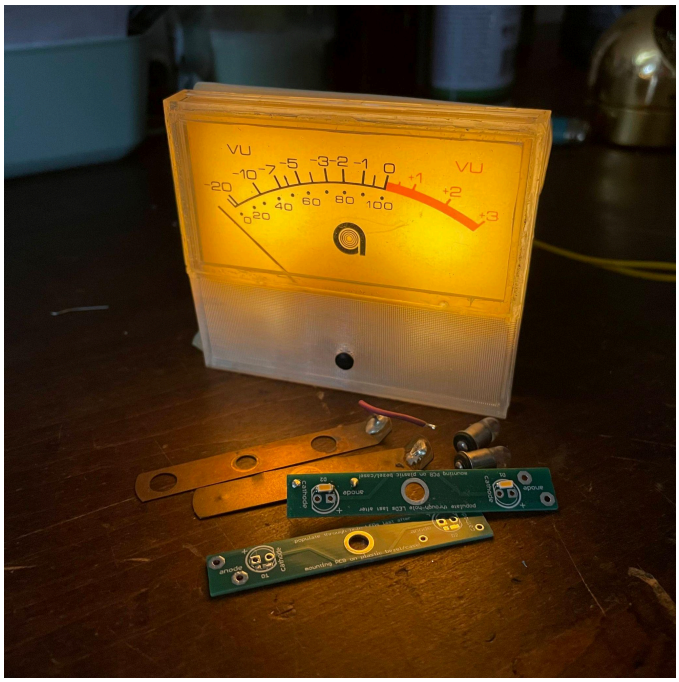
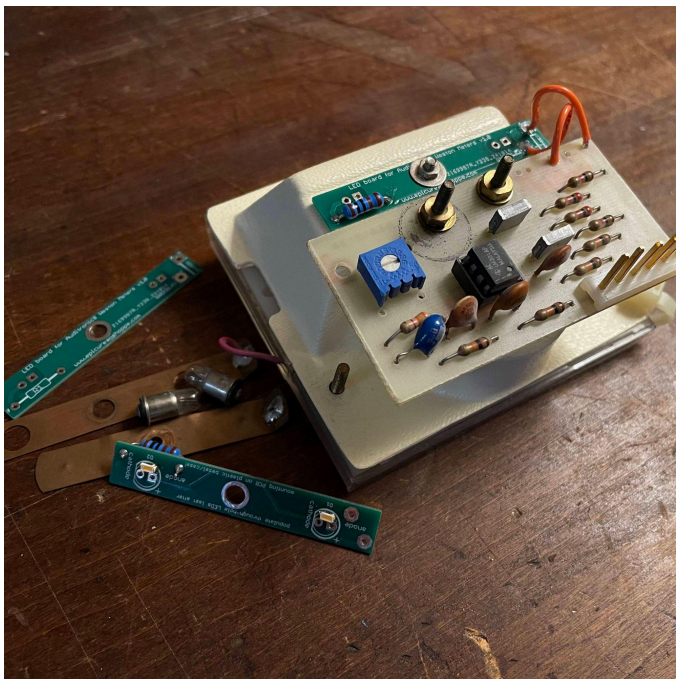
Auditronics Meter 2-Bulb Retrofit PCB

Auditronics Meter 2-Bulb Retrofit PCB (for Weston Meters)

by THE EPICUREAN SHOPPE - www.epicureanshoppe.com

(last updated February 2026)





RELEVANT UPDATE: *on my console, several of the plastic mounting bezels for this style meter were either broken or missing, so I designed and 3D printed my own. They are a perfect drop-in replacement. Here is a Drive link to the .stl file so you can print your own.*

INTRO:

Are your Auditronics meter bulbs burnt out? Or burn out frequently because of the unregulated 24V rail powering them?

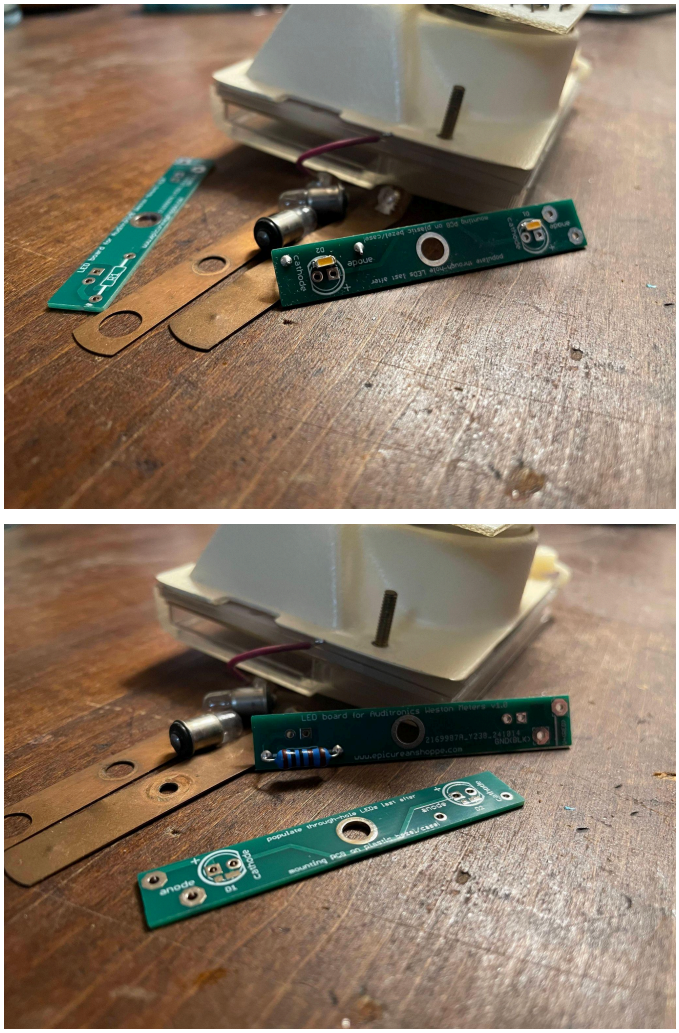
I designed these drop-in replacement PCBs allowing you to use LEDs instead of the 327 incandescent bulbs. I realized that it was surprisingly cheaper and required the same amount of effort to assemble a simple PCB and drop it in place of the original bus bar rather than take apart the existing one and replace the bulbs which will inevitably blow up.

This board specifically fits the “two bulb” Weston 7532 meters with the white plastic housing (*“MA-2” buffer board, and probably works with the later 700 series boards too*), but I’ve also made **a separate version for the older Modutec & Beede meters with three bulbs** and docs for that can be found here.

This PCB drops in to replace the existing metal bus bar where the incandescent bulbs are mounted. The assembly is not as simple as the 3-bulb Modutec/Beede bulb boards and does require some simple soldering and desoldering of wires, but it is still very straightforward.

Other console manufacturers of the era such used the same or similar Weston meters with different setups for backlighting them, so this board could likely be adapted for those as well. For example, Sphere used the exact same backlight setup as Auditronics on their Weston meters.

You can reuse the two wires coming from the voltage connections on the meter buffer PCB. On some variations of the buffer PCB, there is an old carbon comp resistor coming off the bulb “common” lead going to the 0V line – that can stay on and doesn't necessarily need to be removed, but the “red wire” going to the bulbs might need to be moved to a different pad. More on that in the Google Doc. Much more on that below.



New PCB compared next to the original bulb bus bar.

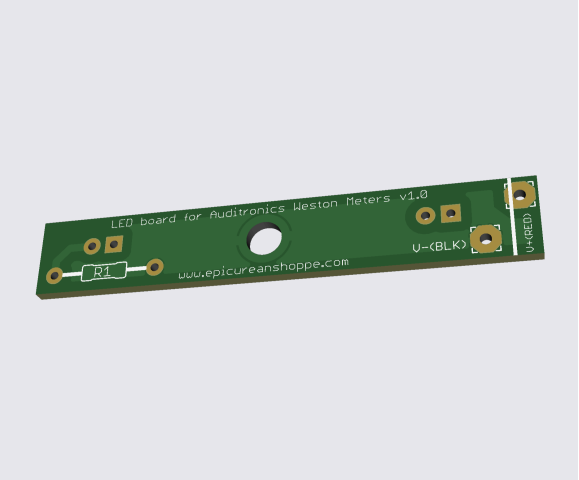
ORDERING THE BOARDS:

These are very simple 2-layer PCBs.

I’ve uploaded them as a project to PCBWay for simple ordering as many as you need:

https://www.pcbway.com/project/shareproject/Auditronics_Meter_2_Bulb_Retrofit_PCB_for_Weston_Meters_b726e898.html

Alternatively, you can download the Gerber files .zip and have your own boards fabricated elsewhere. I use JLCPCB.



PARTS:

I chose to use high-quality “warm white” SMD LEDs because they’re cheap in large quantities, have a better viewing angle than through-hole parts for this application and just look great. However, a footprint is available on the board for standard 5mm through-hole LEDs. You can use any color you want though. Purple, orange, yellow… your pick!

The basic bill-of-materials is really just:

- 2x LEDs of your choosing, either 3216 Metric/1206 footprint SMD or 5mm through-hole
- 1x 2W current limiting resistor. See section below about picking the correct part.

…and that’s it! I can solder one of these boards together in about 3 minutes.

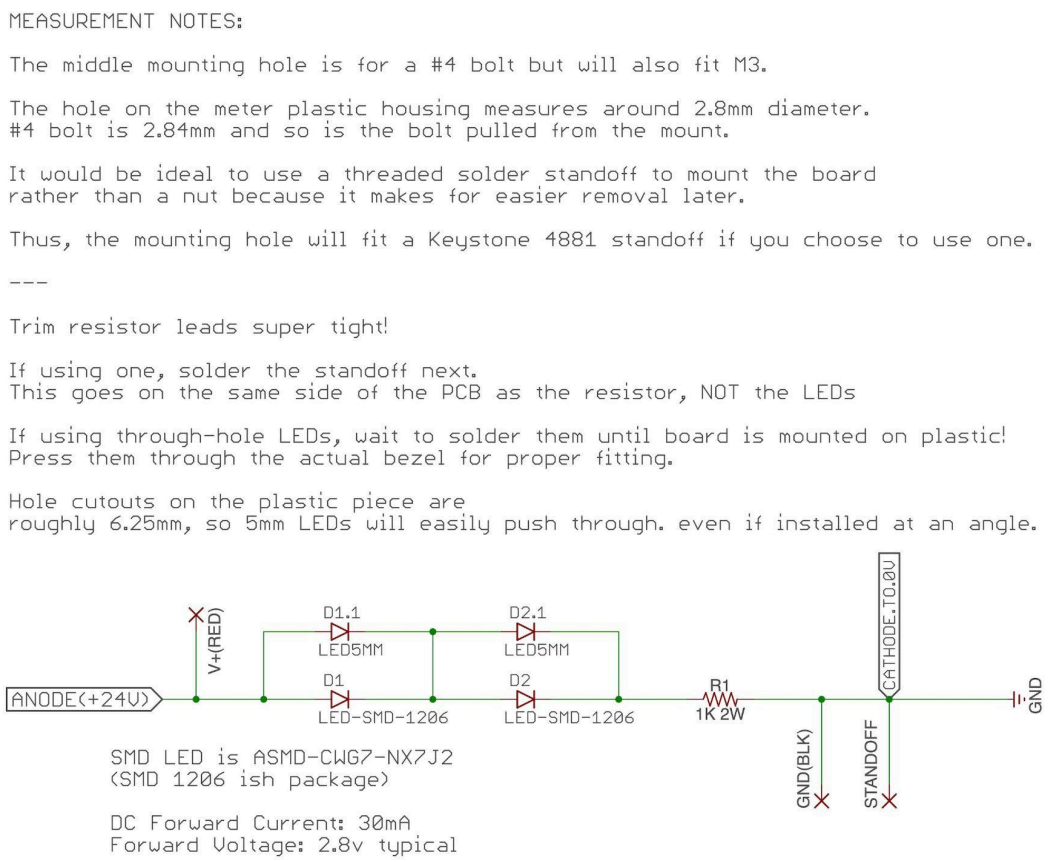
However, there is optional items and their purposes are described below:

- 1x Keystone 4881 Threaded Solder Standoff

Here’s a link to a Digi-Key cart (as of October 2024) with any and all of the parts you might need. Treat this as more of a “template” than an actual shopping cart, as you don’t need two different types of LEDs, might want or need different current limiting resistors, might not want the Keystone standoff, etc etc.

The quantities are just for one meter, so be sure to order enough parts for as many meters you plan to work on. https://www.digikey.com/short/7tv9wphr

Here's the schematic (and some notes) so you know exactly what's going on, and a link to download the schematic in full resolution.



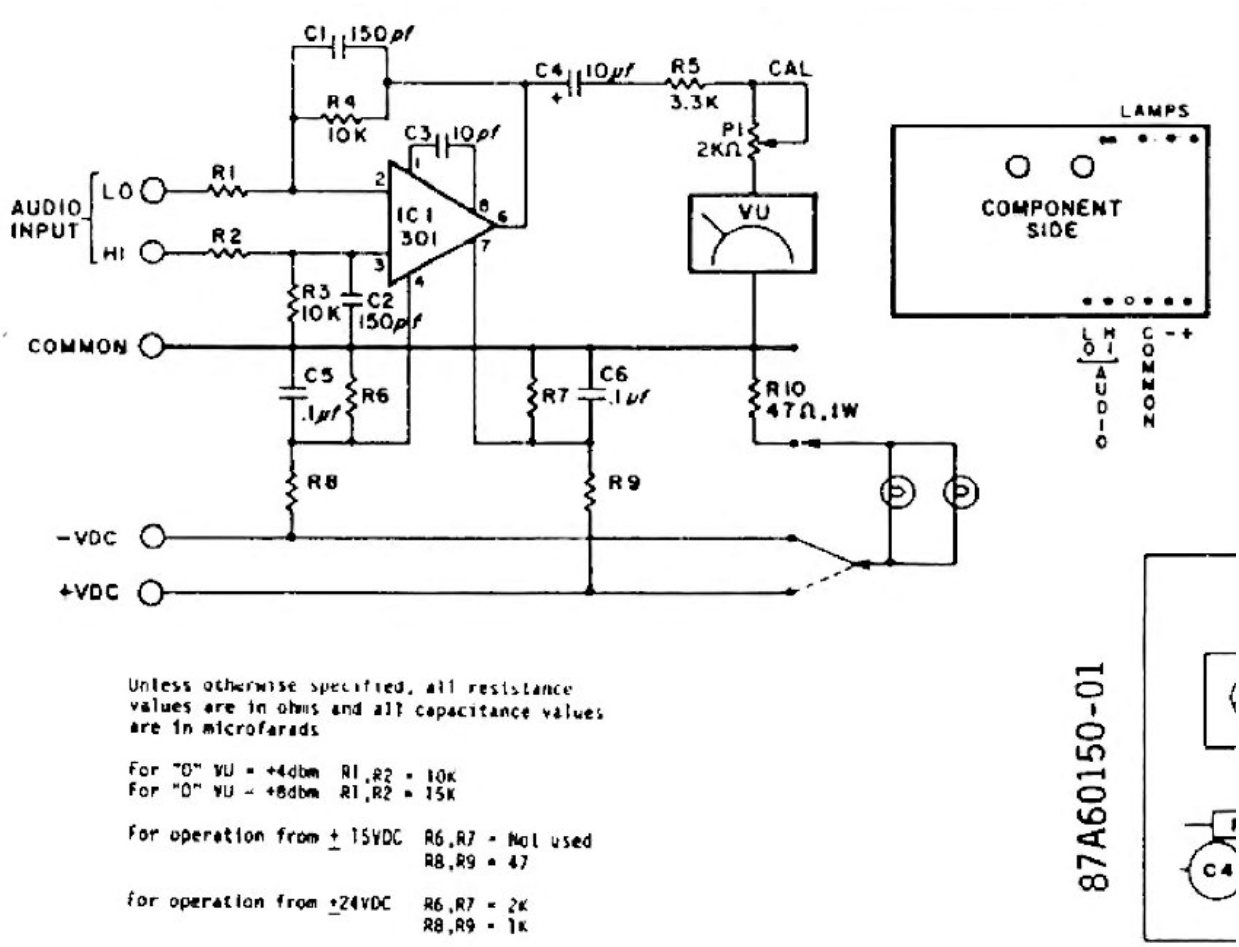
ELECTRICAL CONNECTIONS:

Since the original used incandescent bulbs and polarity didn't matter, it seems sort of random which terminal ring lug Auditronics decided to hook up the positive lead to or the 0V lead to. In fact, Auditronics even notes this on their stock schematics!

Measure the power rails to your existing bulb PCBs with a multimeter before removing them and make a note of the polarity. I've attached a photo below of which rails go where on my meter buffer PCBs, but there were **various revisions of this board** so don't rely on that pinout.

Measuring is also good for determining the exact voltage since the 24V rails are unregulated in stock Auditronics power supplies.

Since the power leads to the existing bulbs could be negative voltage rather than positive, so keep that in mind for your hookups. Other times, the meter buffer PCBs were configured to run off the 15V rail rather than the 24V rail, so that is also something to consider when choosing your current limiting resistor value for the new PCB.

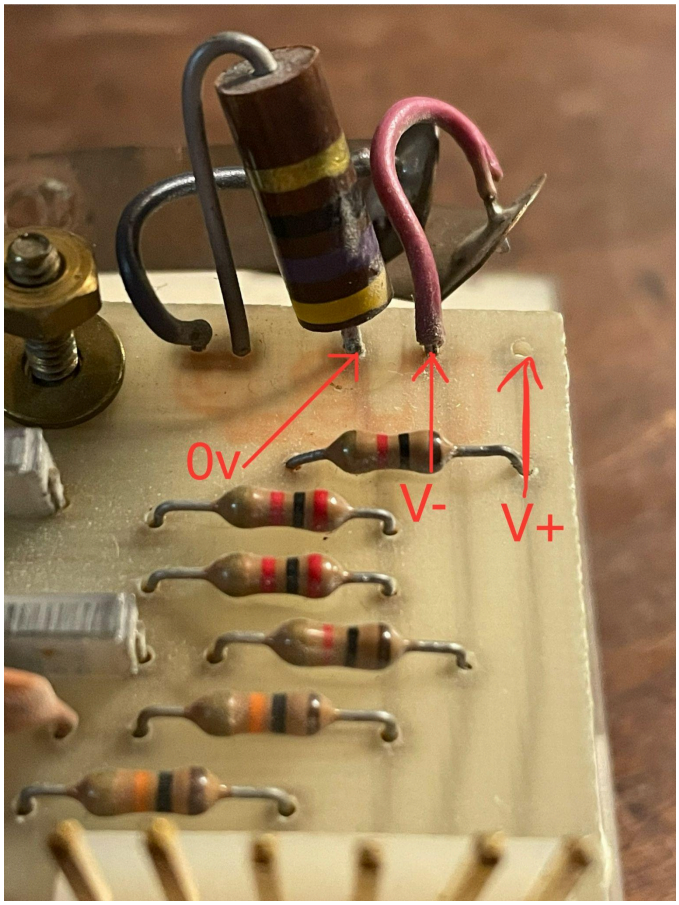


Here is the stock schematic showing how the incandescent bulbs are configured. [Full schematic below](#).

Note from other units: "Meter lamps may be returned to either pos. or neg. 24V bus."

This PCB is **designed to run off of a positive voltage rail** and then return to the 0V pad, so you might need to desolder a wire on the stock meter buffer PCB and move it over depending

on how Auditronics assembled your meters. Scroll down to the "**SUPPLY VOLTAGE LEAD HOOKUPS**" chunk of this doc for more info.



This is the "pinout" going to the bulb bus bar on a late 1970s "MA-2" VU meter buffer board. On this particular board, the "red" wire is coming off of the negative V- rail, so the bulbs are powered by -24V. The black wire returns to a 47 ohm 1W resistor which is then tied to the 0V / "common" trace.

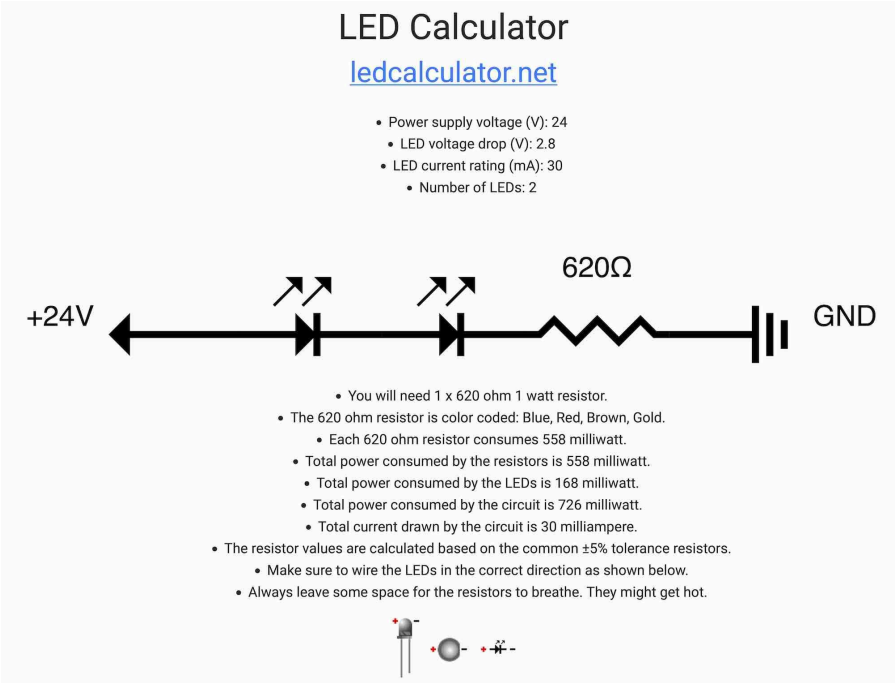
So if you're asking: ***"My buffer board is wired so the supply voltage to the bulbs is -24V rather than +24V – what if I don't want to remove the wires from the PCB?"*** In this case, I'd recommend just installing the LEDs backwards on the PCB. You would remove the two wires from their respective copper bars and then run them to the correct pads on the new LED PCB. Again, see the **"SUPPLY VOLTAGE LEAD HOOKUPS"** section for more options.

CALCULATING OPTIMAL LED BRIGHTNESS:

LEDcalculator.net will be your friend in determining your resistor value(s) based on the LEDs you plan to use. Grab the datasheet for your part and locate the forward voltage and current draw. Even if the website suggests 1/2W or 1W resistors, I recommend using good 2W parts (metal film or metal oxide).

Mentioned above: **your 24VDC rails might not actually be 24V** if you are using the stock unregulated Auditronics power supply, so be sure to measure the actual voltage on the stock bulb board and use that measurement for calculating your current limiting resistor value.

The calculator spits out a resistor value giving you basically the brightest possible safe operating conditions, and sometimes this resistor value isn't standard.



LED EXAMPLES:

Realistically a 1 kOhm resistor is usually a safe bet for most configurations. I suggest the TE Connectivity RR02 series. They are 2W flameproof and have a small footprint.

For example, the calculator tells me to use a 620R 1/2W resistor. I don't stock those, so I am just going to use a 1 kOhm 2W resistor which will be plenty bright.

The warm white SMD LED I am using is **ASMD-CWG7-NX7J2** by Broadcom with a 3216 Metric/1206 footprint. Other SMD LEDs will probably fit, though. According to the datasheet, this part has a current draw of 30 mA and a typical forward voltage of 2.8 volts.

Again, it is important to use a high watt resistor for the current limiting LED!

POPULATING THE PCB:

1. A) If using 5mm through hole LEDs, skip to “Step 2” as you will be installing those last (see Step 7).

   B) If using SMD LEDs, populate these first. Pay attention to polarity and use a product like Chip Quik and tweezers to help keep your parts in place. SMD parts are small but you’ll get the hang of it. Once all set, be sure to clean off any flux 99% isopropyl alcohol.

2. Populate resistor - elevate it slightly off the PCB for heat dissipation.

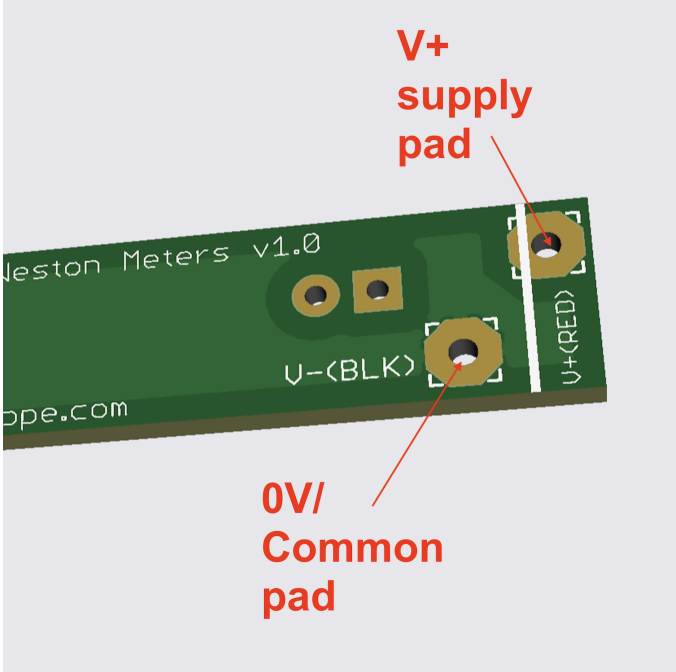
3. ***OPTIONAL*:** the large hole in the center of the LED PCB can be populated with a soldered in threaded standoff: Keystone 4881. Using this threaded standoff makes installation and removal of the LED PCB more straightforward because won’t need nuts and washers, just a #4 bolt.

   If you want to use the Keystone 4881 standoff, install it on the back side of the PCB (the resistor side) and not the bulb side.

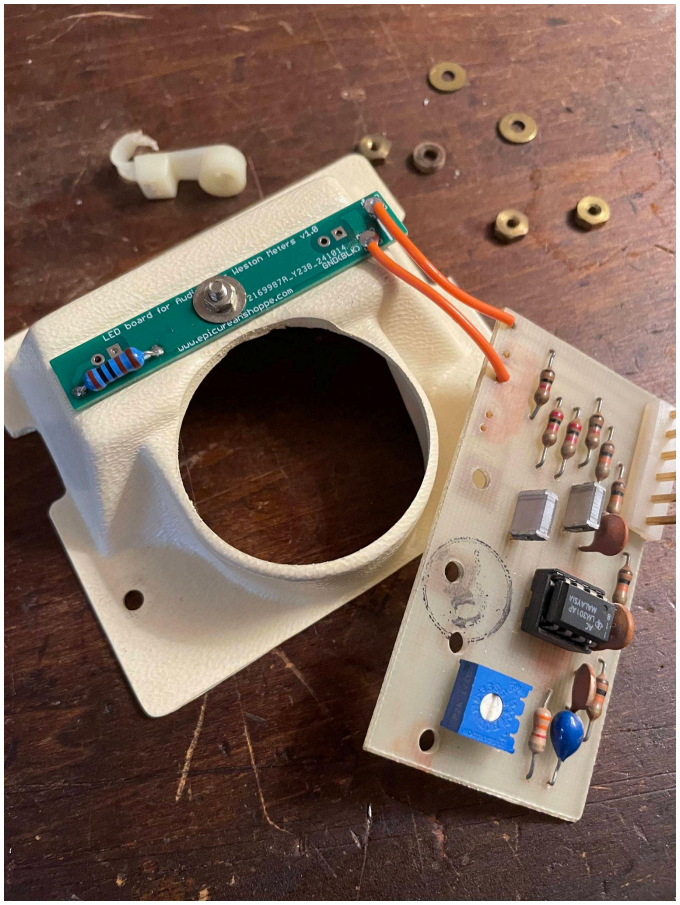
4. Make sure you use low profile cutters to gently trim the component legs on the bulb side of the PCB so it will mount flush against the plastic meter bezel/housing.

5. Solder the supply voltage and 0V/common wire leads coming from the pads on the VU buffer board to the respective pads on the new PCB.

   Since supply wiring configuration could be different unit to unit, see the "SUPPLY VOLTAGE LEAD HOOKUPS" section below for detailed instructions and potential options.

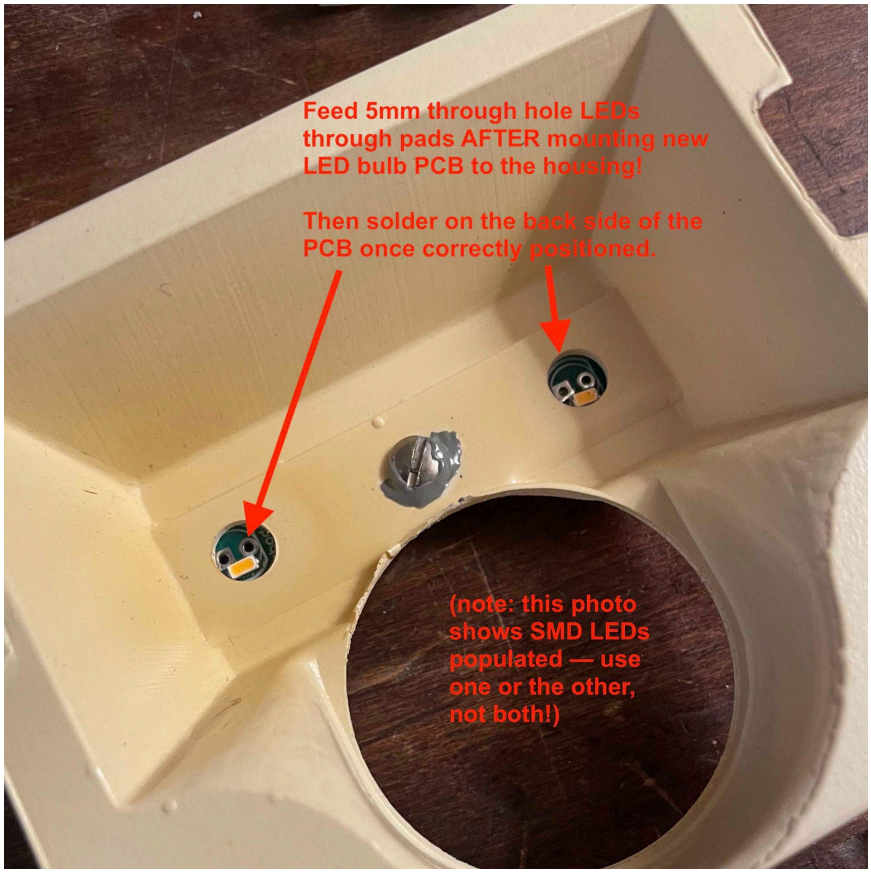


6. Mount fully assembled LED PCB to plastic housing. Everything should now look like this:



7. ***HOWEVER***: if **using 5mm through hole LEDs**, mount the fully assembled LED PCB to the plastic housing before populating the LEDs.

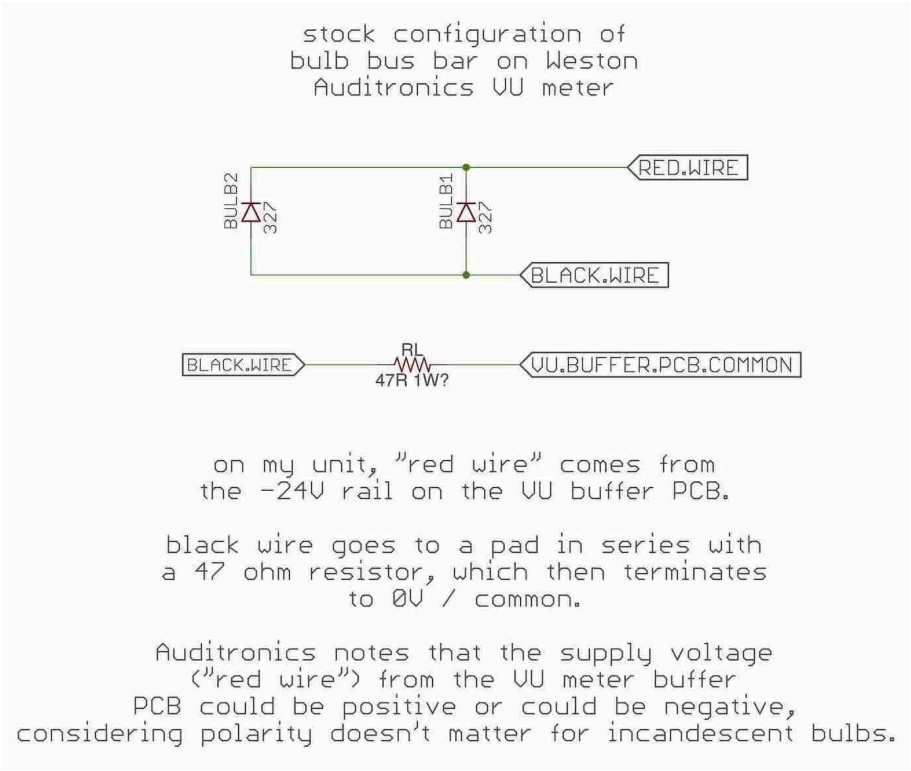
   Once you have the LED PCB cleanly mounted onto the plastic housing, feed the LEDs through the bulb side. This way you can be sure your LEDs are correctly positioned into the two holes on the plastic housing.



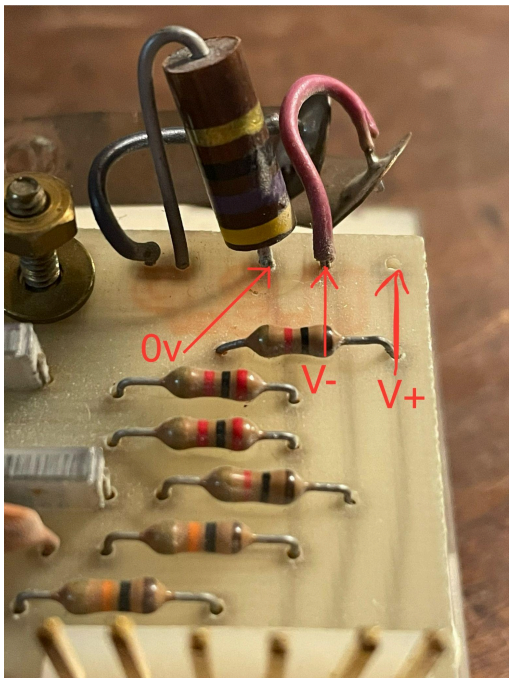
SUPPLY VOLTAGE LEAD HOOKUPS:

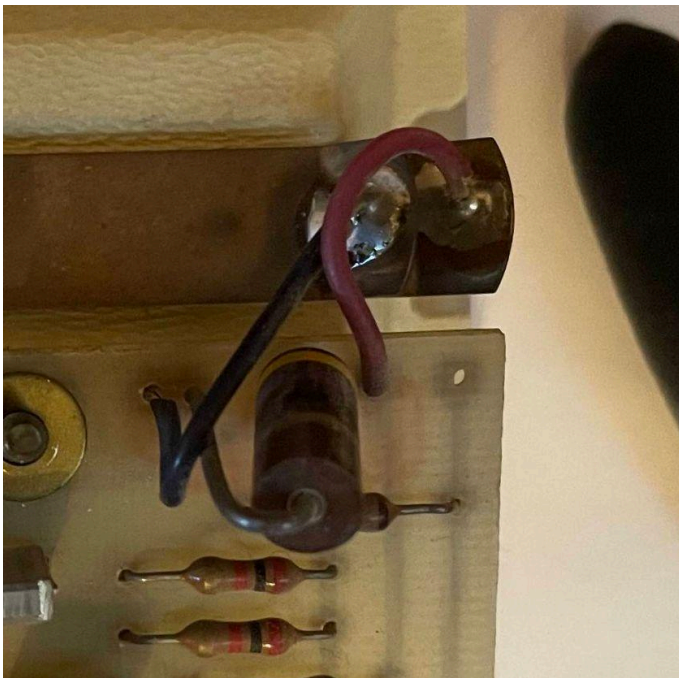
I covered some of this under the "ELECTRICAL CONNECTIONS" section of the document, but here's some more in the context of assembling the PCB.

This PCB is **designed to run off of a positive voltage rail** and then return to the 0V pad, so you might need to desolder a wire on the stock meter buffer PCB and move it over depending on how Auditronics assembled your meters.



This is the scheme on my own late 1970s meter – the bulbs were powered by the -24V rail.





…and the "pinout" labeled on the 1970s VU meter buffer board.
As mentioned, on my board the "red" wire goes from the -24V rail to the bulb bus bar.

So there are a few different ways to approach running supply voltage to your new LED PCB, all of which require some form of soldering.

1. Assemble LED PCB with LED polarity normal according to schematic.

2. If it isn't already connected to the "V+" pad on the VU buffer PCB, desolder the "red" (supply) wire lead (and from the respective copper bus bar if you'd like to re-use it).

   If the "red" wire lead is connected to the correct "V+" pad on the VU buffer PCB, leave it there and just desolder it from the copper bus bar so you can re-use it.

3. Desolder the "black" (common) wire lead from its respective copper bus bar, but it can optionally stay where it is on the PCB depending on some things (see step 4)

4. *Optionally:* remove the old carbon comp current limiting resistor and "black" (0V/common) wire lead from VU buffer PCB.

   If the resistor measures fine and doesn't look like it was ever burning up, it doesn't really hurt to keep it in place – it will just be an additional 47 ohms in series with the new current limiting resistor on the LED PCB. This will not make a significant difference in brightness.

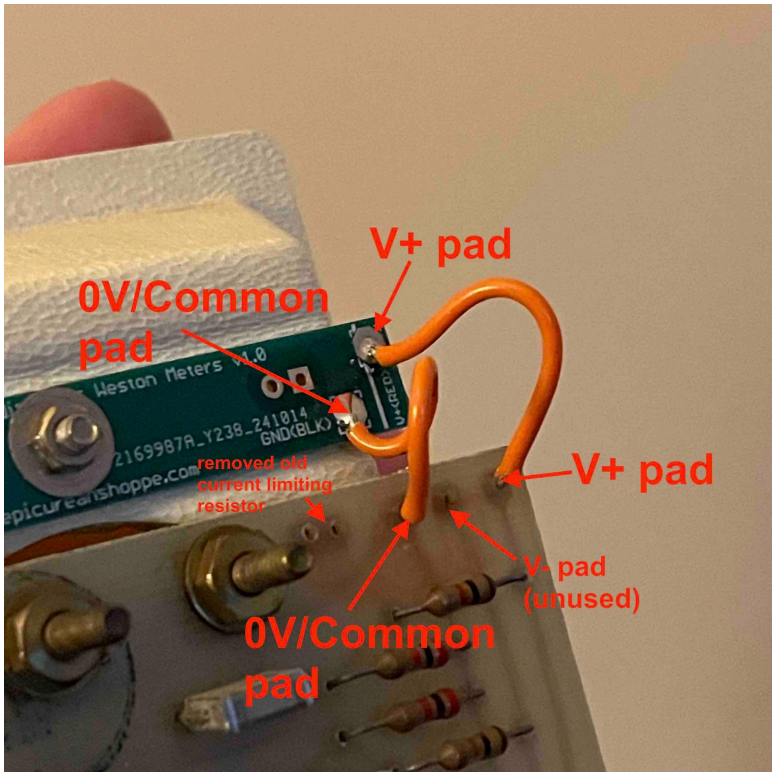
   HOWEVER: I chose to remove mine because it's one less 50~ year old component to go bad.

5. Solder a new (or re-used) "red" wire lead to the **"V+" supply pad** on the VU buffer PCB.

6. If you removed the "black" wire lead and the old carbon comp current limiting resistor from the VU buffer PCB, solder a new (or re-used) "black" wire lead to the **"0V" / common pad** on the VU buffer PCB.

7. Solder the correct "V+" and "0V"/Common wire leads from the VU buffer PCB to their respective pads on the new LED PCB.

   Photo is of one of mine wired up – I used all new wires and removed the old resistor from the VU buffer PCB:

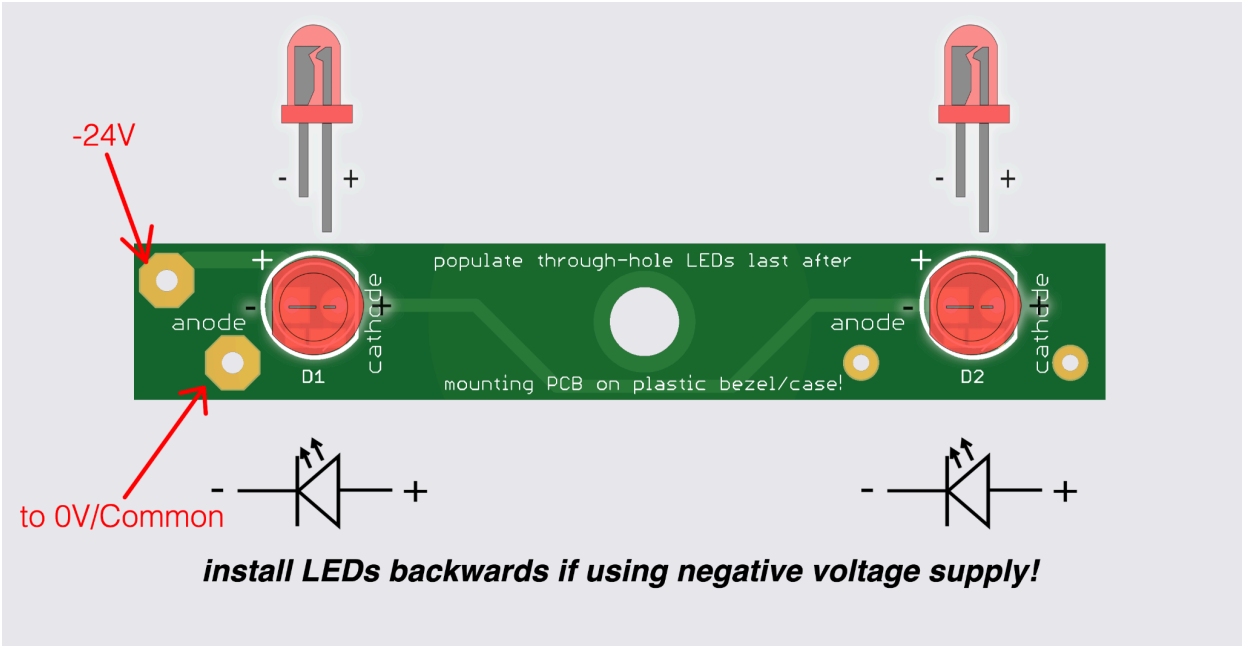


8. Once complete and all wired up, you can reassemble the meter.

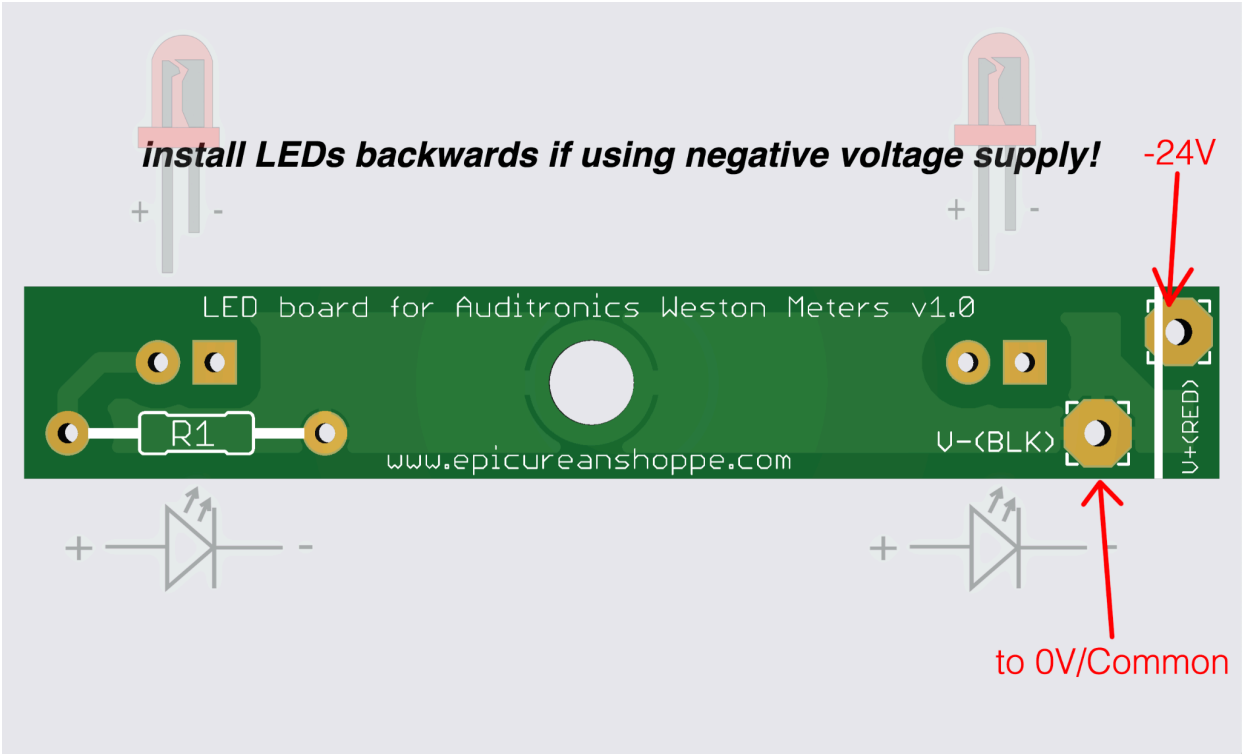
So if you're asking: ***"My buffer board is wired so the supply voltage to the bulbs is -24V rather than +24V, what if I don't want to remove the wires from the PCB?"***

In this case, I'd recommend just installing the LEDs backwards on the PCB so they can be powered from the negative supply. Otherwise, follow the exact "**SUPPLY VOLTAGE LEAD HOOKUPS**" steps.

Here is the wiring diagram for installing LEDs backwards to use the negative supply. This is easier done with through hole LEDs than it is with SMD:



install LEDs backwards if using negative voltage supply!



GUIDE TO MECHANICALS, DISASSEMBLY, & REASSEMBLY:

The Modutec/Beede meter 3-bulb board is significantly easy to install. This one requires some soldering and some more work.

The info below makes use of a late 1970s meter as an example. There are a few different variations of this, so not all configurations will be the same, but hopefully this will be rough enough to help you get around.

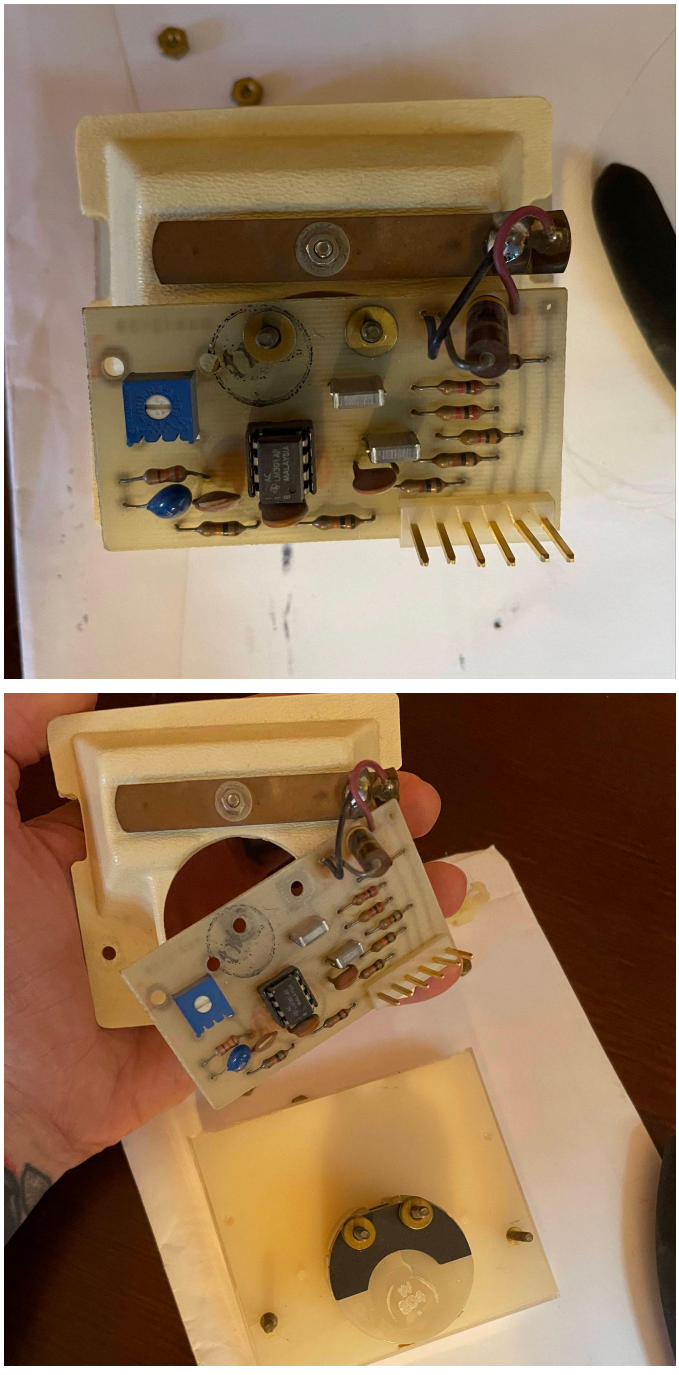
This project is definitely easiest to complete if you remove your entire meter from the meter bridge. I think it could be done while leaving your meter bridge as-is, maybe – just might be a sort of clunky approach because you still have to remove the plastic rear housing either way to access the bulbs.

Tight squeeze… good luck.

Step 1:

Remove the two nuts holding the meter buffer PCB onto the meter. There are two additional nuts located below the meter buffer PCB also holding the plastic housing to the meter itself. **Keep track of the flat washers, the nuts, and any other mounting hardware and where it**

was located! Once you do this, you can remove the entire plastic housing assembly from the meter itself.

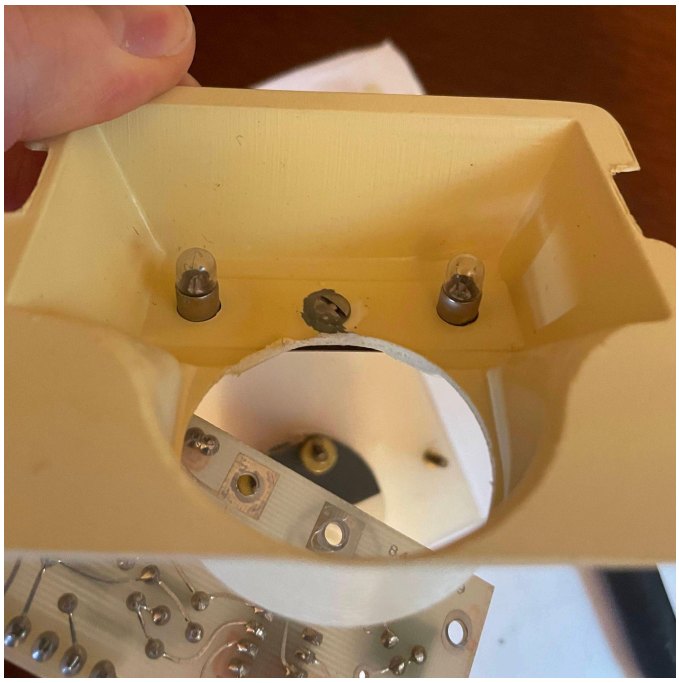


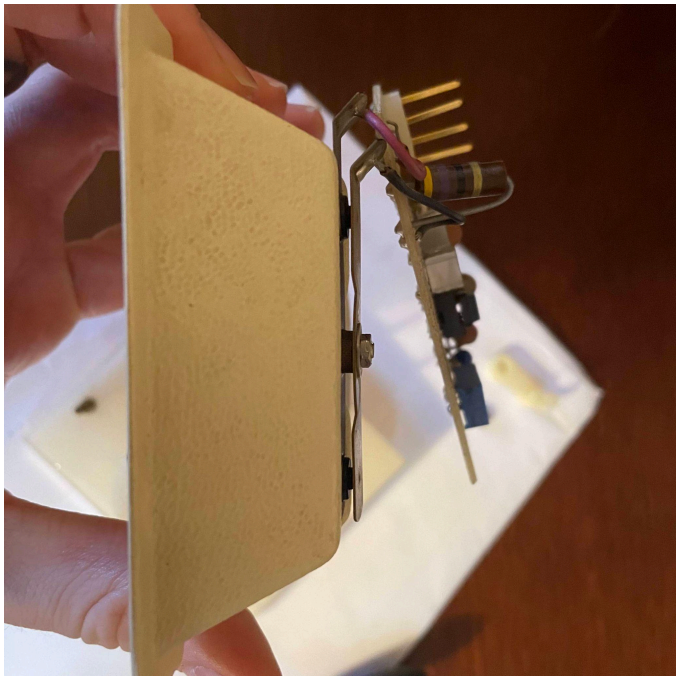
Housing and meter buffer removed from meter.

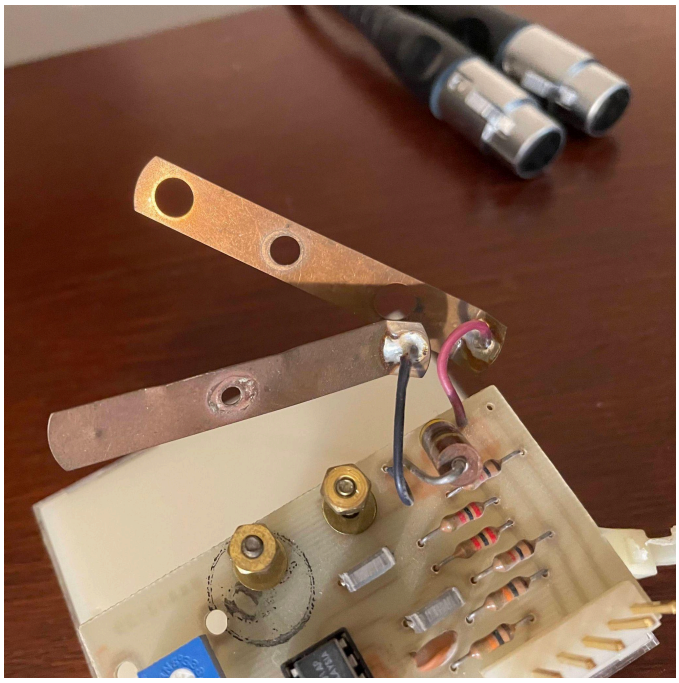
Step 2:

Take some notes on how exactly the bus bar holding the bulbs is mechanically connected to the plastic housing.

To disassemble the bus bar, remove the #4 bolt on the inside of the housing which might be held in place with some glue. **Be very delicate with this** considering you don't want to damage the potentially brittle plastic of the housing. Mine was a huge pain to remove. On the rest of my meters, I plan to throw it in the trash and replace it with a new bolt + new washers + nut.







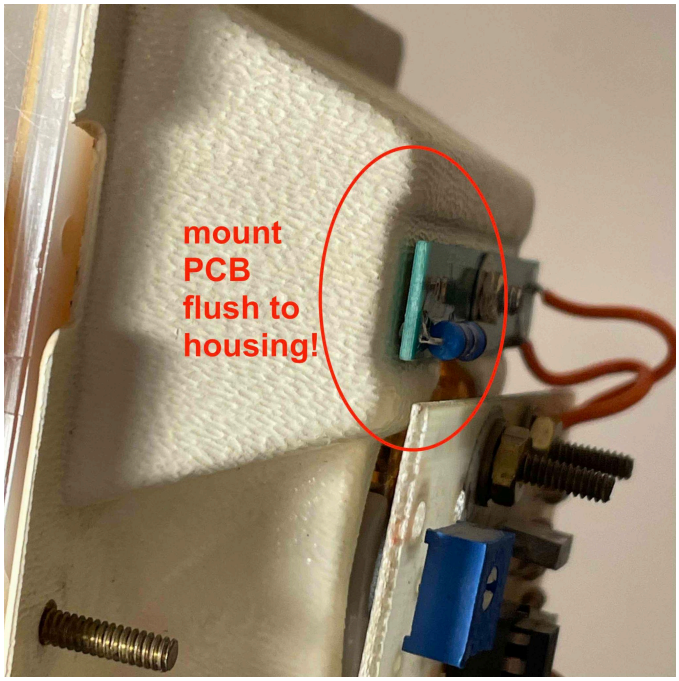
Photos showing how the bus bar was installed on my meters.

Step 3:

Assemble your new LED PCB per instructions above.

Step 4:

Assemble LED PCB onto plastic housing.



As mentioned earlier, I've seen some of these meters with the old bulb bus bar mounted flush to the plastic housing of the meter and others with an additional non-conductive washer serving as standoff pushing it back a bit. I suggest disregarding any standoff parts and mounting the new board flush to the meter housing, since LEDs are not as physically long as the original incandescent bulbs.

If you used a Keystone 4881 threaded standoff on your PCB, installation is as simple as screwing a #4 bolt through the bulb side of the plastic housing into the threaded standoff.

Otherwise, your mounting situation should look like this:

nut

=

(optional) lock / tooth washer

=

flat washer

=

PCB

(with bulbs facing out, of course)

=

plastic housing

=

(optional) flat washer

=

#4 bolt

There is no lock/tooth washer with the configuration – just the #4 bolt, a flat washer, and the nut. I'd suggest adding one for better mechanical connection between the flat washer and the nut. The stock #4 bolt has a pretty wide head – if you plan to replace the bolt, you might need to add another flat washer between the bolt head and the plastic housing.

Step 5:

Reassemble the rest of the meter in the same order you took it apart!

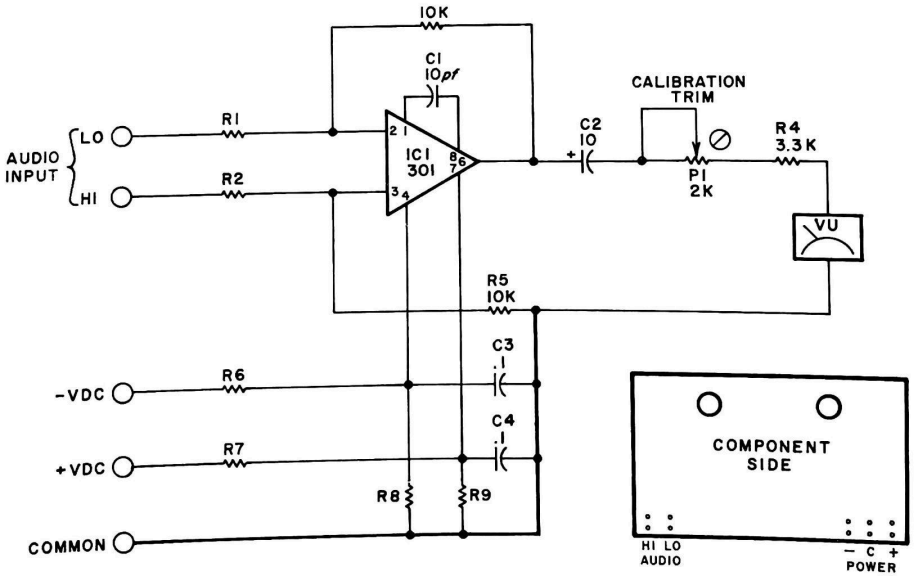
ADDENDUM – SCHEMATICS FOR REFERENCE:

Here are various schematics for Auditronics VU buffer boards.

1st version with LM301:

The first schematic shows the original VU buffer scheme from the original Grandson 110 console.

The later 1970s version I have pretty much matches this schematic in terms of circuit, but is a different board layout with a 6-position pin header on the bottom (one of the 6 pins is not connected).



UNLESS OTHERWISE SPECIFIED, ALL RESISTANCE VALUES ARE IN OHMS AND ALL CAPACITANCE VALUES ARE IN MICROFARADS.

FOR "0" VU = +4dBm R1, R2 = 10K
FOR "0" VU = +8dBm R1, R2 = 15K

FOR OPERATION FROM ± 15 VDC R6, R7 = 47
R8, R9 = NOT USED

FOR OPERATION FROM ± 24VDC R6, R7 = 1K
R8, R7 = 2K

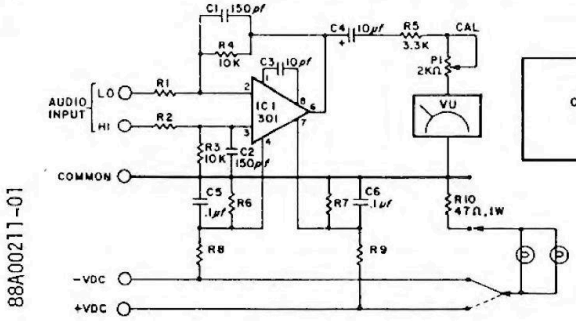
SCHEMATIC DIAGRAM
w/Parts List
VU Meter Bucc

Later 1970s version “MA-2”:

This second attached page is the MA-2 VU meter board, which is exactly what my late 1970s console has.

PARTS LIST — MODEL MA-2 VU METER AMPLIFIER

Ref. No.	Description	Part No.
-	P.C. Board	12139
P1	Potentiometer; 2K ohm trim	08022
IC1	Integrated Circuit; 301 Select	16033
R1	Resistor; Carbon, 10K ohm, ½ W., 5%	E103
R2	Resistor; Carbon, 10K ohm, ½ W., 5%	E103
R3	Resistor; Carbon, 10K ohm, ½ W., 5%	E103
R4	Resistor; Carbon, 10K ohm, ½ W., 5%	E103
R5	Resistor; Carbon, 3.3K ohm, ½ W., 5%	E332
R6	Resistor; Carbon, 2K ohm, ½ W., 5%	E202
R7	Resistor; Carbon, 2K ohm, ½ W., 5%	E202
R8	Resistor; Carbon, 1K ohm, ½ W., 5%	E102
R9	Resistor; Carbon, 1K ohm, ½ W., 5%	E102
R10	Resistor; Carbon, 47 ohm, 1 W., 5%	J470
C1	Capacitor; Disc Ceramic, 150 pf	00052
C2	Capacitor; Disc Ceramic, 150 pf	00052
C3	Capacitor; Disc Ceramic, 10 pf	00028
C4	Capacitor; Dip Tantalum, 10 uf	00005
C5	Capacitor; Polyester, .1 uf	00068
C6	Capacitor; Polyester, .1 uf	00068



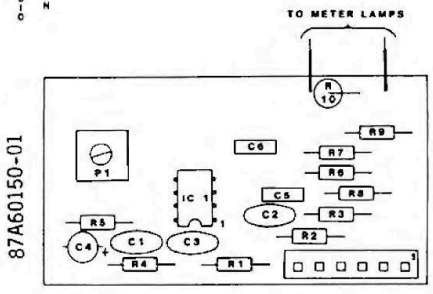
Unless otherwise specified, all resistance values are in ohms and all capacitance values are in microfarads

For "0" VU = +4dbm R1,R2 = 10K
For "0" VU = +8dbm R1,R2 = 15K

For operation from ± 15VDC R6,R7 = Not used
R8,R9 = 47

For operation from +24VDC R6,R7 = 2K
R8,R9 = 1K

SCHEMATIC DIAGRAM
Model MA-2
Meter Amplifier



COMPONENT LOCATION
Model MA-2
Meter Amplifier

700 Series and related:

This docs below are various VU meter schemes from the rare 700 series consoles – images were sourced from the Auditronics Owners Facebook group. Slightly later than the “MA-2”, but similar schemes.

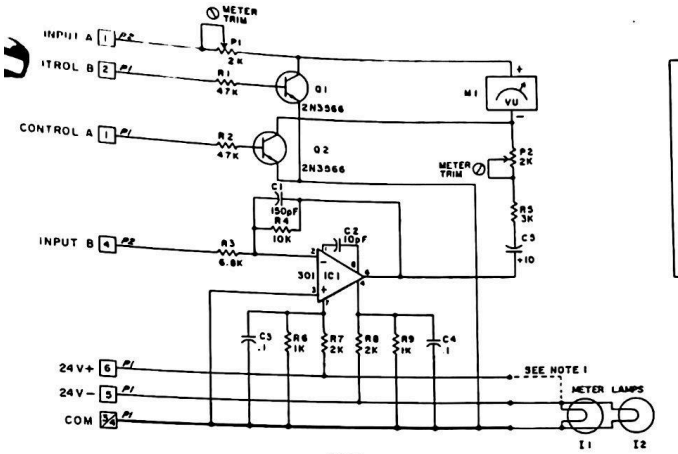
This clearly uses the same Weston 7532 meter and presumably same plastic housing / bezel, but a different buffer / driver circuit.

WESTON MODEL 1502

88A00260-01

87A60184

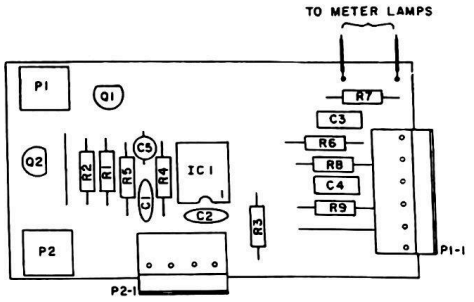
5-99



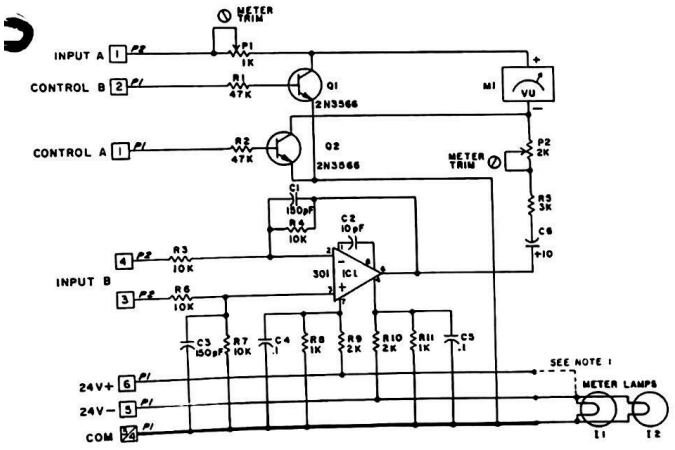
NOTE
UNLESS OTHERWISE SPECIFIED:
RESISTANCE VALUES ARE IN OHMS.
CAPACITANCE VALUES ARE IN MICROFARADS.

NOTES:
1. METER LAMPS MAY BE RETURNED TO EITHER POS. OR NEG. 24V. BUS

SCHEMATIC DIAGRAM
SMA-1
SWITCHING METER AMP-1



COMPONENT LOCATION
MODEL 700-SMA-1
SWITCHING METER AMP

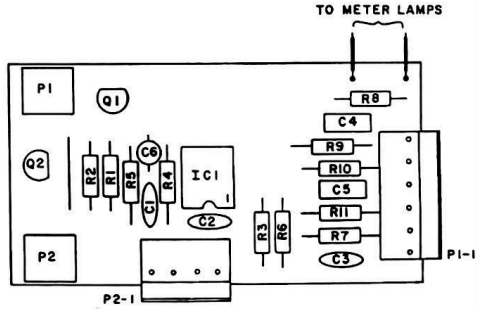


NOTE
UNLESS OTHERWISE SPECIFIED:
RESISTANCE VALUES ARE IN OHMS.
CAPACITANCE VALUES ARE IN MICROFARADS.

NOTES:
1. METER LAMPS MAY BE RETURNED TO EITHER POS. OR NEG. 24 V. BUS

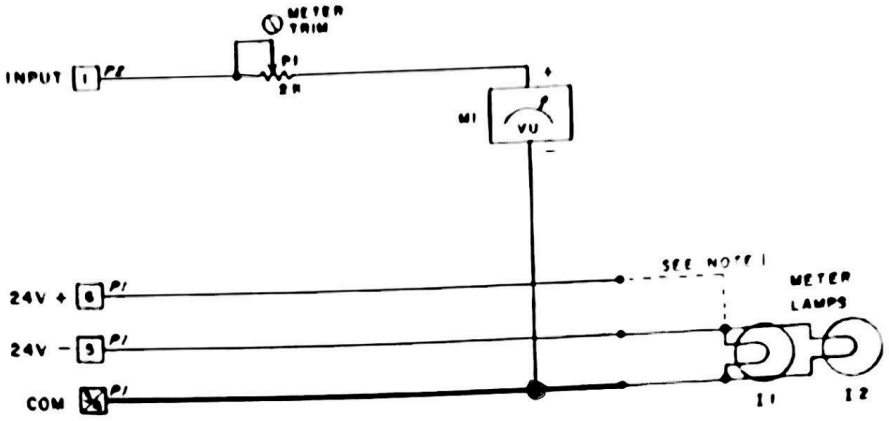
SCHEMATIC DIAGRAM
SMA-2
SWITCHING METER AMP-2

88A00261-01



87A60185

COMPONENT LOCATION
MODEL 700-SMA-2
SWITCHING METER AMP



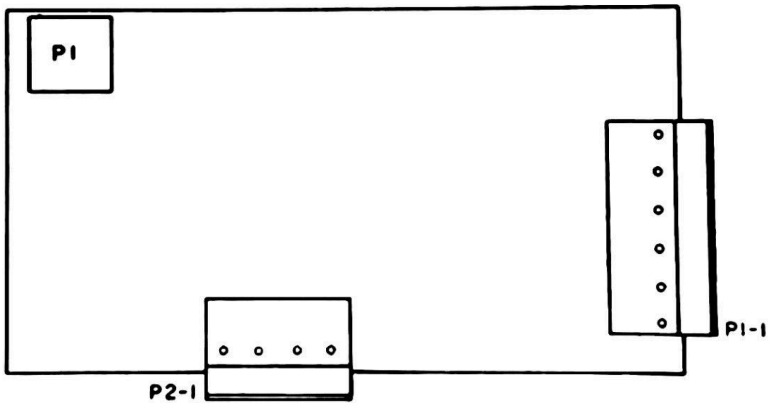
NOTE

UNLESS OTHERWISE SPECIFIED:
RESISTANCE VALUES ARE IN OHMS.
CAPACITANCE VALUES ARE IN MICROFARADS.

NOTES
1 METER LAMPS MAY BE RETURNED TO EITHER POS OR NEG 24V BUS

SCHEMATIC DIAGRAM
MTB-1
METER TRIM BOARD-1

88A00259-01



COMPONENT LOCATION
MODEL 700-MTB-1
METER TRIM BOARD

87A60183