Desktop Headphone Amplifier Build Instructions

agdr 7/14/2013

PREPARTION

1. Select your build options

This Desktop Headphone Amplifier 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 chassis and wired to the PC Board. I've included an estimate of the parts cost on each option. **The BOM includes default parts for all of these options so you can just build using the BOM as it sits.** This section just explains more about what the defaults values are and why there were chosen.

- **Number of output buffers.** The PCB board holds 3 NJM4556AL chips per channel, which is the default in the BOM. 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, and 3 chips supply 360mA. If a chip is left off of one channel, then one should also be left off the other channel to be symmetrical. Each chip has two associated resistors to parallel the outputs for a total of about $2.70 per chip, so a small cost savings in not populating all 6 chip/resistor sets.

  For example, in the case of IC9 on one channel and IC10 on the other, the parts that would be left off are: IC9, IC10, R51, R52, R53 and R54 for a total savings of about $5.

- **X2Y EMI capacitors.** I've included surface mount pads on the power jack, input jacks, and output jack for X2Y radio interference suppression capacitors. These capacitors can be left unpopulated 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 which is an ordering hassle. To leave these parts off, omit: C1, C2, C5, and C65.

One exception: the C5 X2Y capacitor is 0.01uF, meant to take the place of the 0.01uF snubber C8 if used. So leave C8 off If you use C5, otherwise leave C5 off if you use C8.

- **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 clipping indicator parts can simply be left off the PC board, if desired, for a cost savings of about $10 and avoiding some surface mount soldering. Leaving the clipping LED off will 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. R35. R38. R39. R42, LED1, LED2, LED3, D7 – D10, C27, C42, and Q1.

- **Power rail voltage selection.** The voltage switch in the back will select either +/-7Vdc or +/-16Vdc 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 +/-18.5Vdc 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. 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.

  The highest voltage selection should not exceed +/-16Vdc since the NJM4556AL chips have a +/-15V maximum on input voltage swing. The gain stage drops the extra volt with no more than +/-16Vdc rails.

- **Leave off the LT low noise voltage regulators and voltage selection switch.** Another option is to leave the LT final low noise, low dropout voltage regulators off entirely and just use the LM regulators. The net result is slightly higher ripple and noise on the power supply lines and operating at a single +/-16Vdc power supply voltage.. The LT regulator chips and associated parts are left off and the input to output PCB holes are jumpered on the board. To make this modification:
  - Leave off the following parts: IC6, IC7, R31-R34, R36, R37, C25, C26, S4. Saves about $18.
  - Jumper pins 2 and 4 of the IC6 PCB holes using insulated 20 gauge wire. This passes the positive rail voltage along with the LT regulator removed.
  - Jumper pins 3 and 5 of the IC7 PCB holes using insulated 20 gauge wire. This passes the negative rail voltage along with the LT regulator removed.

    - Replace R24 and R25 with 3.0K resistors. This changes the output voltage of the LM regulators to +/-16Vdc from 18.5Vdc.

- **Attenuation resistors and volume pot.** The Desktop Headphone Amp is designed with pads for attenuation resistors, R29 and R30 , 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 will go 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 S3 on the PC board and run the wires from the PC holes to the external switch. In addition to providing 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. Bass boost will not work with 1x gain. The parts are: R20, R21, C18, C19 and external switch**.**

If bass boost is not used, then R22 and R23 are 1.5K. If bass boost is used then R20 - R23 are all 3K as shown in the BOM for the "optional" boost.

- **Output stage ground return resistors.** The value for R40 and R41 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 and is the default in the BOM. For a 5K pot use 24K resistors and for a 10K pot use the 40.2K resistors.

  The value of the ground return resistor should always be kept at least 5x more than the value of the pot to reduce effects of the resistor on the pot's taper curve.

- **Coupling capacitors.** This Desktop Headphone Amp has film coupling capacitors in the middle to protect against DC from the source at the input winding up at the output and damaging headphones. There 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, C28-C33 and C35-C39 +C41, with yield a 1.12 LPF corner frequency, lower than the O2's 1.8Hz. Six caps per channel are the default in the BOM and will work with any resistor selections.

  But that many capacitors are expensive, about $50 total, so 2 capacitors at a time per channel can be installed with the corresponding loss of some low frequency response. Then more capacitors added as funds permit. Another build option is to build it up with a 10K pot (10k attenuation resistors if used), 40.2K ground return resistors, and just one coupling cap populated on each channel, C31 and C39. 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.

  Some typical values of attenuation resistor, volume pot, coupling capacitor and ground return resistor are:

  * 1K atten resistor + 1K pot + 6 (or few as above) 4.7uF caps + 4.99K gnd return
  * no atten resistor + 1K pot + 6 (or few as above) 4.7uF caps + 4.99K gnd return
  * 5K atten resistor + 5K pot + 2 4.7uF cap + 24K gnd return
  * no atten resistor + 5K pot + 2 4.7uF caps + 24K gnd return
  * 10K atten resistor + 10K pot + 1 4.7uF caps + 40.2K gnd return
  * no atten resistor + 10K pot + 1 4.7uF caps + 40.2K gnd return (like the O2 amp)

- **Output damping factor resistors.** This version of an ODA has PC board holes for a series resistor on each channels output (R70 & R71) 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 0.083 ohms. That is the default in the BOM. Some headphones may sound better to the user with a lower damping factor, especially 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.

- **Pre-amp out RCA jacks, buffer/amp, and external switch.** The front panel pre-amp out RCA jacks can be wired up a number of ways. A series DPDT switch can also be used to cut the headphones off while the pre-amp outs are one and vise versa.
    - **Input pass through.** Connect a twisted set of left, right, and ground channels from the PCB holes at the rear RCA input jacks to the similar terminals on JP15 feeding the front panel pre-amp output RCA jack. Leave off all the pre-amp parts: R27, IC8, C45-C49, R43-R46. This option simply passes the input signal on the rear RCA jacks on to the front panel RCA pre-amp outputs with no voltage gain or current buffering. This is a good option for sources with enough voltage swing and enough drive current to handle the additional load.
    - **Input pass through with current buffering.** Similar to above, but with the pre-amp chip installed and wired up as a current buffer. The twisted wires from the rear input RCA jack are instead run to JP14. Nothing is connected to JP15. The pre-amp parts except R43 and R45 are installed with the following values:
        - R27 = 1206 jumper (o ohm)
        - R44, R46 = 1.5K 1206 resistors
        - C45, C46 = 150pF 1206 MLCC COG ceramic, 50V.
    - **Input pass through with voltage gain and current buffering.** Similar to above, but with the pre-amp R43 and R45 chips adjusted for the desired voltage gain. The formula is the same for the amplifier's gain stage: gain = (1 + 1.5k/R45). For example, an R45 of 1.5K would give a voltage gain of 2x.
    -
- **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 R68, R69, C63, C64.
- **Skip the gain stage and run the input signal directly to the output current buffers.** This option can be used when no voltage gain is needed, either 1x or attenuation, just current buffering is required. The two gain stage LME49990 ICs are left out and all associated parts including the gain switch and gain setting resistors. The two input 10K resistors, R16 and R17, are left out. The input pad of each gain stage chip is shorted to the output (pin3

shorted to pin 6) to run the input signal directly to the attenuation resistor (if used) and to the pot. To maintain a 10K input impedance the total of the attenuation resistor and pot must add up to 10K. For example, a 5K attenuation resistor, 5K pot (50% attenuation), and matching 25K output stage ground return resistors (see ground return resistors, above). Or no attenuation resistor (shorted or 0R resistor used) and a 10K pot, with a matching 40.2k ground return resistor. Note that the gain stage can be left in and set to 1x to achive the same thing, but this option saves the cost and installation time of the parts and removes any small amount of noise generated in the gain stage. Parts left out are: R16, R17, IC1, IC2, R20, R21, R22, R23, C15, C18, C11, C12, C16, C17, C21, JP5, JP6, R7 – R12, R14, R15, S3. Note that leaving off the gain switch S3 will leave a hole in the front panel.

- **FET input op amps for the gain stage.** I'm adding this due to some good ideas post on the DIY Audio thread. The single LME49990 op amp per channel in the voltage gain stage has the same pin-out as the surface mount SOIC8 versions of the OPA627 and OPA827 FET-input op amps. You could use those FET input op amps instead of the LME49990s. All the AC parameters of the OPA627 and OPA827 are worse than the LME49990 though. The only real benefit is reduced input bias current due to the FET inputs, which would reduce current noise through the input resistor, but that is already likely smaller than the other Johnson and inherent chip noise sources.

NOT optional

- **Relay circuit is required.** The relay time delay circuit does several important things. It provides the no-thump protection on both amp turn-on and turn-off. It also provides protection against accidentally using a DC power adaptor instead of AC.

2. Order your parts

The bill of materials (BOM) is written to primarily source from Mouser Electronics 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 options above to decided which BOM parts to remove or add, then place the order. I have double-checked the BOM numbers for Mouser and Digikey, but it is possible there are a few errors in the other supplier numbers. Be sure to double check with the part type listed.

3. Tools you need

a. A model 350 Panavise is *extremely* helpful. They go for about $90 on Amazon.com. The vise is able to hold the PC board by the edges in any position for soldering, including flat.

b. 0.33 or 0.20 rosin core eutectic solder with lead. NOT the "lead free" stuff. Cardas quad eutectic is great, as is just the 020 silver bearing solder from Radio Shack in the US.
c. Magnifying glass on a stand and hand held magnifying glass. These are helpful for surface mount parts.
d. Temperature controlled solder station. Something with 80W or above is best. I use xytronic stuff from Jameco and Howard Electronics. Hakko and Weller have good stations
e. De-solder wick (braid). A vacuum desoldering tool like the Hakko 808 is also extremely useful. Radio Shack has lower cost vacuum desoldering tools with a bulb.
f. A couple of feet of 22 gauge solid hookup with with Teflon insulation for jumper wires. Sellers on eBay have lots of this. The Teflon insulation won't melt and shrink with soldering heat.
g. Digital multimeter for testing.

4. Surface-mount soldering technique

Use a pair of tweezers that have a 90 degree bend in them and sharp points to hold the sides of parts for soldering. Tack solder one side of the part first, or just one lead for parts with more than one lead like ICs. Holding the part with tweezers and using a magnifying glass, reheat that one soldered leg as needs to orient the part correctly on the PC pads. Then solder the remaining leads. Then go back with a little more fresh solder and re-solder that initial lead to make a good joint.

Using a PC board vise, like the Panavise 350, makes surface mount soldering vastly easier. The vice acts as a 3rd hand.

ASSEMBLY

Things to be careful about in the build

- Polarity of vertical (tombstoned) diodes. There are a few places in the power supply and relay section where the diodes are soldered in on edge, or "tombstoned". The PC board has words next to the diodes whether the banded end (cathode) is up or down against the board. The two 2A power diodes are mounted at a 45 degree angle, with one end down against the board. **One of the 2A diodes has the band down, the other has the band up.** Double check that the polarity of all these vertical diodes are correct.

- Be careful to get the electrolytic capacitors in with the correct polarity**.** There is a "+" on one PCB hole and "NEG" on the side with the negative stripe.
- Jumper wires**.** There are a few places that call for jumper wires (parts named "JMPR" in the BOM). The jumpers should be made of insulated 20 gauge solid wire. Wire with Teflon insulation is best since it won't shrink back when heated with a soldering iron. Several places on eBay sell Teflon wire – I've used "John's wire shop" with good results. The length of the jumpers are listed in the BOM, but it is probably easiest just to mark the wire with a pen from the circuit board hole distances. **The jumper wires need to be top-soldered as well as bottom soldered.** This means solder the jumper wire to the top PCB trace as well as the bottom PCB trace. This is done to insure there are no current carrying ability problems with the via holes.
- Down is "off" on the power switch**.** When you are first testing the power supply make sure the power switch is off when you plug in the AD adaptor.
- Solder in just one part lead first, then flip the board over and make sure the part is positioned correctly. If not, just grab the part heat the solder on the lead briefly to allow the part to be re-positioned. Then solder in the rest of the leads. This will get capacitors, switches, and jacks oriented correctly.
- The banded end (with the white stripe) of the NJM4556AL buffer chips goes in closest to the bottom of the board where the power supply is.
- The cathode (banded) side of the two blue clipping indicator voltage reference SMD LEDs are the end with a tiny amount of green paint on the lead
- The middle lead of the IC3 pre-regulator needs to be top soldered (soldered to the top PCB trace) as well as bottom soldered for current carrying ability reasons.
- Resistors R1 and R2 are used to ground the 3.5mm input jack when nothing is plugged in, reducing noise. They should only be left off – or removed – if something is wired into external input jacks J17 and J18. J18 is one channel, the bottom pin (furtherest from board edge) on J17 is the other, and the top hole on J17 is ground.
- Related to the above (h), the RCA rear input jacks do not have a switch or grounding resistors. You may hear some atmospheric noise if the input selector switch is set for the rear jacks and no source is plugged in. This is normal. There are no resistors to remove to use the rear external input pcb holes, JP1-JP3. JP1 is ground, JP2 is one channel and JP3 is the other. These can also be used for the ODAC or a rear panel external input jack.
- Remember to check the positions of the two push button switches when first testing. "In" on S4 in the back is the low +/-7Vdc power rail position used for output swings of 3V or less. "Out" is +/-16Vdc. "in" on S2 on the front panel selects the rear RCA input jacks (or JP1/JP2/JP3 if used for external input) while "out" selects the front panel 3.5mm jack (or J17/J18 if used).
- The 3.5mm input jack J2 and the 3.5mm output jack J4 look nearly the same on the outside, but are different. J2 has a switch inside and has 5 leads coming out. J4 does not have a switch and has just 3 leads.
- The 1000pF (0.001uF) axial MLCC COG capacitors solder across the leads of every 0.22uF film decoupling capacitor on the back of the PC board. This was done to avoid having to solder small 0603 or 0805 SMD parts. First solder the 0.22uF capacitor but leave the leads long. Then do

one circular wrap of the axial capacitor leads around the film cap leads, solder, and trim the excess.

- R70 and R71, the optional damping factor resistors need to be jumpered with 20AWG wire if not being used, which will probably be the case in most builds. These resistors are in series with the output so no signal will come out unless they are installed or jumpered.
- The Zobel network parts, R68, R69, C63, C64 will be left off In most builds. They are there if osciallations occur while powering small PC speakers.
- Both the 3.5mm and ¼ inch output jacks are active, along with the output tapoff JP16, and all can be used at the same time. All three are after the relay, so there will be a 6 second turn on delay for all.
- Remember that the relay has a 6 second delay to turn on!

2. Solder in the power supply power jack, switch, diodes and filter capacitors first and test.

These are J3, S1, C10, C11, C14, C15, L1, L2, D1, D2, C8, C9, R13, C5 (if used), R18, and R19. Solder the surface mount parts under the PCB first, then the diodes since they are harder to get to later with capacitors installed, then the rest of the parts. Be careful to get the polarity of the diodes and electrolytic capacitors correct. One diode band goes down and the other up as marked on the PCB. Only solder in one lead of the jack and switch first, turn it slightly as needed to get it square on the PCB (with respect to the edge where the back panel will go), then solder the rest of the leads in. The inductors L1 and L2 do not have a polarity – either direction is fine. The surface mount resistors and capacitors under the board in this section also do not have any polarity and either direction is OK.

<u>Testing</u>. Turn the power switch off (down) and plug in either a 20Vac or 24Vac AC supply. Make sure the supply is plugged into a working wall outlet. Set a DMM to a DC voltage range that will go up to 50Vdc. Place the negative meter lead on either one of the top two holes in the 4 holes marked "GND" on the PCB. Put the positive meter lead on the leftmost PCB hole for IC3 (looking at the PCB from the back). Turn on the power switch and verify that you read around +37Vdc if using a 24Vac transformer or 31Vdc if using a 20Vac transformer. The readings should be around +/-2Vdc of those numbers, so 36Vdc to 38Vdc for the 24Vac transformer. If you read anything significantly different, turn off the power switch and double check your build so far. Double check that the electrolytic can capacitors and diodes are In the right way, ,with polarity matching the PCB markings. Make sure the 2A diodes are not mixed up with the slightly small 1A diodes used in the rest of the power supply. Also verify the AC adaptor is working correctly by putting your DMM on the AC volts setting, pulling out the power cord from the PCB jack, and measuring. It should be around 24-26Vac or 20-22Vac.

If the positive voltage check above measures OK then check the negative voltage by setting the DMM on the DC volts range, putting the negative lead on one of the two GND PCB pins, and then putting the positive DMM lead on the middle lead of IC4. You should measure around -37Vdc if using a 24Vac transformer and around -31Vdc if using a 20Vac transformer, again +/- about 1Vdc. If either of these readings are significantly off then troubleshoot the parts are per the above.

<u>Theory.</u> A "24Vac" transformer will output around 26Vac if fed by 120Vac wall voltage on the primary and if not loaded. Under load it will drop a volt or two, and it will drop a volt or so if lower input line voltage is present (like 108 or 110Vac). The transformer secondary voltage and current ratings are in RMS.. Multiply by 1.414 for sine waves to get the peak voltage, which is what the capacitors will charge up to with no load. The diodes subtract 0.7Vdc on each rail, so that gives: V = [(26Vac) * (1.414)] – 0.7Vdc = 36Vdc.

3. Solder in the rest of the power supply parts and rear RCA jack parts.

NOTE! The voltage regulators MUST be bolted onto the rear panel, with mica insulating washers, for proper heatsinking before powering up and testing.

Surface mount parts first then the through-hole parts. The voltage regulators must be soldering it at a certain height from the top of the PCB to the hole to correctly match up with the back panel. See the table of hole locations and sizes for the back panel. Then test the output voltage of the pre-regulators at the test points and of the overall power supply in both the low voltage (+/-7Vdc) and higher voltage (+/-16Vdc) position.

Since incorrect power supply voltage or polarities (or missing power supply rails) can damage other parts on the ODA, the first step is to build up the power supply section and test it in both the low voltage (7Vdc) and high voltage (17Vdc) positions. If the power supply does not test correctly then stop right there and troubleshoot as required until fixed. Don't build up any of the rest of the ODA until the power supply voltages all test correctly as outlined here. This procedure may be a little different from some builds that advise soldering all the parts in first then testing the voltage. By doing things this way it guarantees the power supply is working correctly so that incorrect voltage don't take out the remaining surface mount parts.

Solder in D3-D6, D11, D12, C3, C4, C20, C21, C23, C24, C25, C26, C60, C61, R3, R4, R24, R25, R28, R31, R33, R34, R36, R37, J1 and S4. Solder in the surface mount parts under the PCB first. Then solder in the diodes, then solder in the (non surface mount) capacitors. It is very important to get the polarity right on the 4 yellow tantalum capacitors. The positive lead is marked on the PCB for those 4 capacitors. It is also very important to get the polarity on the 6 diodes correct. The banded end should match the markings on the PCB. **Note that the vertical diodes near the voltage regulators have one with band down, and the other with band up as marked on the PCB.** None of the surface mount parts in this section have a polarity, so either direction is OK.

Finally solder in the voltage regulators**, IC3, IC4, IC6, IC7, at the board-to-hole heights in the back panel table.**

Next mount the 4 TO-220 case voltage regulators (IC3, IC4, IC6, IC7) onto the rear panel heatsink holes using a mica thermal washer, nylon hole insulator, and M3 (metric size 3) bolts, lockwashers, washers, and nuts. Zinc Oxide heat sink compound (white color) should be used on both sides of the mica washer. All of this TO-220 transistor mounting hardware and heat sink compound is available at Radio Shack in the US, as the BOM mentions. For now don't tighten the transistor mounting bolts all the way down – leave them loose enough to move back and forth. Once all 4 are mounted insert the mounting screw in the RCA jack and line up the DC power jack hole. Then tighten down the 4 regulator mounting bolts.

<u>Testing</u>.

Once the power supply parts are installed as per the above, set your DMM for the DC volts range, up to 20Vdc or higher. Plug the power adaptor into a working AC outlet. Make sure the power switch is off (down) and plug in the AC power cord from the adaptor. Press the voltage selector switch "in" for low voltage. Put the negative DMM lead on either of the top two holes of the GND test point and the positive lead on JP10, the V+ pre-regulator test point. Then flip on the power switch briefly for a couple of seconds and you should get +18.5Vdc, then flip the power switch back off. Then repeat with the negative DMM lead still on GND but the positive lead moved to JP11, the negative pre-regulator test point. Flip the power switch on and you should measure -18.5Vdc. Both pre-regulator voltages should be accurate to within +/-0.3Vdc or so. If you don't measure the proper +/-18.5Vdc pre-regulator voltages, stop right here and troubleshoot the problem. Check the polarity of the tantalum (yellow) capacitors, make sure the right voltage regulator is in the right position, make sure the 6 diodes are all in the correct way to match the board markings. Check for solder bridges or bad solder joints (especially with surface mount parts, use a magnifying glass).

Once the pre-regulator voltages are measuring correctly at +/-18.5Vdc, move on to measuring the final output voltages. With the negative DMM lead always on the GND test point on the PCB (either of the upper 2 holes), put the positive lead on JP13 the V+ (positive) power rail test point. Turn on the power switch and you should measure +7.0Vdc. If so, press the voltage selector switch to the "out" position and you should now measure +16.0Vdc. Then repeat the test with the DMM positive lead now on JP12, the negative power supply rail test point. With the voltage selector switch out you should measure -16.0Vdc, and with the switch in you should get -7.0Vdc. If any of the 4 output voltages (+/-7Vdc, +/-16Vdc) do not measure correctly, stop right here and troubleshoot your problem. Check the polarity of capacitors, diodes, and make sure all 4 voltage regulators are in the right positions. Check for solder bridges and bad solder connections.

If both pre-regulator voltages and all 4 output voltage measeure correctly, then you are ready to move on to the rest of the build.

3. Solder all the surface mount parts and jumper wires next, then test the power supply voltages again to make sure nothing is shorted before moving on to the through-hole parts.

Once all the surface mount parts are soldered on, with the AC power disconnected to the board set your ohmmeter on the DMM to the low ohms range. Then measure the resistance from the ground test point to the V+, then ground to V-. The resistance should not be a dead short, like 1 or 2 ohms. It should be hundreds or thousands of ohms. The ohm reading might slowly increase or decrease to as capacitors charge. If the ohm reading is low, like shorting the DMM test leads together, you have a short that must be found. Look for solder bridges between part leads (especially between the small IC lead)s and make sure none of the IC s are in upside down.

Then once nothing is shorted go back to step 2 and once again test both power supply rail voltages to ground in either the low or high voltage position (doesn't matter). This steps makes sure that none of the surface mount parts are shorted with power applied, and/or that none of the ICs are in backward (they will short and probably start smoking if in backward when the power is applied).

5. Solder in the smaller through-hole parts

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

Repeat the steps in section 6 above about checking resistance from ground to either power rail, with the AC power unplugged to the ODA, to make sure the rails are not shorted. If so go back and find the solder bridge or other short.

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

Solder in the big and tall parts including the the front panel switches and jacks, the 6 output NJM4556A ICs, and the relay.. Once again check the resistance from ground to both power supply rails to make sure the power supply is still not shorted.

7. Double check the polarity of every capacitor, diode, and the orientation of every IC (the dot in the corner of the IC). Double check the part number on each IC to make sure they are in the correct places. Check that all of the jumper wires are installed.

Compare them all to the polarities in the layout.

8. Test the power supply output voltage again.

Go back to step 2 and once again verify that the power supply rail voltage is still +/-7Vdc in the low position and +/-16Vdc in the high position. If not, you have a short that needs to be tracked down. Double check that no ICs are in backwards (they will likely get very hot and start smoking if so – and are then damaged and have to be replaced) and the the diodes and capacitors are in the right way. Check for solder bridges and re-heat any bad solder connectors. Make sure all of the jumpers are installed.

9. Test the output DC offset voltage

The headphone relay takes about 6 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 1.6mV on each channel output to ground. 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 2mV or more. Any DC reading above 10mV is too much and troubleshooting should be done to find the problem.

10. Check the relay

The headphone output relay has a 6 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 6 seconds, and the relay should close. If not then troubleshooting is in order to find the problem.

11. Listen for sound!

Using throw-away cheapo test headphones! With the volume pot turned all the way down (counterclockwise), the gain switch in the lowest setting (counterclockwise), and the power supply switch on the lowest voltage setting (pushed in) 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 material is playing. Connect disposable cheap test headphones to the output jacks of the ODA. Turn on the ODA power, wait the 6 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. Then push the input selector switch to see if it is in the right position. If still no sound then 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. If the clipping indicator light comes on, push the voltage selector switch on the back to the higher voltage position.

REMEMBER THAT THE RELAY HAS A 6 SECOND DELAY! WHEN YOU FIRST TRY IT WITH HEADPHONES YOU WON'T HEAR ANYTHING UNTIL 6 SECONDS HAVE GONE BY SINCE POWER UP. YOU CAN FEEL THE RELAY ENGAGE BY LIGHTLY KEEPING A FINGER ON IT.

THEORY OF OPERATION

Power Supply

The incoming AC is fed to a voltage doubler circuit. Since the ground point is in the middle, at one transformer tap, rather than at the negative side the result is 2x single voltage forming a dual rail power supply. If the ground point was at the negative lead it would have been a single rail voltage doubler.

A 24Vac rms transformer secondary will produce a peak voltage of about 24 x sqrt(2) = 34Vpeak for the incoming sinusoid wave shape. Each half wave rectifier reduces that by a diode drop, about 0.7Vdc, for a total of 34 – 0.7 = 33.3Vdc which is then fed to the C-L/R-C filter. AC line voltages can fluctuate by up to 15%, in which case the highest voltage going into the filter could be 33.3 * 1.15 = 38.2Vdc. This is why the capacitors in the filter are rated at 50Vdc rather than 35Vdc, the next common lower voltage, along with my personal experience over the years that electrolytic capacitors run near their rated voltage have short lifespans. This is also why the pre-regulator ICs are specified to be the On Semiconductor LM317/LM337, which are rated at 40Vdc maximum input, as opposed to some brands which are only specified to 35Vdc maxim

In the power supply design I'm making the assumption of "bad" incoming power. That is, AC power that is noisy, which is probably more common that clean AC power. Such as having loose connections on the buildiing's AC power circuit breaker panel (which I have run into before) that sit and sizzle at full load and throw spikes back into the power line. Or the building next door on the same pole transformer is a welding shop or electroplating facility. I see a lot of audio designs run the AC power right into the regulators assuming the incoming power will always be clean. I don't.

The EMI filter capacitor on the AC input helps clean up higher frequency RF picked up by the power cord. The snubber capacitors and resistors are from Hagermann's paper. When the rectifier diodes cut off they can causing ringing in the transformer secondary due to leakage inductance. The cap across the line lowers that frequency so the other cap in series with the resistor can damp it out as energy lost in the resistor.

The voltage regulator chips have good line (incoming) rejection up to about the third harmonic (180Hz), but it drops off rapidly from there. The C-L/R-C filter has a corner frequency of around 100Hz to start rejecting about where the rejection of the voltage regulators drops off. The filter works in both directions. Downstream it keeps AC line noise in the audio band out of the regulators. In the reverse direction they keep the audio signal frequency voltage caused by the load current through the filter caps times the capacitor ESR from migrating back out to the AC power line, or at least reducing the voltage that does. The series 0.51R resistor is used to reduce the Q of the filter slightly. Simulations showed some overshoot in the response, even with the 0.2R resistance of the inductor, without the added resistance.

The major problem with half wave rectifiers is the roughly 3x higher filter capacitor charging currents that flow for short periods of time at the starting 25% of each 60 cycle half-wave. In other words, 25% of the cycle the filter capacitors are charging and the remaining 75% they are discharging supplying the load current to the circuit. Since the maximum steady state load of the amplifier will be about 325mA per channel or 750mA total, that is 750mA x 3 = 2.2A (amps!) current spike through the transformer secondary, rectifier diodes, filters caps and inductors. That is why the secondary of the transformer has to be rated at 1A for standard load and 1.8A for full load. Those secondary ratings are rms current ratings, and Hammond's ap note advises the 60% de-rating for half wave use. That is also why the rectifier diodes are 2A (2A diodes are 2A continuous, up to about 5A with spikes like this), the ripple rating on each capacitor is about 1.6A, and the inductor rating is 2.8A. Since the capacitors are in parallel they split up the 2.2A total. Note on capacitor ripple current ratings there is a chart de-rating them for frequency. So even if a capacitor is rated for 2.5A at 100kHz, at 60Hz it may be only 1.6A or less.

What all of this means is that the power supply parts going from the AC input jack to the first voltage regulator are larger due to the half wave supply than they would have been given a center tapped transformer with a full wave supply. The benefit though is that the UL/CSA- listed AC transformer remains at the wall socket, meaning this amplifier remains a "low voltage" device and does not need to be UL certified. There is also no need to wedge the power transformer in the box, leading to a smaller amplifier chassis.

The Linear Technology (LT) low noise low dropout (LDO) regulators unfortunately have rather low maximum input voltages. 20Vdc for the LT3015 negative regulator and 30Vdc for the LT1963A positive regulator. So some sort of voltage dropping circuit needs to be added in front of the regulators. In this design I've used traditional non-low dropout LM317 and LM337 chips as pre-regulators outputting +/-18.5Vdc to feed the LT chips. Not only does that solve the problem of the low input voltage on the LT chips, but provides a full pre-regulation ripple and noise reduction going into the LT chips. The result is extremely low ripple and noise out of the final LT regulators. So low, in fact, that it is likely to be lower than what would even matter given the noise factors in the rest of the chips. In other words, likely overkill, but it comes for free since the incoming LT chip voltage had to be reduced anyway.

The pre-regulators serve a second purpose of dissipating about half of the heat generated when using a 24Vac transformer. The ideal transformer for this amp would be about 20Vac, but those are hard to find. 24Vac transformers are much more common but produce about 6Vdc more than is needed which has to be burned off in heat. The LM and LT regulators both have built in temperature safe area cut-outs in case the chips overheat. By spreading out the dissipation like this it helps insure SOA cutoff won't occur, even with 24Vac transformers.

The design of the LM regulator circuit is the standard way, right off the datasheet, with the option in place of bypassing the adjustment resistor for even lower noise. The protection diode option from the datasheet is also in place, given the various capacitors in the circuit and the chance of reverse current flow through the chips. No capacitors are needed on the inputs of the LM regulators due to their close proximity to the main filter capacitors.

The output 10uF capacitors on the LM regulators serve the dual role as input capacitors for the LT regulators. Both are needed to prevent oscillations, although they are optional on the LM but required on the LT. Not that since the LM regulators are "in the way" between the LT regulators and the main filters capacitors, these input caps are **required** as per the data sheet. Tantalum capacitors are used as per the data sheets for their high frequency characteristics. Also note the 5mA or so minimum load on the LM regulators is taken care of by the current through the adjustment resistors and the quiescent load of the LT regulators, even with no further load, which insures no startup problems due to insufficient load. There is a fraction of a second of time delay in the rest of the circuit load passing current on start up, so it can't be relied upon for the initial 0 second minimum load on the voltage regulators.

The LT LDO regulators are designed with a switch that sets the output voltage to either +/-7Vdc or +/-17Vdc. The basic circuit has a fixed resistor that sets the +/-7Vdc. The switch then places a resistor in parallel across the fixed resistor to set the +/-17Vdc. Using this method there is never a time when the LDO voltage "set" resistors are unconnected, even for an instant, which would happen if switch contacts were used for both voltages. The movement of the contacts would break (ring actually) for a fraction of a second, causing the LDO voltage to go up to a 20V maximum. With this design the voltage can never be higher than +/-17Vdc. That is extremely important since the maximum voltage of some of the chips is +/-18Vdc.

The feedforward capacitor option from the LT datasheet is used to reduce noise. The output capacitor is specified to be a 390uF 25V organic polymer type. These new organic polymer caps have very low ESR – 20mR in this case – and good high frequency characteristics. Low ESR can actually be a problem with the LT LDO regulators. There is a table in the datasheet showing that for some low values of output capacitors the ESR must stay above a certain level to prevent regulator oscillations. But at 390uF the datasheet shows the capacitance value completely swamps the effect, allowing ESRs down to 0 ohms to work just fine. It is iimporant to keep this particular capacitor close to the LDO regulators to prevent oscillation, hence the layout postions of those two. Two more 390uF caps are placed directly in the output buffer section as decoupling and bypass caps.

The long PC board trace to the output buffers is done on purpose to introduce some inductance between the main power supply 390uF caps and the local 390uF caps in the output buffer section. That helps isolate the output buffer surges from the main power feed, keeping the power cleaner for the gain stage and DC servo stage. The output stage is also heavily bypassed locally with the 0.22uF film caps and the 1000pF COF / NPO axial ceramic caps across their leads. Some datasheet – such as the LME49990 sheet – state that experimentally they have found that having a electrolytic, a film cap, and a ceramic cap in parallel on each power rail gives the best results. Each cap is good for different frequency ranges. The electro for lower, the film for higher, and the ceramic for highest. The electro also provides significant energy storage, of course.

The shutdown pins on the LT regulators are not used. If board space had existed I would have added a circuit that used those pins to shut down both regulators in the even someone plugged a DC power adaptor in instead of an AC. That works by placing a small capacitor in series with an AC sense lead that is in parallel with the AC input jack. If DC is plugged in no current flows through the cap and no voltage is made for the shutdown pins. If AC is present a small bridge rectifier chip rectifies it and used the resulting DC to power the LT regulator shutdown pins, turning them on.

Gain Stage

The incoming signal into the 3.5mm or rear RCA jacks is first fed to the X2Y EMI filter, if installed. These filters eliminate RF frequencies in the 10's of mHz. Then the signal is fed to a RF filter composed of the 274R resistor and 220pF capacitor to ground. The filter gets rid of lower frequency RF noise, in the single digit mHz. The signal then progresses to the input select switch and onto the gain stage.

The gain stage is composed of two LME49990 op amps, one for each channel, in a standard non-inverting op amp configuration with a 150pF compensation capacitor to limit bandwidth to the audio band. The ground return feedback resistor is controlled by the gain switch, according to the standard op amp formula gain = (1 + Rf/Rg), where RF is 1.5K.

The LME49990 was chosen for excellent AC parameters (low distortion, low noise) and low cost. The DC parameters of the chip (input offset voltage, input bias current) don't matter in this application since the coupling capacitors after this stage block any DC from being passed along to the output stage.

Another benefit of the LME49990 is the ability to drive loads down to 600 ohms without loosing any AC performance. In fact, the chip is fully specified at 600R in the datasheet. This amplifier makes full use of this feature by specifying a 1K volume pot as standard. The total load on the chip would then be the 1K pot in parallel with the 1.5K feedback resistor and series 300R feedback resistor, for a total parallel combination of 818 ohms, still above the 600 ohm capability of the chip. But using the 1K pot instead of the more common 10K pot lower circuit noise considerably, as does the ability to use a 4.99k ground return resistor in the output stage instead of 40.2K. Johnson/thermal noise is dependant on the total resistance of the resistor involved.

The circuit has a provision for a "attenuation resistor" in series with the pot, typically the same value as the pot to form a 50/50 voltage divider. If a 1K pot is used the attenuation resistor would be 1K. The net effect is to divide the total voltage gain in half. So a set of gain switch settings with no attenuation resistor that would produce 1x, 2x, 4x, and 8x would instead produce 1/2x, 1x, 2x, and 4x with a 50/50 resistor/pot pair. This is very useful for "hot" sources that put out high source voltages. The 1/2x setting helps to add more range to the pot rather than "just barely up". For very hot source the attenuation resistor can be increased as much as needed. For example a 2k atten resistor with a 1K pot would produce a 2/3 – 1/3 voltage divider. Most of my sources require the 1/2 x position.

One downside of the lower value pot and output stage 4.99k ground return resistor, for lower Johnson noise, is the need for a much larger coupling capacitor in the middle. A high pass filter is formed by the coupling capacitor feeding the 4.99K resistor to ground. 4.99K + 6 x 4.7uF gives a corner frequency of 1.12Hz. 5 capacitors would yield 1.3Hz, while 4 would give 1.8Hz (same as the O2 headphone amplifier). Most headphones have a response curve that drops off dramatically below 50Hz (and several below 100JHz), so none of these are likely to make any audible difference. The film capacitors are pricey, about $5 each, so this is good place to save some money and/or solder in more capacitors later as funds permit.