ODA BUILD INSTRUCTIONS

1. Select your build options

This version of an ODA has a number of build options. Some of these are parts that can be included – or left off – the PC board. Others are external parts that can be mounted on the chasis and wired to the PC Board. I've included an estimate of the parts cost on each option.

- **DC servos.** The output DC offset voltage of the amplifier is already very low, thanks to the feedback loop around the buffer chips with the low-offset LME49990 amplifier. The DC offset without the servo should be around 200-300uV. With the DC servo parts installed the DC offset can be as low as 25uV. If the DC servos are not desired the parts can simply be left off the PC board, saving about $20. The servo parts are all surface mount too, so skipping the DC servos saves a lot of surface mount soldering. The DC servo parts are: IC8, R44-R46, R50-R56, R59, C42, C43, C45, C49, C51-C56. Note that if the DC servos are used then R41 and R42 are not used (not populated) as the ground return is through the servo via R44 and R45.
- **Number of output buffers.** The PCB board holds 3 NJM4556A chips per channel. Each chip supplies about 120mA of current with the two sections paralleled. Only 1 or 2 buffer chips per channel can be used if that much current is simply not needed, however having 3 buffers per channel is required if powering PC speakers and not headphones. One chip per channel supplies 120mA out per channel, 2 chips supply 240mA out, and 3 chips supply 360mA. Each chip has two associated resistors to parallel the outputs and one optional bypass capacitor for a total of about $2 per chip, so a small cost savings in not populating all 6 chips. For example, in the case of IC9, the parts that would be left off are: IC9, R57, R60, and C57.
- **Output buffer SMD bypass capacitors.** I've included pads for a 0805 sized 0.01uF bypass capacitor across the power pins on each output buffer chip. In testing so far those have not been required for stability and can be left unpopulated. In rare cases if a stability issue should arise then populating these bypass chips would be the first thing to try. Cost savings is only about $0.20 per chip plus the associated surface mount soldering. C57 – C60, C63, C64.
- **X2Y EMI capacitors.** I've included surface mount pads on the power jack, RCA input jack, and output jack for X2Y radio interference suppression capacitors. These have been included as essentially a "hail Mary pass" at trying to get rid of cell phone and other hard-to-tame EMI. These capacitors can be left unpoplulated on the build and then added later if RF interference becomes a problem. Cost savings is about $0.50 each, some surface mount soldering, and these particular parts are only available at DigiKey and not Mouser, an ordering hassle. **One exception:** the C4 X2Y capacitor is 0.01uF, meant to take the place of

the 0.01uF snubber if used. So leave C7 off If you use C4, other wise leave C4 off if you use C7.

- **Clipping indicator.** The clipping indicator lights a front panel LED when the input signal, after the gain stage amplification, is within 2V of either power supply rail (works for either the +/-7V or +/-17V power rail selections). The indicator just makes it a bit more obvious when bad sound is being caused by over-driving the input gain stage than was the case with RocketScientist / NwAvGuy's O2 amplifier. The clipping indicator parts can simply be left off the PC board, if desired, for a cost savings of about $10, avoid some surface mount soldering, but leave a hole in the front panel if one had been cut for the LED (the LED can still be populated to plug the hole and the rest of the parts left off). The clipping indicators parts to leave off are: IC5, R26. R36. R39. R40, R43, LED1, LED2, LED3, D7 – D10, C26, C39, and Q1.
- **Power rail voltage selection.** The voltage switch in the back will select either +/-7Vdc or +/-17Vdc with the parts in the BOM. Since the regulators are adjustable and set with resistors any two voltage levels can be chosen by simply picking different resistors. The selection switch only affects the LT1963A and LT3015 low-noise LDO final regulators, not the LM317/337 pre-regulators which are fixed in value to always produce 18.25Vdc going into the LT regulators. The data sheets for those two LT parts give the math to use in figuring out resistor values to produce a certain voltage. The circuit is set up so that it produces 7Vdc by default and then the switch puts resistors in parallel with the "set" resistors to raise the voltage. For example, to have +/-12Vdc and +/-15Vdc rails instead as the two switch positions, R = and R = . An external 4 or 6 position rotary switch could also be mounted on the back panel and wires run to the PC board holes for the "set" resistors to give more voltage selections.
- **Attenuation resistors and volume pot.** The ODA is designed with pads for attenuation resistors, R and R , in series with the volume pot to further reduced the signal. This feature is especially useful with sources that need less than 1x gain, such as 0.5x (1/2 or 50%) gain with a source that outputs 2 volts or more (as all mine do). The default values in the BOM has a 1K attenuation resistor on each channel in series with a 1K volume pot to produce 50% attenuation, which cuts the output of the gain stage in half. So the 1x position on the gain switch (see gain switch description below) becomes 0.5x, the 2x position becomes 1x, 4x becomes 2x and 8x turns into 4x. For larger attenuation just increase the attenuation resistors. For example, 2K attenuation resistors with a 1K pot would form a 2/3-1/3 attenuator (67% attenuation). If no attenuation is needed, such as when using only iPods and other devices that only put out 0.5V or so, simply bypass the two attenuation resistors with wires (or short them out by soldering a wire across them if installed) for zero attenuation. With zero attenuation the gain stage goes straight through – 1x gain is 1x, 4x is 4x, 8x is 8x, etc. The default 1K volume pot on the BOM should yield the lowest Johnson noise, but optionally a 5K or 10K (the O2 uses a 10K pot) could be used. The attenuation resistors (if used) have to be increased if a higher value pot is used. For example, for 50/50 attenuation with a 10K pot the attenuation resistors would be 10K each, etc.

- **Gain switch gain selections.** The amount of gain in each of the 4 rotary switch positions is set by resistors on the switch. The formula is just the standard one for a non-inverting op amp: gain = (1 + Rf/Rg) where Rf = 1.5k and Rg is the resistor being selected by the switch. For example, in the case of the 301R gain resistor R , gain = (1 + 1500R/301R) = 1 + 5 = 6x gain. The default gain settings using the parts on the BOM are 1/2x, 1x, 2x, and 3x IF THE DEFAULT 1k ATTENTUATION RESISTORS ARE USED. If the attenuation resistors (discussed above) are bypassed / jumpered then the gains willgo straight through with no further reduction. For the values in the BOM bypassing the attenuation resistors gives gains of 1x, 2x, 4x and 6x. Using the formula any gain position can be modified. For example, to get 8x gain in the highest position rather than 6x, the resistor would be: Rg = 1500R / (8x – 1) = 214 ohms. An external 6 position rotary switch could be mounted on the front panel to increase the number of gain choices to 6. Just leave off the rotary switch s on the PC board and run the wires from the PC holes to the external switch. In addition to just more amplification, the gain switch is also useful to help "center" the pot further along it its rotation to avoid the inherent channel imbalance that comes with low rotation angles. For example, if a gain of 2x is being used and the pot is only turned up to about the 9:00 position, lowering the gain to 1x would allow it to rotate to about the 1:00 position.
- **Bass boost.** The PC board has holes for an additional resistor and capacitor on each channel to enable bass boost, plus holes for an external DPDT switch that makes the boost selectable. To have bass boost on all the time, just leave the switch off with the parts installed. The total cost of the bass boost parts are about $5. The parts are: R20, R21, C17, C18 and external switch.
- **Output stage ground return resistors.** The value for R41 and R42 depends on the value of the volume pot chosen (see above under attenuation and pot). For a 1K pot use 4.99K resistors, which would produce the lowest Johnson noise. For a 5K pot use 24K resistors and for a 10K pot use the 40.2K resistors found in RocketScientist / NwAvGuy's O2 amplifier in that position. **If a DC servo is being used (see above under DC servo) then R41 and R42 are not populated as the ground return is through the servo via resistors R44 and R45.**
- **Coupling capacitors and direct DC (no coupling caps) operation.** This version of an ODA has a film coupling capacitor in the middle for the same reasons that the O2 amplifier did, but here there are several build options since the capacitor interacts with the output stage ground return resistors, above section, to form a low pass filter. If the default 5K ground return resistors are being used with the default 1K pot then populating all 6 4.7uF capacitor positions on each channel, c and C, with yield a 1.12 LPF corner frequency, lower than the O2's 1.8Hz. But that many capacitors are expensive, about $50 total, so instead the capacitors can be bypassed /shorted (only one on each channel has to have the PCB holes shorted since they are in parallel) and the DC servo installed (see DC servo above) to null out input offset voltage from the source and gain stage. Another build option is to build it up like the O2, with a 10K pot (10k attenuation resistors if used), 40.2K ground return resistors, and just one coupling cap populated on each channel, C and C. This will still slightly beat the O2's corner frequency since he used 2.2uF caps. Another build option is to use the default

BOM 1k pot (and 1k attenuation resistor if used) but the 40.2K ground return resistor (slightly higher Johnson noise but same as the O2) and then just one 4.7uF coupling capacitor per channel. Using the default BOM 1K pot and 4.99K ground return resistor but with just one 4.7uF capacitor would be a bad idea, since this would put the LPF corner frequency at about 7Hz and start rolling off the bass end of your music at about 25Hz, enough to possibly be noticeable.

- **Output damping factor resistors.** This version of an ODA has PC board holes for a series resistor on each channels output that changes the damping factor. This resistor can simply be omitted and the PC holes shorted out with a jumper wire to get the lowest output impedance and highest damping factor, about 0.25 ohms. Some headphones may sound better to the user with a lower damping factor, expecially adding more bass. In an email exchange with AKG (Harman International) they said that the recommended amplifier output impedance for my AKG K550 headphones is the same as the phones, 32 ohms, +/-20 ohms in either direction. Not zero ohms. An external 6 position rotary switch can be mounted on the front panel with wires running to these PCB holes to give more damping factor selections.
- **External pre-**amp out RCA jacks and switch. External RCA jacks for a pre-amp output can be mounted on the front or back (or both) panels and wired to the PC board holes for the damping factor resistors (see section above on damping factor resistors). A series DPDT switch can also be used to cut the headphones off while the pre-amp outs are one and vise versa.
- **Zobel network on the output.** PC board pads are in place for a resistor and capacitor on each output to form a Zobel network. The network maintains constant impedance in the face of changing speaker or headphone driver impedance to prevent osciallations. The network should not be needed for most headphones but should be used with PC speakers. The Zobel parts are R74, R75, C68, C69.

2. Order your parts

The bill of materials (BOM) is written to primarily source from Mouser Electronics, as RocketScientist / NwAvGuy did with his O2 amplifier, but I have also cross referenced to Digikey, Allied (source of the B4 cases), Farnell / Element14 / Newark (merged, soon to be one company) and RS components for a few things. Mouser doesn't have the cases so those have to be bought elsewere, at Allied Electronics or Newark / Farnell. Just go through the optioins above to decided which BOM parts to remove or add, then place the order.

3. Solder the surface mount parts first.

This advice goes for any DIY build! It is always easier to solder the tiny surface mount parts if the large stuff – including through hole parts – is not on the PC board yet.

4. Solder in the smaller through-hole parts

Next populate the resistors, capacitors, transistors, and output ICs that don't stick up very high.

5. Solder in the tall parts and front panel jacks and switches.

Solder in the big and tall parts including the power supply filter capacitors C , the 4 voltage regulators, and all the front panel switches and jack.

6. Mount the rear panel parts on the panel.

Next drill the rear panel holes according to the panel hole guide, or use a custom panel, and mount all the rear panel parts. **The voltage regulators must be used with a mica washer, screw insulator, and zinc heat sink compound between the part and the panel.** Mounting the regulators to the panel without the mica washers will cause a short. The RCA jack is mounted with a screw that goes ito the hole on the side of the jack. The switches come with nuts. Once the rear panel is in place the PC board can be slid into the case from the back. Check that the panel lines up with the case mounting holes in the rear.

7. Test the power supply output voltage.

The PC board has holes in the power supply section labeled V+ (positive power rail), V- (negative power rail), and GROUND. Turn the power switch off (down), plug the AC adaptor into the wall, and then plug the power cord into the ODA. Put a DMM on DC volts and attach to the V+ and V- test points. Briefly flip on the power switch and check that the voltage is 17V x 2 = 34Vdc for the 17Vdc voltage switch position and 2 x 7Vdc = 14Vdc for the 7Vdc position. Also make sure the two green front panel power rail LEDs light up. If either power supply rail should fail one of those LEDs will not light.

8. Test the output DC offset voltage

The headphone relay takes about 12 seconds to turn on to allow the amplifier to stabilize. We want a DC offset reading quicker than that so this measurement will be taken between ground an the contacts of the relay. Using a low DC volt range (if not autoranging) on the DMM check for around 500uV (0.5mV) on each channel output to ground. Up to 1mV is OK if not using the DC servos. Note that some DMMs may not go this low, or if they do the accuracy will not be good. If the DMM says it reads down to 1mV then the reading may be rounded up to 3mV or more. Any DC reading above 10mV is too much and troubleshooting should be done to find the problem.

9. Check the relay

The headphohne output relay has a 12 second delay when power up and immediate disconnect when the ODA power switch is turned off. This one can be checked just by putting a finger on the relay and feeling for the "click" when it engauges or disengages.   Power up the amp, wait about 4 seconds, and the relay should close. If not then troubleshooting is in order to find the problem.

10. Listen for sound!

With the volume pot turned all the way down (counterclockwise) and gain switch in the lowest setting (counterclockwise) connect a signal source to one of the input jacks. Flip the input selector switch to connect that jack – up for 3.5mm in front and down for RCA in back.   Make sure that the source is not muted and audio source materieal is playing.   Connect disposable cheap test headphones to the output jacks of the ODA. Turn on the ODA power, wait the 12 seconds for the relay to close, and then slowly turn up the volume control.   There should be sound! Increase the gain switch setting as needed. If no sound comes out first suspect the source for being muted or other reasons that no sound is being fed to the ODA. IF still no sound begin troubleshooting to find the problem. The first thing to check is the power supply DC voltage again at the V+, V- and Ground test points for +/-7Vdc or +/-17Vdc.