Upgrade PC Board for NwAvGuy's O2 Headphone Amplifier

V2.0, agdr, 2013, with design input from Sergey888 on diyaudio.com

Description

This DIY PC board slides into the top slot of NwAvGuy's O2 headphone amplifier in the standard B2-080 extruded aluminum enclosure. The board plugs into the O2's U3 and U4 IC sockets in place of the two NJM4556A output chips.

What the upgrade board does:

99.3% Lower Output DC Offset Voltage To The Headphones.

The original O2 has a DC offset voltage of about 3mV on each output channel going to the headphones. Half of that is inherent DC offset in the NJM4556A output chips. The other half is an input offset voltage to the NJM4556A chips caused by the input bias current of the chip going through the O2's 40.2K ground return resistors, multiplied by the 1x voltage gain of the stage. The new OPA140 chip solves this problem in two ways, resulting in an ultra-low DC offset of around just 20uV! The original 3mV offset of the O2 is 3000uV, so that is a 93% reduction. The DC offset is so low that the tiny IR voltage drop (current times trace resistance) in the PC traces going to the output jack will be about half of the offset voltage.

The OPA140 is a DC precision op amp, meaning it has very high input DC offset precision of no more than 120uV. Then in addition the chip is FET input (just pico-amps of input current), as opposed to the current-hungry bipolar inputs on the NJM4556A chip, resulting in nearly zero voltage drop across the O2's 40.2K resistors. The net result is the vanishingly small 20uV DC offset. Lower than some DMMs are even capable of measuring (they would just show zero volts).

This low offset becomes especially important on high sensitivity headphones and IEMs where just 60mV of voltage swing, or less, can be full volume levels.

Even Lower (datasheet numbers, not measured) Distortion

Replaces the standard NJM4556A paralleled output chips in the O2 with an OPA140 FET-input, low-distortion, op amp looped around a LME49600 power audio output buffer on each channel. The OPA140 + LME49600 combination has much lower distortion figures in their datasheets, by more than 10 times through the audio band, than the NJM4556A chips.

NwAvGuy's dScope THD+Noise measurements of the O2 amp on his website generally show "double zero" levels at larger voltage swings for the respective various loads. In the order of 0.001% to 0.003%. Low voltage swings showed numbers on the order of 0.0008% or so. The datasheet numbers for the OPA140 show "triple zero" numbers on the order of 0.0007% to 0.0002%.

NwAvGuy noted in his O2 design website write-up that the distortion levels of the NJM4556A chips are the limiting factor on distortion in the O2 headamp and swamps the lower distortion of the gain stage NJM2068 chip.   In fact, he mentioned there was no point in replacing the gain stage chip with a lower distortion chip since the output stage would bury it. The new lower distortion output combination can allow the low gain stage distortion levels to make it through to the headphones.   Lower distortion input chips (than the O2's original NJM2068) can now be successfully used in the O2 amp, such as the LME49720 or two LME49990 on a dip adaptor.

Higher Current Output For Low Impedance Headphones.

Each half of each of the O2's NJM4556A chip can sink or source 70mA of current. With both halves in parallel, as is the case with the O2 amp, that roughly doubles to 140mA. The new LME49600 low distortion audio buffer chip can sink or source up to 250mA of current, 100mA more than the original paralled O2 chips on each channel. The extra current capability can help with low impedance headphones.

It is important to note that due to limitations in the O2 headamps's power supply, such as a lack of heatsinking on the voltage regulators, that extra current capability in the new LME49600 chips is useful in a intermittent "music power" sense where music tends to alternate with low and high loudness levels. The O2's power supply would not however support a continuous draw of 200mA - 250mA through both buffers, as with sine wave testing.   If you want that take a look at the version of an O2 Desktop Amplifier (ODA) I've posted on DIY Audio in the headphone forum. That amp will support a continuous output of 320mA on each channel.

Nearly Zero Output Impedance

No output resistors are required with the O2 upgrade board. The 1 ohm balancing resistors on the O2 that were required by the paralleled NJM4556A sections can be shorted across. The 1R resistors can also be left on to keep things plug and play, in which case the output impedance will be the same 0.5 ohms that the O2 has.

With the O2's 1R resistors shorted, the only output impedance of the O2 upgrade board will be the tiny resistance in the PC traces leading to the output jack.

Capable of Higher Voltage For High Impedance Headphones.

Although the original NJM2068 and NJM4556A chips in NwAvGuy's O2 headamp are specified for higher +/-15Vdc power rails than the O2's +/-12Vdc rails, he noted that the resulting power dissipation would be too much for the NJM4556A chips in their small DIP8 packages. The LME49600 chip is in a larger

TO263 package soldered to a large area of heatsink foil on the upgrade board. So with the new board the O2 amplifier could be run at +/-15Vdc by replacing the two O2 voltage regulators and one capacitor. The higher voltage is useful for higher impedance headphones where the O2's original +/-12Vdc just doesn't provide enough volume.

Optional After-MOSFET Dual Power Rail LEDs.

NwAvGuy's O2 amplfier has one single LED across the power supply rails before the MOSFETs , which in turn are controlled by the power management circuit. Although the O2's LED tells when the power switch is on or off, it does not let the user know if the MOSFETs have turned on and if both power rails are working.

The upgrade board includes two optional LEDS, one on each power rail, after the O2's MOSFETs which tell when the power management circuit is on or off and if both rails are working. Two small holes are drilled in the O2 front panel to let the LED light be visible. If the LEDs are not wanted they are simply jumpered using zero ohm 1206 resistors. The two new LEDs are in series with the new optional anti-thump resistors, below, so there is no additional current draw.

Anti-Thump Turn-Off Resistors.

NwAvGuy only had IC chips as the load after the O2's MOSFETS. Since the ICs are only specified down to a certain minimum voltage level, it becomes possible for one power rail to drain faster than the other on their way to zero volts when the MOSFETs turn off, possibly leading to a turn-off thump in the headphones.

The upgrade board includes two optional resistors that allow a small amount of current to flow which allows the power rails to linearly discharge all the way to zero. The new power rail LEDs are in series with these two new resistors to make good use of that small current flow.

Power Management Latch Circuit.

In NwAvGuy's original O2 power management circuit it is possible for the batteries to recharge slightly all by themselves once the O2's MOSFETs cut off, causing the PM circuit to turn on again. The result is a motorboating or oscillation sound in the headphones when the batteries are low and the PM circuit tries to cut off.

The upgrade board includes an optional latch circuit that wires into the O2's power management circuit. The latch circuit causes the O2's MOSFET's to "latch" off, preventing oscillation, when they do turn off. Turning the O2 off for 30 seconds resets the latch circuit, although it will re-latch again if the O2 is turned on when the batteries are still low and in need of charging.

Note that with the new power rail LEDS on this upgrade board you now know for sure when the MOSFETs have turned off and the latch circuit is engaged. The O2's power light will be on, but the two new power rail LEDS will be off.

¼" Output Jack For Use With The B3-080 Case.

If the board is used in the taller B3-080 case, a Neutrik ¼" jack can be mounted underneath the board, sitting above the O2's 1/8" jack. The pads for the ¼" jack are hooked directly into the new board's output, so no further wiring is required. It just works!

Second Order Unity Gain Feeback Loop.

Credit goes out to Sergey888 on Diy Audio for his suggestion of a 2nd order feedback loop around the OPA140 and the LME49600s. The circuit limits bandwidth above the audio band to help prevent oscillations from the two chips' unnecessarily large (for audio) bandwidth capability. The 2nd order circuit drops off much faster at the corner frequency that a 1st order circuit.

Still Battery Friendly – Uses Only Slightly More Current Than the Basic O2 Amp.

Credit also goes out to Sergey888 for suggesting the OPA140 chip. In addition to having high DC precision, low noise, low distortion and a FET input, the OPA140 has very low quiescent current draw. Each NJM4556A chip in the O2 pulled about 6mA. The OPA140 only draws about 2mA. The LME49600 is run in "low bandwidth" mode of 110mHz, still ridiculously large for audio (see the feedback bandwidth limiter above) which only uses 7mA of current. So combined the new OPA140 + LME49600 combo only uses about 1mA more than the basic O2. The power rail LED option adds another 1mA to that, for a total of about 2mA over the O2, an extremely small additional draw that will only have a slight effect on battery runtime.

Can Support Op Amp Rolling

The board will support most single SOIC-8 package op amps including: OP140, OPA827, OPA627, OPA1641, LME49710, and LME49990.

The choice of the OPA140 (thanks again to Sergey888 on DIY Audio for the suggestion) is a trade-off on several factors. I was originally looking at the more expensive (single package only) OPA827 FET input op amp, a later model of the (very expensive) OPA627 op amp. The OPA627 and OPA827 both are DC precision op amps like the OPA140, meaning they have very low inherent DC offset in the microvolt range. In actual testing though the OPA140 beat the pricier OPA827 in DC offset, by quite a bit.

Numbers like 140uV vs. just 20uV. The OPA827 shows about half the THD+N numbers though at the higher frequency end of the audio spectrum.

And then there is quiescent (idle) current. The OPA140 is specifically designed and sold as a battery-friendly chip with a quiescent current draw of just 2mA or so per channel, vs. about 8mA for the OPA827 or OPA627.

Using bipolar input chips, which have even better noise and distortion numbers than these FET input chips (like the LME49990 or the LME49710) would re-introduce the large DC offset caused by input bias current through the O2's 40.K ground return resistor). Significantly improving the O2's DC output offset was a major design goal of this project, so the bipolar input chips were skipped, although they would fit and work on the board.

So there is the trade-off. The OPA140 provides significantly better DC output offset and 10x distortion improvement vs. the original NJM4556A chips in the O2 headphone amp, all at an idle current just a couple of mA above the NJM4556A (they idle at about 6mA per channel). The OPA827, on the other hand, would have even lower (by the datasheet numbers) distortion, but higher DC output offset (although still 80% less than the standard O2 amp)

Using bipolar input chips, like the LME49990, would provide even lower noise and THD figures than the FET input chips, but would re-introduce a higher DC output offset. Depending upon the chip, such as the LME49720, that offset would be about the same as the original O2's with the NJM4556A.

So… for those wanting to try other chips with the above trade-offs… single op amps in SMD SOIC-8 packages like the OPA827, OPA627, OPA1641, LME49710, LME49990 would fit and work as direct replacements for the OPA140. Keep in mind that the other chips will draw more quiescent (idle) current than the OPA140, drawing the batteries down faster.

Can Be Used Just For the Modifications

It is entirely possible to use the upgrade board just for some or all of the O2 headphone amp modifications and not populate or use the new OPA140+LME49600 output chips.   In this case the O2's NJM4556A chips would be left in place and the header pins on the upgrade board left off.   Only two additional wires, for V+ and V- would need to be run from the upgrade board to the O2 board in this case.

If the ¼" jack is being used by itself, with the upgrade board in a B3-080 case, without the board's chips populated, then two wires would need to run from the ¼" output jack to the P2 output connector on the O2 amp. The board includes zero ohm jumpers that are left off to isolate the ¼" jack from the board's circuitry.

Downsides

Everything is a tradeoff. ☺ Here are some of the downsides of this O2 Headamp Upgrade board.

- **All surface mount parts.** A higher level of soldering skill is required than through hole, although I've kept the parts 1206 sized (large surface mount) or larger to make things as DIY friendly as possible.
- **Slightly reduced battery run time due to slightly increased current draw.** Using the battery-friendly OPA140 op amps the extra draw might reduce runtime by 30 minutes or so out of 10 hours. If some of the more power hungry op amps are used, such as the OPA827, that runtime will be further reduced. If the amp is primarily used on A/C though it is a moot point.

Board Build Instructions

First double check all your parts against the part numbers and description in the upgrade BOM.

Remember that when soldering surface mount parts, things work best when you put a little solder on one of the part's PC pads first. Then with the part held from the sides with tweezers, heat that pad again and solder on that one part lead while positioning the rest of the part's pins on the pads. Then solder the rest of the pins. Then go back and touch up the initial pin that had the pad pre soldered.

Build For The Basic Upgrade Board

The 10uF capacitor C8 should be soldered on first since it sits nearby to the pins on the output chips.

Then solder on the two LME49600 output chips. First put a thin layer of solder on the heat sink area for one chip on the PC board and put some solder on the back of the chip's heatsink. I have photos of this posted on this project thread on DIYAudio.com in the headphone forum. Then, using a wide chisel tip on your soldering iron, position the LME49600 over the PCB pad and use the iron's chisel tip to heat both the end of the pad and the PC board pad, while positioning the chips' front pins. Try to get the part soldered in 10 seconds or less to prevent damage to the chip.

Next solder on all the parts except the op amp chips IC1 and IC3, and the header pins JP1 and JP2: C7, C14, C13, C11, R7, C12, C4, C5, C6, R5, R6, C3, C10, C9, R8, C1, and C2. **NOTE: the brown band on C1 and C2 is the positive end an must line up with the band shown on the upgrade PC board marking.**

Next cut off the required header pins from the 12 section strip in the upgrade BOM. You will need two rows of 4 for JP2 and 2 rows of 3 for JP1.

EXTRA CREDIT: If you wish, you can cut a 4 pin strip for the left row of JP1, such that one pin hangs off the end of the PC board tab. Then use a short length of 22AWG or 24AWG wire to solder that pin to JP5. This just provides a second connection to the O2's V- power rail. The board already has a second connection to the V+ rail via pin 6 of JP1. The upgrade board will work just fine without that extra V- connection since V- is already connected via JP2.

Next, with the O2's power supply unplugged and the batteries removed, remove the two NJM4556A chips U3 and U4. Then plug in the two 4 position strips with the short pin length down (into the O2 socket) in U3. Then do the same with the two pin strips in U4.

Now carefully line up the pins sticking up from those 4 headers and insert them into the JP1 and JP2 holes on the upgrade board. Position the upgrade board so that the left and right edges of the PCB line up with those of the O2's PCB. Then solder the two end pins on each header. Pull the upgrade board up, which unplugs the 4 headers from the O2 U3 and U4 sockets, and finish soldering the rest of the header pins to the upgrade board.

Additional Build For Anti-Thump Resistors And The After-MOSFET LEDs

Solder on R3 and R4. Notice that on the back of the LEDS there is a green band. That banded end needs to match up with the dots on the ends of the two LED markings on the PC board. Go ahead and solder on the LEDs.

Additional Build For The Power Management Latch Circuit

Solder on R9, C15, Q1, and Q2. Then cut 10 inch lengths of 22 or 24 gauge wire. I highly recommend high strand count silicon covered wire since it is as flexible as a wet noodle. Solder one end of each wire to JP6.

The two wires should route through the middle of the O2's two filter capacitors as shown in a picture posted on DIYAudio.com. The post also includes pictures showing where the other ends of the two wires solder onto part leads on the top of the O2's PC board.

Additional Build For ¼" Output Jack When Using a B3-080 Case

Solder on JP1 under the board. If the jack is to be powered by the upgrade board, solder on zero ohm jumpers R1 and R2. If the jack is instead intended to be powered by the O2 amplifier, with a build where the upgrade board is only used for the ¼" jack or other modifications, run wires from the ¼ jack pins to connection P2 on the O2 PC board, which connects to the O2's output jack.

Installing The Board

To install the O2 upgrade board, perform the following steps.

1. Remove the O2's batteries and unplug the power adapter.
2. Carefully remove U3 and U4, the two NJM4556A chips in the middle next to the batteries.

3. Plug the pins sticking out of the bottom of the upgrade board into the U3 and U4 sockets. All 8 socket positions of U3 are used, but only the top 6 socket positions of U4 are used. The bottom 2 socket holes of U4 are not used.
4. Connect the ground wire. Either a wire from the GND hole on the upgrade board near the batteries to one of the two center battery terminals, or a wire from the upgrade board GND hole near P2 to either of the upper two holes in the O2's P2 socket near the output jack.
5. If the optional LEDs on the upgrade board were installed, drill the two holes in the front panel to let the LED light out. The dimensions are given in the build instructions section.
6. If the optional power management latch circuit is installed, run the two wires to the points on the O2 board shown below and solder.
7. Reinstall the O2's batteries, if they are being used.
8. Slide both the O2 and the upgrade board into the O2's case. The O2 goes in the usual bottom slot while the upgrade board goes in the top slot. If the parts on the upgrade board hit the top of the case, just pull it to bend it up very slightly.
9. Reinstall the O2's front and back covers. Don't forget to re-install the O2's ground wire from the input jack to a cover screw.

Operation

Turn on the O2 and test your new board! The two new power rail LEDs should light up about 1 second after the O2's power switch it turned on, when the O2's power management circuit turns on.

Once the batteries run down, the new power management latch circuit will "lock" the mosfets off to prevent oscillation until the batteries are recharged. The latch circuit can be reset by turning the O2 off for 30 seconds, but if the batteries are still rundown it will just re-latch until the batteries are charged.

The two new LEDs will turn off when the batteries get low and the power management circuit turns off. You now have a way to know that has happened. With the original O2 it would just "die" with no real indication the batteries had run down and the power management circuit turned off, rather than some problem had occurred.

If only one of the new LEDs lights and the other doesn't that means an electrical problem has occurred in the O2 and only one of the two mosfets has turned on. If this happens turn the O2 off and troubleshoot the problem, keeping in mind that LED1 is the positive rail and LED2 is the negative, so you already know which power rail has died.

O2 Headphone Amplifier +/-15Vdc Power Supply Upgrade

This modification replaces a few parts in NwAvGuy's O2 Headphone Amplifier to upgrade the power supply rails voltage from +/-12Vdc to +/-15Vdc.

Although the NJM2068 and NJM4556A output chps in the O2 heapdhone amp are rated for +/-15Vdc power supply rails, NwAvGuy himself noted that this voltage increase wouldn't work with the basic O2 since it could cause the NJM4556A chips to dissipate to much heat (with certain loads) in their DIP8 packages and burn up.

The LME49600 chips on the O2 output board have no such problem, being in TO263 packages and being heavily heatsinked, allowing the power rail voltage to be upgraded. The extra 3 volts on each power supply rail translates into about 2.5Vdc of output voltage swing, which can make the difference between some high impedance and low sensitivity headphones having enough volume. Many up amp chips also tend to have slightly better specifications at +/-15Vdc than +/-12Vdc.

The parts needed are listed in on the fourth page of the BOM. Here is the procedure. **Again, just to emphasize, you can only use +/-15Vdc power rails if the two NJM4556A chips in the O2 are replaced with this O2 output upgrade board.**

1. Replace the O2's positive 12Vdc regulator U5 with the new positive 15Vdc regulator listed in the upgrade BOM (MC7815). Lay a piece of solder braid across all 3 pins of the regulator with the O2 PC board upside down (to spread the soldering heat), then heat the braid to unsolder all 3 pins at once. The regulator will just drop right out on the ground.
2. Replace the O2's negative12Vdc regulator U6 with the new negative 15Vdc regulator listed in the upgrade BOM (MC7915). Lay a piece of solder braid across all 3 pins of the regulator with the O2 PC board upside down (to spread the soldering heat), then heat the braid to unsolder all 3 pins at once. The regulator will just drop right out on the ground.
3. Remove the two battery charging resistors R1 & R2. Replace them with the new 620 ohm 1W resistors in the upgrade BOM. The higher ohm resistors are needed to keep the battery charging current the same at the new +/15Vdc power rails as it was with the original +/-12Vdc rails.
4. Remove C1 and replace it with the 1uF 50V capacitor listed in the upgrade BOM. The O2's original C1 is only rated at 25V and is between the two power rails, which are now 30V.
5. Make sure that you are using the two MOSFET's from NwAvGuy's list of choices in his BOM that are listed in the upgrade BOM. This pair has +/-30Vdc gates which will be required with +/-15Vdc rails. The other mosfets he lists only have +/-25V gates. He made the +/-30V gate parts standard in the BOM, so you most likely already have the right mosfets installed.
6. Re-install the upgrade PC board.
7. **Without headphones plugged in yet**, power up the O2 amplifier and test the output DC offset voltage with the upgrade board installed to make sure it is still under 1mV or so.
8. **Enjoy your new higher voltage O2 Headphone Amplifier!** The main difference is that on full volume the output can now swing up to around 9.5Vpeak, from the O2's original 7Vpeak. If you have high impedance and low sensitivity headphones that didn't have quite enough volume with the original O2, the upgrade should be providing more volume.