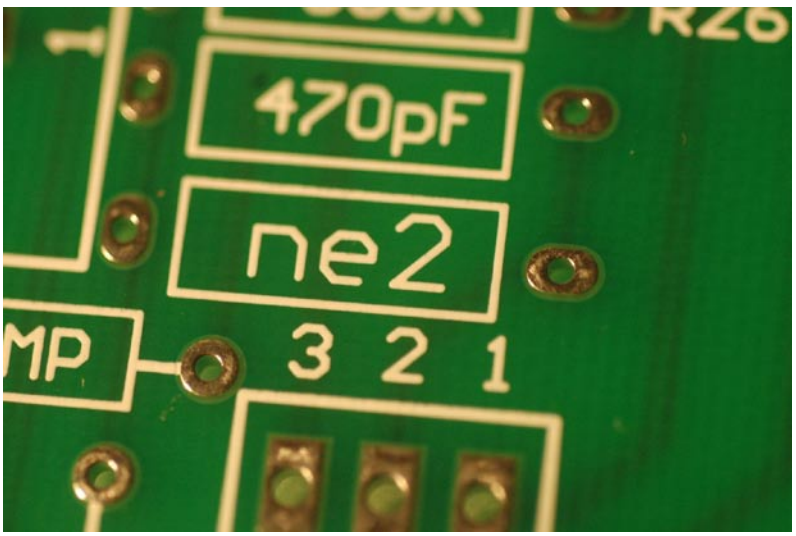
If you wish to use this for C14 , just repeat the steps and solder in to place once again.

Various parts to install :

NE-2



This is a small neon lamp
Simply place the leads
into the pad holes and solder
into place.

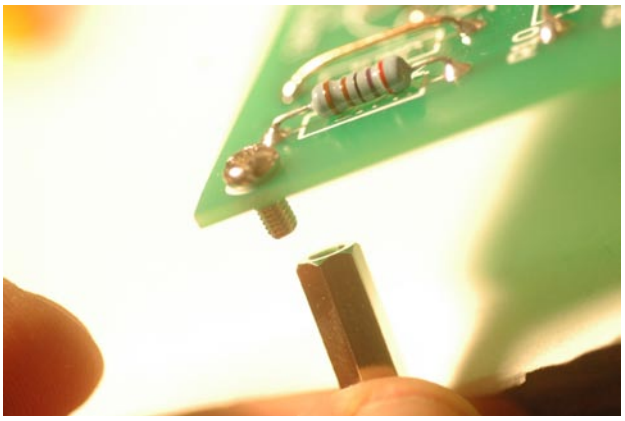


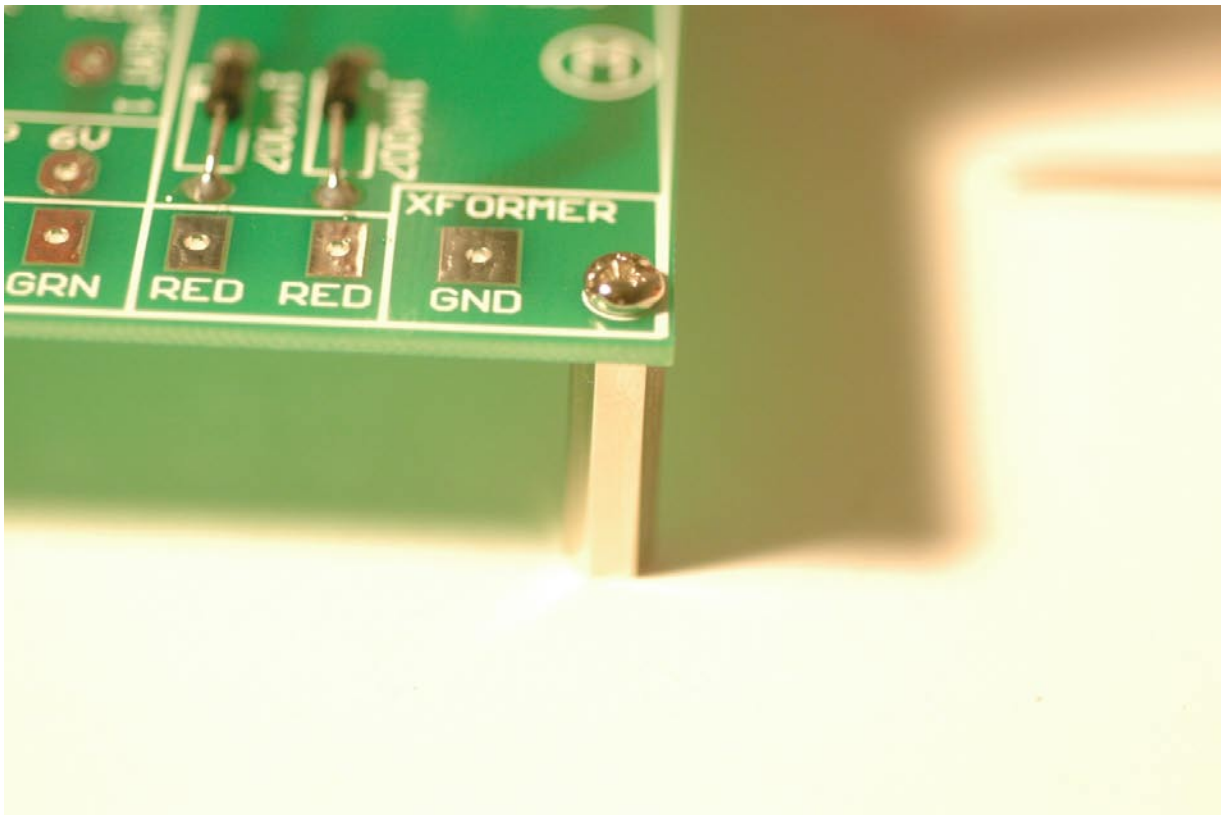
standoffs

there a large 3mm holes in the corners of the board , it is these holes that you place the standoffs to elevate the board above the bottom of the case.

20mm standoffs are ideal ,leaving a nice amount of space under the board for the tube heater wires.



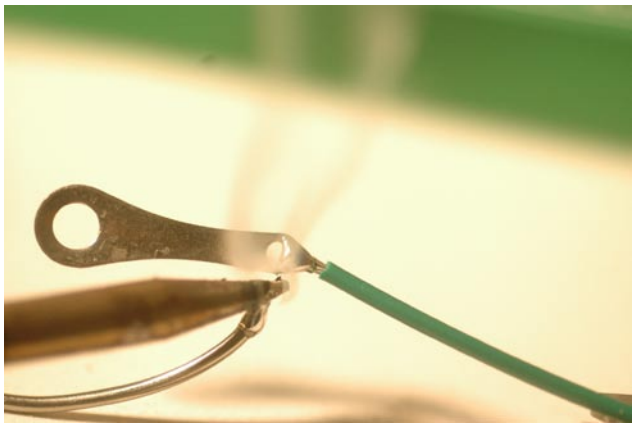


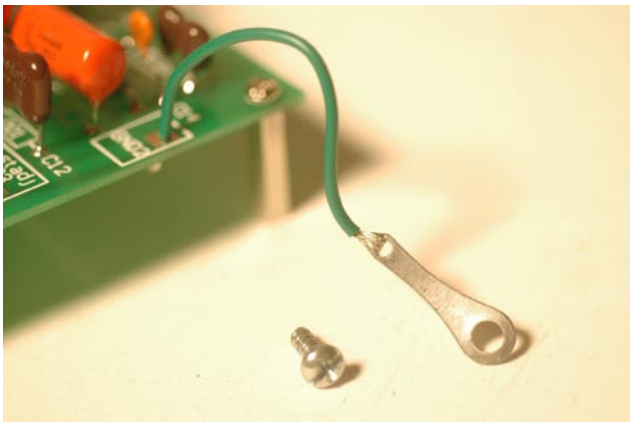


grounds :

there ar two grounds on the board that must grounded to the case. one is the T4 ground , and the other is GND2.

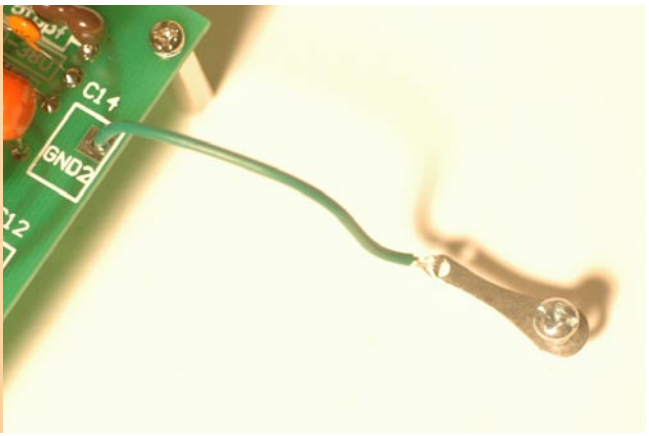
this is best achieved by using a piece of wire and a solder lug and screw.





the wire does not have to be too long , just enough to go from the board to the case.





common problem area :

forgetting to ground these two points will prevent the unit from working.

headers :

the board connections were design for the use of .156 headers. it is possible to simply solder the connections directly to the pads , this method works very well.

there are several areas that you can use the header if you choose.

if you are to use headers , you will need the following

2/ 6 position .156 headers (round)(for st. adj/lim response and audio xformer hook up)

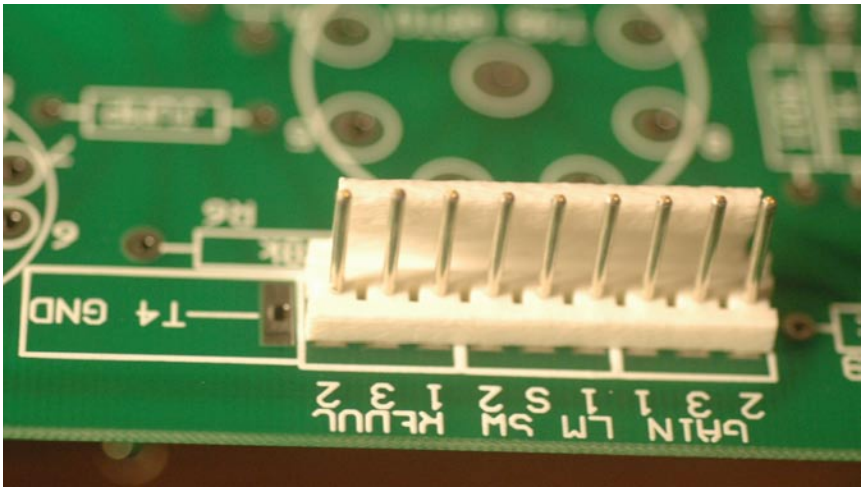
2/ 4 position .156 headers (round)(for meter switch top and bottom)

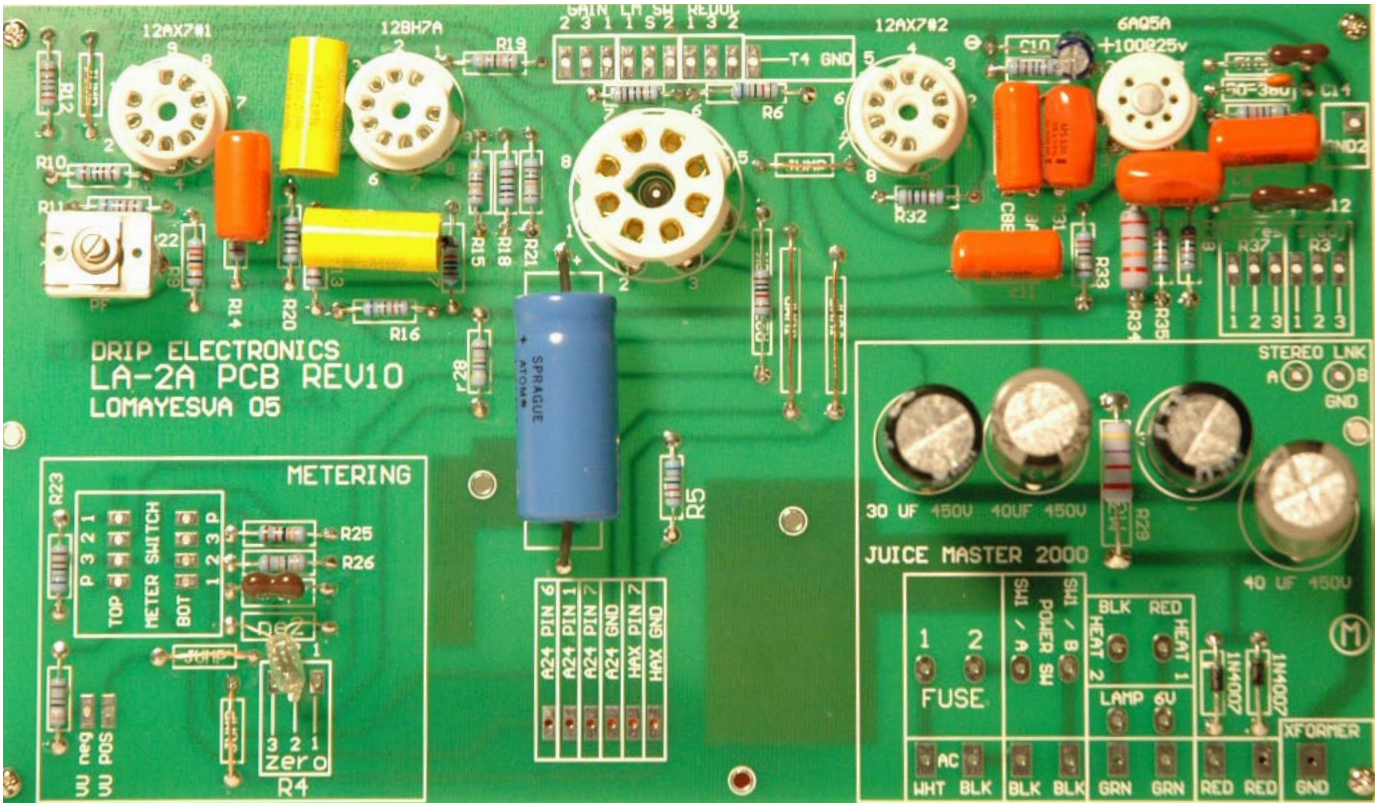
1/ 2 position .156 header (round)

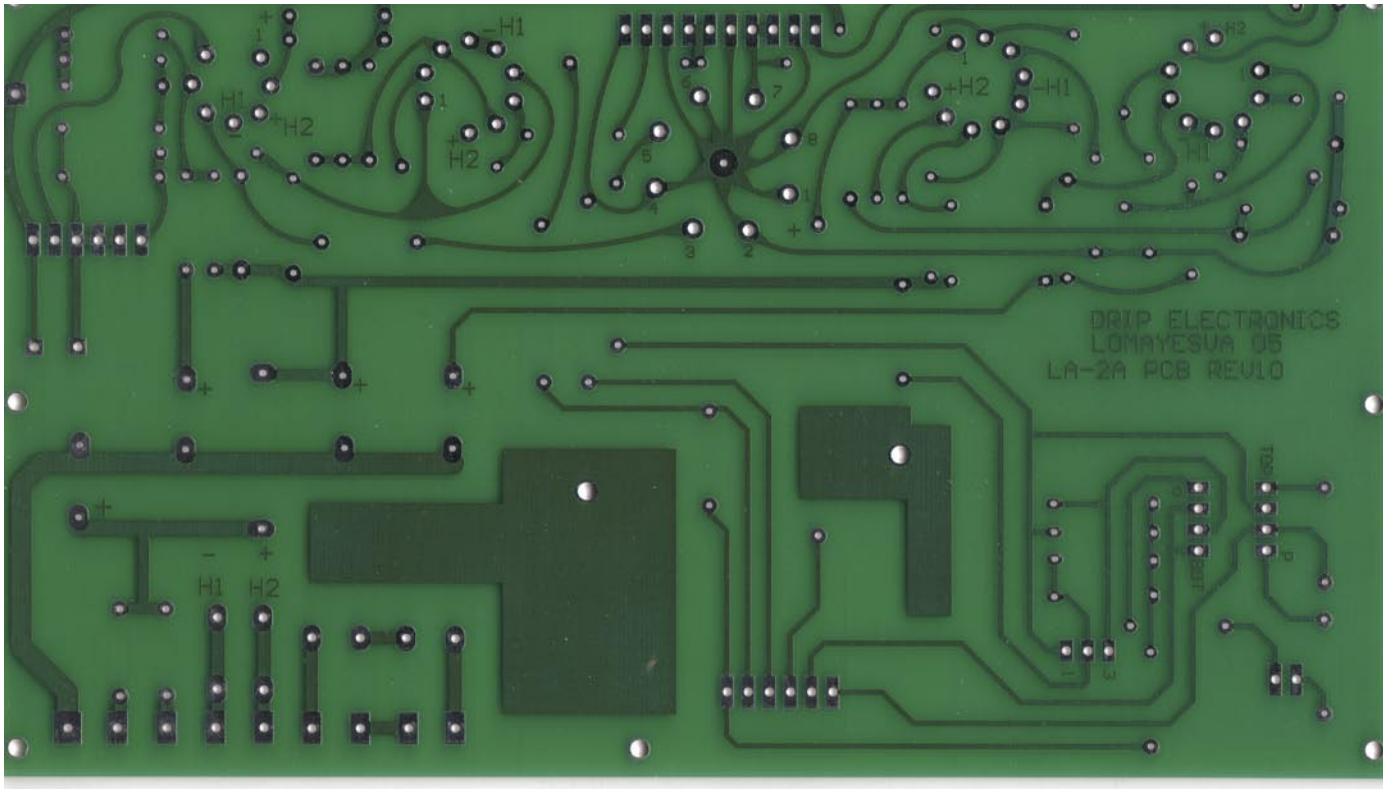
1/ 9 position .156 header or 3 / 3 position .156 headers for gain/reduction/limit switch.

To install , place header into footprint on the pcb and solder from the back of the board.

here you can use just enough solder to form a nice contact with the pad.




DRIP ELECTRONICS
LA-2A PCB REV10
LOMAYESVA 05
12AX7#1
12BH7A
12AX7#2
6AQ5A
METERING
METER SWITCH
JUICE MASTER 2000
STEREO LNK
FUSE
POWER SW
LAMP 6V
XFORMER
30 UF 450V
40UF 450V
40 UF 450V
A24 PIN 6
A24 PIN 1
A24 PIN 7
A24 GND
HAX PIN 7
HAX GND
Zero
R4
R5
R6
T4 GND
WHT BLK
BLK BLK
GRN GRN
RED RED
GND


DRIP ELECTRONICS
LOMAYESVA 05
LA-2A PCB REV10