

O2 Desktop Headphone Amp V2.1 changes from V2.0

6/29/2014

agdr

None of the changes between V2.1 and V2.0 should make any audible difference. The changes fall into four general groups:

- Changes that make the build job easier and more straightforward.
- Board silkscreen errors and text improvements. Part labels that were on the wrong side of the board in V2.0 are fixed, some text is bigger or smaller, some text is rotated, etc.
- Fixes for any outright errors in V2.0, like the 1N4004 diode holes being too small for most 1N4004 diodes.
- Some changes that *might* result in very slightly better testing measurements, but are unlikely to be audible.

Overall PCB changes

- I've replaced the 5mm long library part for the 1/8W resistors with a newly created 6mm long part. This change applies to all the 1/8W resistors on the board, which in turn caused quite a bit of minor layout trace tweaking to make the longer part fit. The net result is the leads on the 1/8W resistors won't have to be bent quite so close to the resistor body. The 6mm long part wouldn't fit for the two power LED current limit resistors R60 & R61, so I've left those 5mm rather than go to surface mount.
- Capacitor labels are now outside the part outline in most cases. Before I put the labels right on the part outlines, which covered up half the text label when the cap was installed. This doesn't apply to the larger electrolytic and 4.7uF caps where the label will fully fit inside the part outline.
- Board text updated to V2.1.
- A great many text updates. Text on the wrong side of the board has been fixed. I've made some text larger, some smaller, some fixed font rather than variable, and rotated some to be more legible.

Power Input section & relay circuit changes

- One lead of C5 now goes to the J3 end of the trace rather than the S1 end. This is the snubber frequency shift capacitor, but it also acts as a HF noise shunt, hence tracing a smaller circuit loop.

- Pads for 0805 sized MLCC SMD ceramic parallel caps added across the leads for the large electros, C7 and C8. In addition silk screen marking on the bottom of the board for placement of radial leaded COG MLCC parallel caps on all 4 electros, including C11 & C12. This is just the typical thing of bypassing power supply electros with lower ESR ceramics. In this case it makes the CRC filter more effective at high frequencies for filtering out power line and power lead RF pickup.
- JP8 and JP9 re-oriented for better build access.
- A couple of top layer traces are routed around and under C8. Looking 20 years down the road those electrolytic caps will eventually leak and eat any traces under them. This just gives less trace for them to eventually damage. The large top layer trace from C8 to C11 is now paralleled by a similar trace under it on layer 2 for the same reason.
- C50 & R70 moved up slightly due to more room from moving J6 out further (more below).
- The error of the 1N4004 holes being too small is corrected.
- The minimum load resistors R84 & R85 re-positioned to make better use of the layout space.
- A test point has been added, JP23, to check the relay circuit's 24Vdc to the negative power rail. This won't touch the heat sink for IC11 since it sits up off the board when the bolt is inserted.

Power supply section & rear RCA jack changes

- C20 & C21 now are 15uF tantalum, up from 10uF before. I ran across a datasheet which suggested the larger output caps for the LM317/337 when using adjust pin bypass capacitors. I've seen no sign whatsoever of pre-regulator transient peaking, but just best design practices here. The new caps are 7mm wide vs. the old 6mm, which required some layout tweaking.
- JP19 is now oriented the same as JP18, with up and down wiring.
- The negative voltage regulator section now has its own star ground back to the system star ground point. In V2.0 the ground for the positive regulators continued on to the negative regulators.
- The 274R resistors on the rear RCA jack R3 and R4 are changed to surface mount from through-hole in V2.0. I did this for consistency, since the 274R resistors on the front panel 3.5mm jack are surface mount. Now there is just one 274R resistor in the BOM, the surface mount type.
- The error of the 1N4004 holes being too small is corrected.
- I have drawn a silkscreen text circle around D4 to help with the problem to the flying lead going into JP11 instead of the proper hole. Also re-oriented some text.

IC5 clipping detect chip changes

- None

Gain chip section changes

- R24 is now end mount to make better use of PC board space.
- R25 now has text saying to mount slightly above the PC board. Just to help eliminate the potential for crosstalk

3.5mm input jack, input select switch, gain switch, clipping detect output changes

- A bunch of small layout tweaks to fit in the 6mm long pads for the 1/8W resistors vs. the old 5mm.

Volume control, 4.7uF caps, DC output offset changes

- A bunch of small layout tweaks to fit in the 6mm long pads for the 1/8W resistors vs. the old 5mm.
- The two trim pots R46 & R53 are moved a little further away from the back of the volume pot just to make turning the adjust screw as easy as possible.

Output chips, their bypass and decoupling caps, and their output resistors

- A big change here. I've increased the spacing between the chips by about 1mm since there was room to do so. Just allows more air flow around the chips. Required tweaking the layout for virtually every part in the output section, along with fitting in the longer 6mm output resistor pads.
- Added a silkscreen bar marking to show where the gold bar end of the output chips go.

Pre-amp section and pre-amp RCA output jack changes

- None

Output jacks, Zobel pads, damping factor pads

- I moved the ¼" output jack forward by about 2mm to bring the end flush with the plastic nut Neutrik sells. This in turn freed up some space in the relay section to move a couple of parts forward, as noted above.