Vacuum Tube Audio MK3
upgrade kit manual *v22 octal*

Thanks for your purchase of our VTA MK3 driver upgrade PCB.
It has been designed to replace the original Dynaco input driver with an improved all triode design that will drop into the original position without any electrical or mechanical changes to your amplifier. Once installed your MK3 will be greatly improved both audibly and with traditional THD & IMD measurements.

I. Introduction
You will need a good soldering iron and a multimeter in order to properly install this modification.
In the following sections we will guide you through the project.
Please follow the same sequence to maximize efficiency and eliminate errors.
It is also very helpful to read each section before beginning to gain a visual idea of the construction.
Your project will be built in stages. If you have purchased the PC Board in its unassembled state, you will first assemble the components onto that board.
Next, you will remove the original Dynaco board and carefully label each of the associated wires. You will then install the new VTA upgrade PCB and reconnect the wires to their corresponding terminals followed by a few electrical tests and power up sequence.
Finally you will adjust the operating parameters of the amplifier. You will be instructed to adjust the DC Bias of the output tubes.
Although you are unlikely to encounter any problems, should they occur they would almost certainly be based on an improper assembly of the PC Board or the wiring to the amplifier. Therefore, to avoid the time consuming and frustrating task of troubleshooting afterwards please remember these simple suggestions:

** **Proper soldering is crucial.**
Be especially careful to avoid solder bridges or "cold" solder joints. Our experience shows that nearly **95% of all problems are associated with soldering related errors**.
** Diodes, semiconductors, and many capacitors (always electrolytic types) are polar – this means that **there is a right and wrong way to insert polarized components in the board**. If placed backwards, the component and probably others nearby will be damaged upon initial power application. Double check each step associated with these parts and once again later after you have had a chance to take a break.
** Some components are color coded with their value (mostly resistors, but there may be others). If you are not proficient at reading these codes, use an ohmmeter to double check the value of each resistor before insertion. It is very easy to confuse a 100 ohm metal film resistor with a 1000 ohm resistor or worse with a 100K ohm resistor.

Building the PCB
You have two choices for parts placement on the PCB.
This choice is purely a personal aesthetic choice, and makes no difference in performance.

1 - The traditional choice is to put most all of the parts on the top side, along with the tube sockets.
The parts that extend beyond the cutout of the chassis MUST be mounted on the bottom side of the PCB. This would include: R39, D1, C17 & C18, C6 and C8.

2 – The alternate choice is to put ALL of the parts on the BOTTOM side of the PCB, EXCEPT the tube sockets and the 2 bias pots.

The sequence of assembly has been selected so that the components with the lowest profile are installed first other components added until the highest profile components are installed last.
Refer to the component placement drawing included with the kit. Be sure to observe correct orientation of polar components (capacitors). Be sure also to check with an ohmmeter resistor values if you are not absolutely sure of how to read metal film resistor color codes.

1 – First install a jumper on V1 from pin1 to pin 6. Then install the 2 tube sockets on the TOP side of the PCB. It is VERY critical that these are oriented correctly. Notice the direction of the slot in the center of the tube socket, and make sure the socket is facing
the correct direction before soldering into place. Do not soldier more than one pin before checking that the sockets are all level with the PCB, otherwise the tubes will not stand up straight. Once the sockets are all level soldier all the remaining socket pins.
Also install the 2 Bias (50K) pots at this time on the TOP side of the PCB.
If you are using the 1st parts placement option just remember to refer to that section concerning which parts must be installed on the bottom side.
Before going ahead, make sure all 4 pots (2 on each PCB) are set at their center position.

2 - Next install all of the ½ Watt resistors.
There is also a 6.8K 3w resistor used on the quad cap B+ connection.
Then there are 4 pieces of 10 ohm resistors (2 on each amplifier) that will replace the original resistors connecting pins 1 and 8 of each output tube.
By checking the voltage across these resistors and using the bias pots you can set each tube exactly the same. Refer to the adjustment paragraph later in this manual.

The board has been designed so that a gentle radius near the body of the resistor will position the leads in alignment with the associated holes in the PC Board.
Note that the ½ Watt resistors should be inserted so that the body of the resistor rests gently against the PC Board. After inserting the resistor leads thru the board, a slight bending outward of the leads will hold the part in place until you can solder it in place.

3a - Install the diode (D1) noting the orientation. The D1 diode stripe will go thru a SQUARE hole.
3b – install the LM334 (U1) oriented as shown the the PCB layout diagram.
4 - Install the capacitors C6, C8, C17 & C18 on the bottom side.
Install C2 on either top or bottom side. Install C12, C14, & C16 on top side.
5 – Check to see that all parts have been installed on the PCB. Visually inspect all of your solder connections. Quality solder connections should be smooth and shiny. Dull surface or large blobs on the PC board should be cleaned and resoldered. In this case remove the old solder with solder wick or similar and resolder the connection.

Removing the original Dynaco board
Refer to the chassis layout diagram below.
1 - Remove the top cover (tube cage) if it is still on the amplifier chassis. It is held in place with screws) along the side. These screws will release both the top perforated metal cage and the bottom "U" cover.

2 - Remove the output tubes (and label them for reinstallation later). You should clean the dust and grime from the topside both inside and outside of the amplifier. A vacuum cleaner with a soft paintbrush will work very well here.

3 - Next begin desoldering each wire from the original PCB, including the 22K resistor connected to the front octal socket. The original Dynaco wires were solid PVC insulated. The PVC is not very tolerant of heat and therefore you should be very careful not to melt the insulation. Normally a "wetted" soldering iron tip will have better thermal transfer causing the solder to melt quickly before the heat travels up the lead of the wire compromising the PVC insulation. Also, when you remove the wire from the PC Board, it is not uncommon for the wire to "snap" at the other end.
I suggest having some replacement wire available. You can use solid PVC wire as in the original however if possible, I strongly recommend using 22 ga. stranded Teflon insulated wire in this application.
Please do not use anything larger – it is not necessary and it will compromise the reliability of your amplifier as you attempt to make reliable connections on terminals unable to withstand the strain.
Desolder each wire and using masking tape (a temporary identifier) label each wire to make installation of the replacement module much easier.

4 - The old bias components (2 - 50uF caps) should be removed and thrown away.
Also the connections to the original bias pots should be removed and thrown away, including the 18K resistor.
The selenium rectifier and the attached 1K resistor should be removed and thrown away.
The choke will remain connected between the 1st and 2nd section of the quad cap.

5 - After you have desoldered all of the wires from the original Dynaco driver board remove it by removing the four 4-40 nuts from the bottom of the board.
The board should fall loose (if not identify if any other wires that still remain).

6 - Make sure the wires on the input jack is at least 1.5 to 2.0" long before installing the new PCB

Installation of the new VTA MK3 driver PCB.

1 - Put a 4-40 screw thru each PCB mounting hole from the top side of the chassis, and on the bottom side attach a ¼" hex spacer to the screw. Then mount the board under the hex spacer, and thread a nut onto the end of the screw. The four screws & hex spacers may need to be very slightly loose when you do this to align the PCB. Once the board is aligned make sure the screws and hex spacers and nuts are all tight.

2 - Begin reconnecting the rest of the wires to the PCB. I suggest placing the stripped and tinned wire into the hole in the board and then quickly soldering the wire to the pad.
As you complete each wire, remove its masking tape identifier.
Some of the wires may need to be replaced if they are too short.
Besides the wires from the input jacks, the following connections must be made:
Attach the input jacks to the L-input and R-input of the new PCB, and the input ground(s).
Attach the 2 wires from the output tubes (pin 6) to the outer edges of the new boards at locations marked OUT.
Filament Power
Attach the **filament** power - connect the original filament wires to tube pins 7 & 8

Attach the NFB wire to the NFB connection on the new PCB.
Attach the ground wire and the B+ wire

Lastly, you will need to cut or desolder the red/black wire that connects to the old diode between the two old bias pots (which are no longer needed and can be removed if you prefer).
This wire will need to be spliced to make it long enough to reach the new BIAS connection on the PCB at the striped end of the diode D1. This diode replaces the old diode on the amp, and the old part can be removed and thrown away. Make sure that the spliced connection is good and cover it with heat shrink or electrical tape to prevent any possible electrical shorts before soldiering the wire to the striped end of diode D1 on the PCB.

3 - Once completed, inspect each connection again and carefully look for signs of melted insulation near adjacent wires or PCB traces to insure that no electrical shorts can occur.

4 – If you haven't done so already, remove the old 11.2ohm resistor and the jumper between each pair of output tubes, but keep the connection between pins 1 & 8 on each tube. Install a 10 ohm resistor from pin 8 to ground on each output tube.
You now have a 10 ohm resistor on each output tube instead of a pair of output tubes sharing the old 11.2 ohm resistor. The tube bias can now be measured across each of these resistors individually.

New DC Bias Circuitry
The original Dynaco used one bias potentiometer to adjust a pair of output tubes (one bias pot for the two tubes in each channel). To measure the bias current flowing in the pair of output tubes, Dynaco had placed a single 11.2 ohm resistor in the current path for both tubes (from pins 1 & 8 to ground), this is replaced with a 10 ohm resistor from EACH output tube (see step 4 above).
The new VTA driver PCB has one bias pot for each output tube which will allow you to balance the currents in each output tube individually. This way you can measure the voltage across each resistor and therefore the current thru each output tube individually.
With the 10 ohm resistors in place the proper bias adjustment voltage reading is now

MK3 should be set for 0.50 VDC (50ma)

This way the output tubes can be perfectly matched and balanced.
This also lowers the current about 20% from the original which will give you much longer tube life and less heat, with no loss of performance.

Dynaco conveniently routed a wire from the original 15.6 ohm resistor to the octal socket mounted on the front of the amplifier. You will now need to add an additional wire to another unused pin to permit both cathodes voltages to be present at the octal socket, for a convenient place to measure while the amplifier is in use.

ADDITIONAL NOTES:
With common house AC supplies now at 120-125vac rather than 110-115vac as it was 40 years ago, the B+ can rise to 500-550vdc at turn-on, and settle to 470-500vdc after warmup.
The B+ on the PCB input should be between 360-400vdc. The 6.8K resistor will drop about 85 volts.
You can change this resistor if needed, to drop 100v use a 8.2K 3w resistor.

Final Checkout & Initial Power Application

1 - Replace the power fuse (3A Slo-Blo) with a 2A AGC (quick trip).

2 - If you haven't already, remove the rectifier tube. Plug in and turn on the amplifier.
Examine the filaments on the new replacement driver board. All three should be illuminated.

If any tube is not illuminated, turn off the amplifier (remove the plug from the AC socket) and check the filament connections to the new PCB. Refer to the paragraph above regarding Filament Power.
Reapply power – all three driver tube filaments should be illuminated. Remove power from the amplifier.

3 - Remove the driver tubes and insert the rectifier tube and the two output tubes.
Replace the original 3A fuse. Make sure all four BIAS pots are set to their center position.

4 - Connect your DC Voltmeter to monitor the current thru the output tubes by measuring the voltage across the 10 ohm resistor located at pins 1 & 8 of the output tubes - connect the Negative lead (Black) to the chassis and connect the Positive lead (Red) to the junction of the 10 ohm resistor and the output tube pins 1 & 8.
Monitor the voltage as the tubes reach operating temperature, which takes about 15-25 seconds.

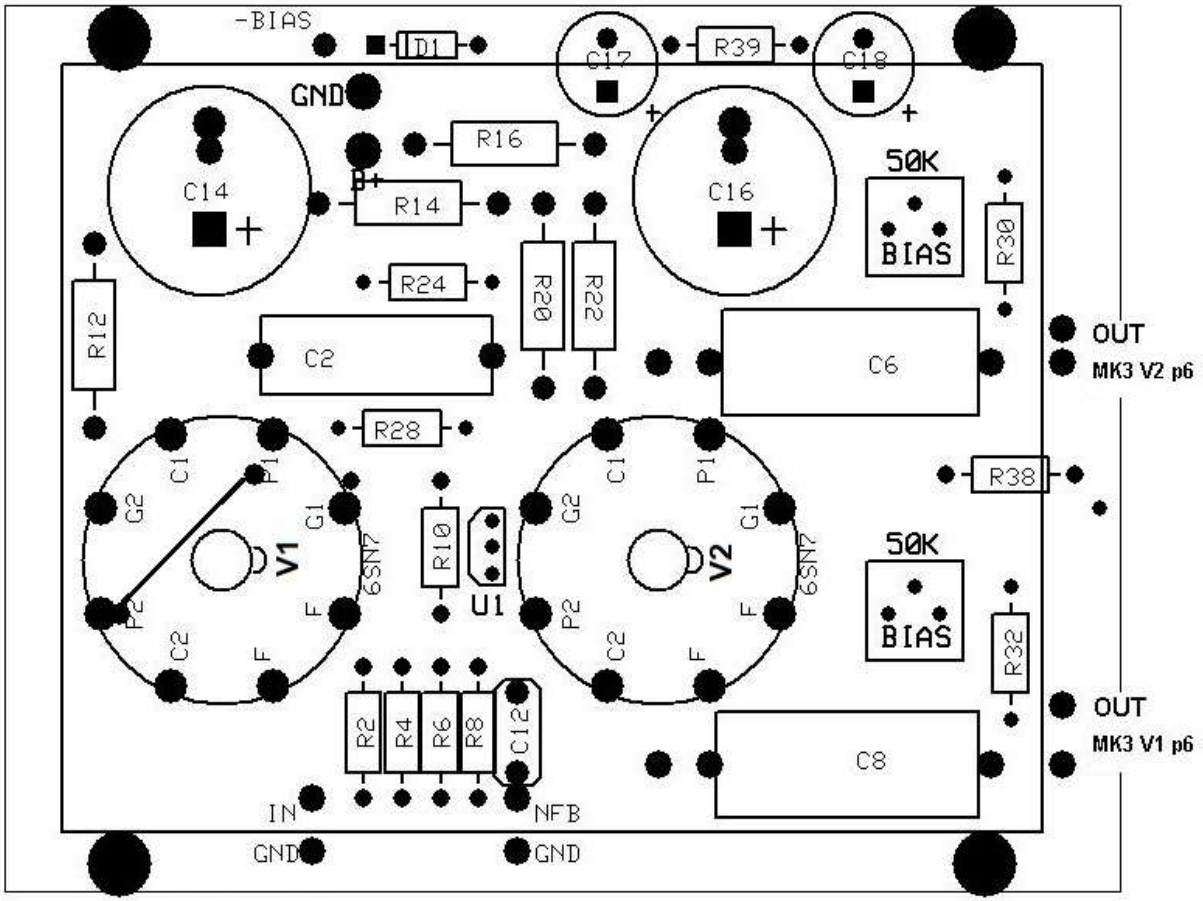
MK3
If the voltage begins to exceed 0.70 volts quickly remove power.
Otherwise adjust the Bias pot to obtain 0.50 volts.
Then check the other output tube and adjust for 0.50 volts.

5 - Install the small signal driver tubes.

6 – Replace the cover and enjoy.

MK3 octal v19

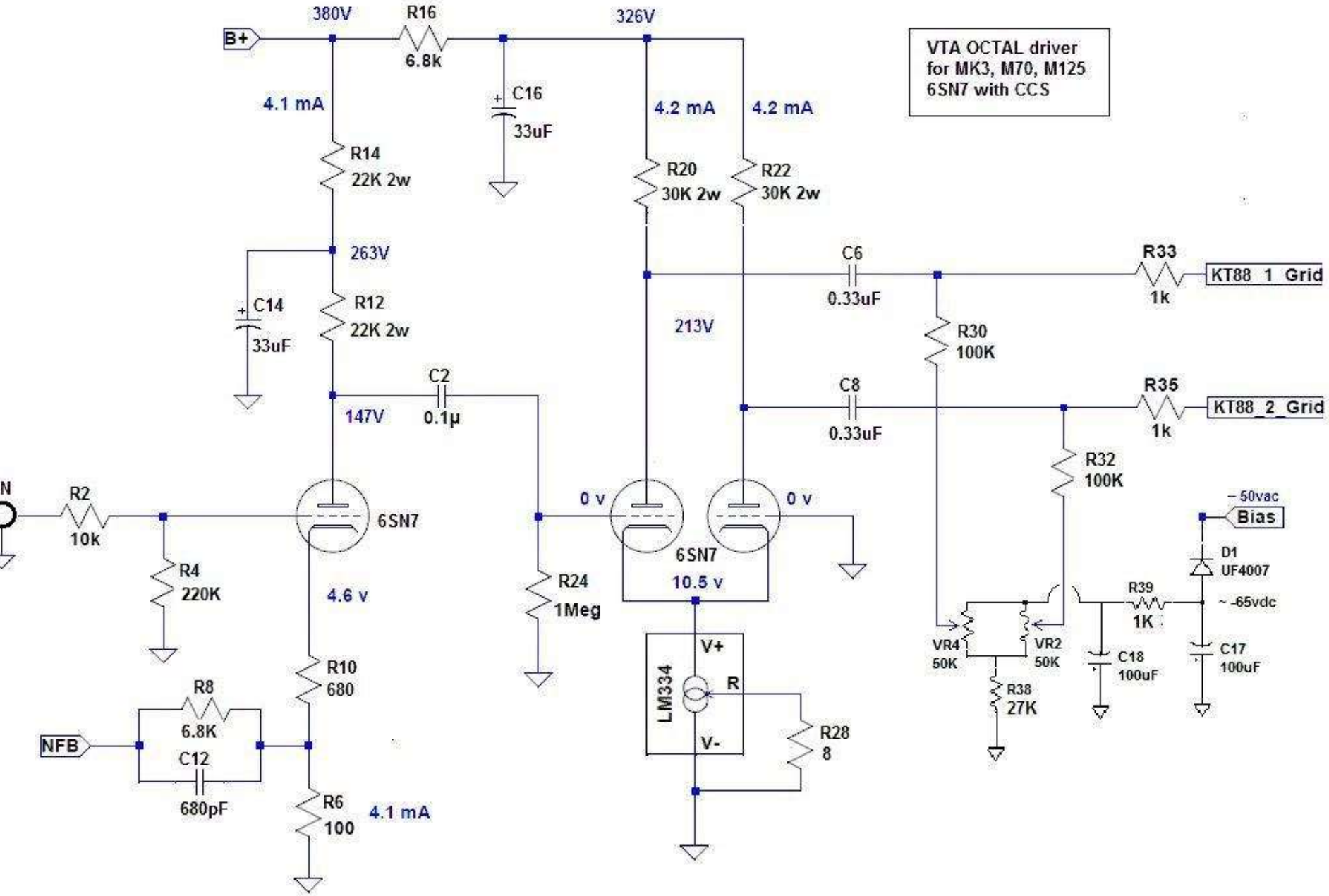
TOP VIEW

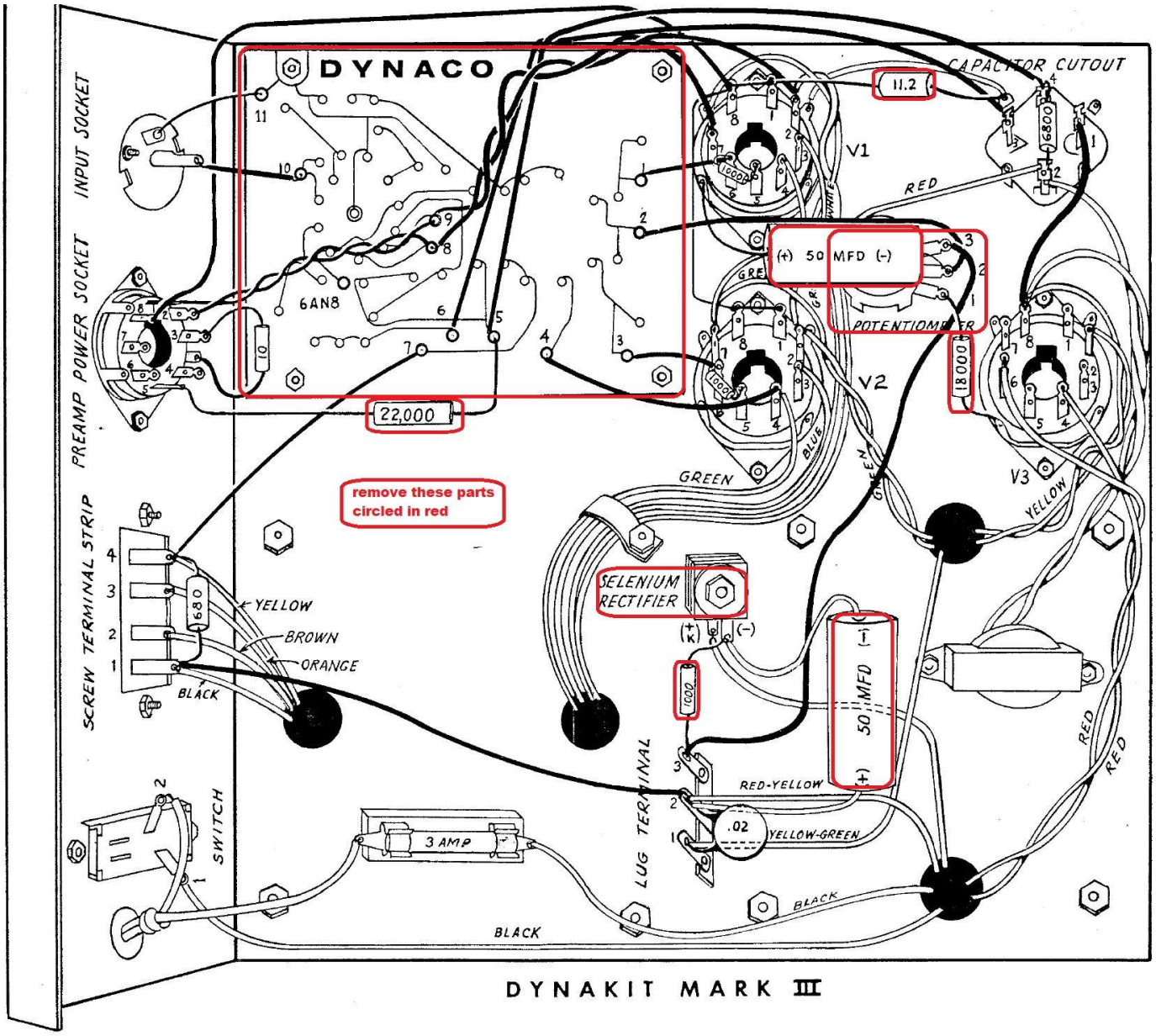


JUMPER P1 - P2 on V1

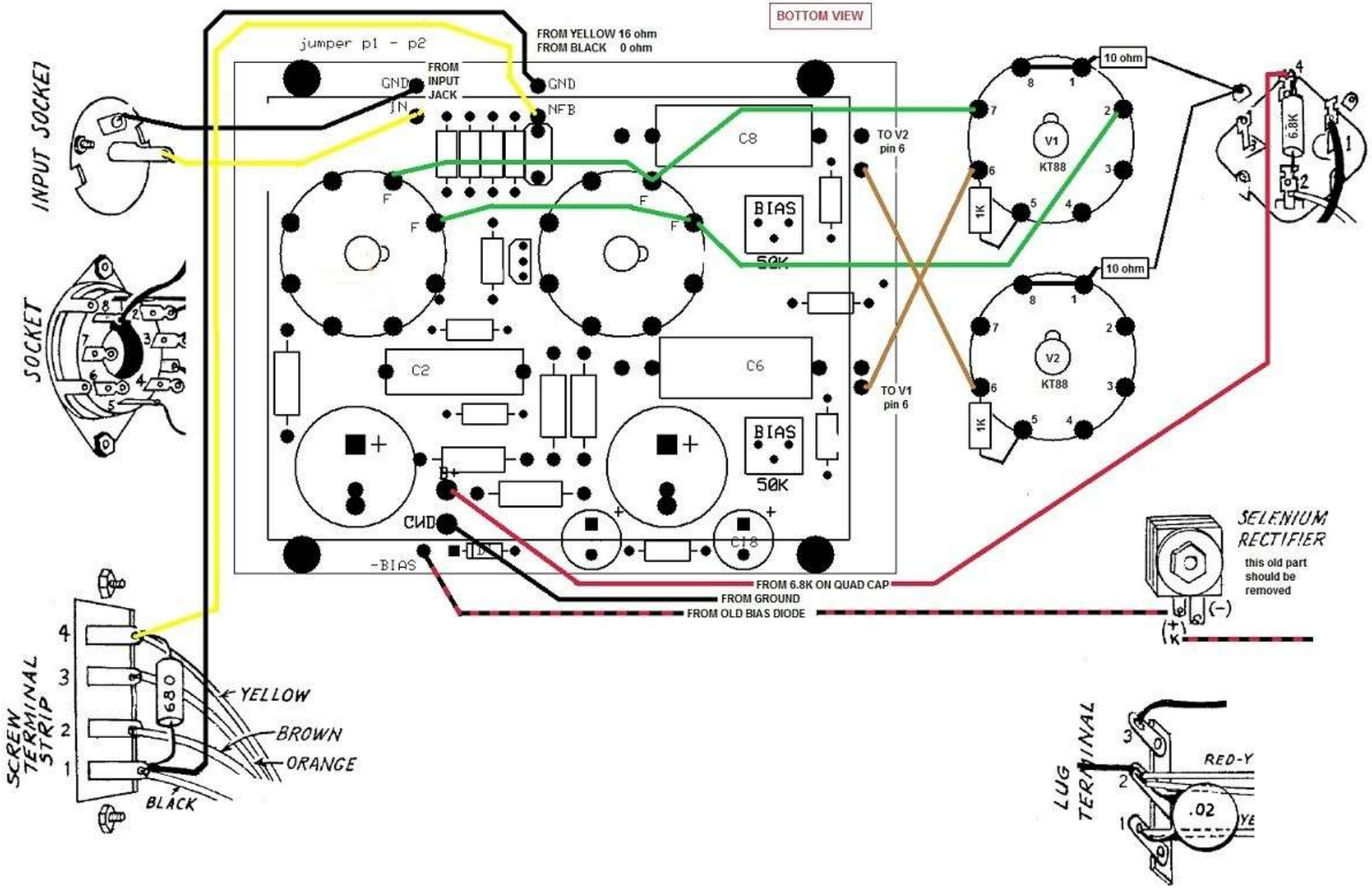
VTA MK3-M70-M125 octal driver

ITEM	VALUE	QTY
PCB		2
R2	10K	2
R4	220K or 221K	2
R6	100	2
R8	6.8K or 6.81K	2
R10	680	2
R12 & 14	20k 2w	4
R16	6.8k 1w	2
R18	not used	0
R20 & 22	30k 2w	4
R24	1M	2
R26	not used	0
R28	8.2 ohm	2
R30 & 32	100K	4
R33 – 37	not used	0
R38	27K	2
R39	1K	2
cathode bias	10 1w	4
quad cap	6.8K 3w	2
D1	UF4007	2
C2	.10 400v	2
C6 & 8	.33uF 400v	4
C12	680pF	2
C14 & 16	47uF 400v	4
C17 & 18	100uF 100v	4
U1	LM334	2
VR1 - 4	50K	4
VS2 & 3	8 pin PCB socket	4
1/4" hex spacer		8
1/2 x 4-40 screw		8
	6SN7 tube	4


VTA OCTAL driver
for MK3, M70, M125
6SN7 with CCS
B+
380V
R16
6.8k
326V
4.1 mA
C16
33uF
4.2 mA
4.2 mA
R14
22K 2w
R20
30K 2w
R22
30K 2w
263V
C6
0.33uF
R33
1k
KT88_1_Grid
C14
33uF
R12
22K 2w
213V
R30
100K
C2
0.1µ
C8
0.33uF
R35
1k
KT88_2_Grid
147V
R32
100K
N
R2
10k
6SN7
0 v
0 v
– 50vac
Bias
R4
220K
4.6 v
6SN7
10.5 v
R24
1Meg
D1
UF4007
R39
1K
~ -65vdc
V+
R
LM334
V-
VR4
50K
VR2
50K
C18
100uF
C17
100uF
R38
27K
R28
8
R10
680
R8
6.8K
NFB
C12
680pF
R6
100
4.1 mA



remove old original PCB
remove 22K resistor attached to front octal socket
remove 11.2 ohm resistor
remove two 50uF capacitors
remove bias pot (or leave but disconnect all wires to it) and the 18K resistor
remove selenium rectifier and 1K resistor


BOTTOM VIEW
jumper p1 - p2
FROM YELLOW 16 ohm
FROM BLACK 0 ohm
FROM INPUT JACK
GND
IN
NFB
C8
C2
C6
C18
BIAS
50K
TO V2 pin 6
TO V1 pin 6
V1
V2
KT88
1K
10 ohm
6.8K
B+
CWD
-BIAS
FROM 6.8K ON QUAD CAP
FROM GROUND
FROM OLD BIAS DIODE
INPUT SOCKET
SOCKET
SELENIUM RECTIFIER
this old part should be removed
(+) K
(-)
SCREW TERMINAL STRIP
680
YELLOW
BROWN
ORANGE
BLACK
LUG TERMINAL
RED-Y
.02