agdr Version of an O2 Desktop Headphone Amplifier (ODA) V2.0 Build Instructions

agdr

Revised 4/25/2014

Sections: Preparation, assembly, testing, mounting in the case, advanced builds, daily operation, theory of operation

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.

- **Attenuation resistors and volume pot.** The Desktop Headphone Amp is designed with PC board holes for attenuation resistors, R31 and R32, in series with the volume pot to further reduce the signal. This feature is especially useful with sources that need less than 1x gain (just current buffering from the output stage, no voltage 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, 3x becomes 1.5x 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).

Using the attenuation resistors does reduce the maximum output swing from the amp though. The amp's maximum output of around 10.5V (rms) [14.8V peak] would be cut in half using the 1K attenuations resistors in the BOM, to around 5.25V (rms). For a lot of headphones this is just fine. For example, my AKG-K550s are full volume at 0.080V (rms). The Shure SRH-940 are full volume at around 1.8V (rms). For these headphones I need current buffering from the output stage, and actually attenuation from the "hot" sources, but no voltage gain.

If however you are running headphones that need the full 10.5V (rms) swing of the amplifier, such as many 600 ohm headphones, don't use the attenuation resistors. 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. These resistors have three holes for each on the PC board to accommodate different sized resistors. Just jumper the outer two holes and ignore the middle hole. It doesn't matter if the (bare) jumper wire touches the middle hole. With zero attenuation the gain stage goes straight through – 1x gain is 1x, 4x is 4x, 8x is 8x, 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 [and the maximum amplifier output swing is 5.25V (rms)] 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 and the amplifier's maximum output voltage swing will be 10.5V (rms). 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 in any mechanical pot. 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, R22, R23, C15, C16, to enable bass boost, plus holes for an external DPDT switch, JP12 & JP13, that makes the boost selectable. To have bass boost on all the time, just leave the switch off with the parts installed. Bass boost will not work with 1x gain. The default values given in the BOM will result in 3dB of bass boost.

  If bass boost is not used, then R24 and R25 are 1.5K, with R22, R23, C15, & C16 not used (not populated). If bass boost is used then R22 - R25 are all 3K as shown in the BOM for the "optional" boost.

- **Output damping factor resistors.** This version of an ODA has PC board holes for a series resistor on each channels output (R88 & R89) 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 (adding in the series resistors), 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. These two R88 & R89 resistor pads have three holes to accommodate different sized resistors. Just run the jumper between the two outer holes, and it doesn't matter if the (bare) jumper wire touches the middle hole.

  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. I've included a suitable optional panel mount rotary switch in the BOM.

  So in most cases the two damping factor resistor positions on the PC board will simply be jumpered across (not used) to obtain the lowest amplifier output impedance of 0.083 ohms per channel. But if you want to experiment with different damping factors the holes are there to insert resistors and give it a try! I would suggest trying the following:

  - 10 ohms, which is a common resistor value used on many amps that don't have output short circuit protection like this amp does. The resistor helps limit current spikes in those amps.
  - A resistor equal to the impedance of your headphones if under 80 ohms or so. In the case of my AKG-K550s, 30 ohms. That will result in a 50/50 voltage divider and cut the maximum output voltage swing to the headphones in half.
  - 120 ohms. A very old standard for headphone amp output impedance was 120 ohms, but that had more to do with the use of tubes at the time. Some older headphones may have been designed with that standard in mind though! Unless the headphones being used are 300R or 600R, a significant amount of the voltage swing would be dropped across the 120R resistor though.

- **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 on and vise versa. If you don't want pre-amp output then the front RCA jack can be left off entirely, although that would leave holes in the front panel in the posted CAD files. The RCA jack can be soldered in and simply nothing connected to it.
  - **Input pass through.** Connect a twisted set of left, right, and ground channels from the PCB holes at the rear RCA input jacks (JP4, JP5, JP1) to the similar terminals on JP21 (ground is the center hole) feeding the front panel pre-amp output RCA jack. Leave off all the pre-amp parts: R49, R52, R55, R56, R57, R62, R90, C41, C43, IC8. 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.

    The 3.5mm input jack on the front panel can be passed through to the pre-amp output jack in the same manner. Run the twisted pair wires from JP2 (ground) & JP3 (left and right channel) to JP21. Also in a similar way the output of the input select switch S2 can be run to the pre-amp output jack. In this case run the twisted pair wires from JP2 (ground) and JP7 (left, right) to JP21. Note that only one input can be connected to the Pre-amp output jack JP21 though. The most handy thing to do for input pass through is simply use that 3rd option of wiring the output of the input select switch to the pre-amp out. That way whichever input (rear RCA or front 3.5mm) you selected is what will be passed to the front pre-amp out RCA jacks.

  - **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 JP20. Nothing is connected to JP21. The pre-amp parts except R49 and R55 (leave these unpopulated) are installed with the following values:
    - R90 = 1206 jumper (o ohm)
    - R52, R56 = 1.5K 1206 resistors
    - C41, C43 = 150pF 1206 MLCC COG ceramic, 50V.
  - **Input pass through with voltage gain and current buffering.** Similar to above, but with the pre-amp R49 and R55 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 R49 of 1.5K would give a voltage gain of 2x.
  - **Gain stage output with current buffering.** The output of the gain stage (what feeds the pot) can also be fed via JP14 & JP15 to the input of the pre-amp stage wired up as a current buffer (JP20). This will simply feed the signal going to the output stage to the current buffer and on to the pre-amp outputs. This configuration may be useful if you know that the output of the gain stage won't exceed 2V peak or so

which is the maximum for line level audio out. But in the case of 600 ohm headphones the output of the gain stage can go all the way up to 14Vdc peak, way too much for line level audio out. One of the 3 previous methods of feeding the pre-amp outputs would be better.

- **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 oscillations. The network should not be needed for most headphones but should be used with PC speakers. The Zobel parts are R86, R87, C60, C61.
- **FET input op amps for the gain stage to bump input impedance up to 50K from 10K.** 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 (IC1, IC2) has the same pin-out as the surface mount LME49880 FET-input op amp. You could use those FET input op amps instead of the LME49990s and then the input impedance resistors, R16 & R17, can be bumped up to 50K from 10K.

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. The relay voltage regulator circuit also acts as a bleeder load on the main 1000uf and 820uF filter capacitors (for safety when handling the board) to bleed those down in about two seconds once the ODA power switch has been turned off.

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, and Allied (source of the B4 cases). Mouser doesn't have the cases so those have to be bought elsewhere, at Allied Electronics or Newark / Farnell. Mouser also does not stock the Linear Technology LDO voltage regulators but Digikey does. Just go through the options above to decided which BOM parts to remove or add, then place the order. Part stock inventories seem to come and go pretty rapidly at both Mouser and Digikey. Please contact me if you run into a part that is currently out of stock. I may be able to recommend a substitute to avoid the wait for their re-stocking.

3. Tools you need

a. A model 350 Panavise, or equivalent, 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. The Cardas solder is available in small lengths, for less money than buying a whole roll, from various eBay vendors. Just search eBay for “Cardas Quad”.
c. Magnifying glass on a stand and a hand held magnifying glass. The hand-held magnifying glass is essential 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. For running flexible wires I highly recommend silicon covered wire with high (30 or 40) strand counts. eBay has several sellers. This stuff is also known as “RC” wire (for radio controlled planes, etc.)
g. Digital multimeter for testing.
h. Tweezers for SMD part soldering. I’ve had good luck with “VETUS ESD-15” tweezers from various eBay vendors. Just $1.60 or so each plus shipping. They are solid enough not to bend at the tips and angled, which makes holding parts by the non-contact sides (1206 resistors/capacitors and SOIC8 ICs) easy.
i. A pair of Xcelite 378D pliers is mighty handy. These have very fine, but sturdy, tips that are extremely useful for bending leads, placing or extracting parts, etc. A pair on Amazon.com marketplace goes for around $12 plus shipping. The 378D have anti-static handles that are a good idea when you are using a grounding wrist strap with sensitive ICs. There is also a 378M version with regular plastic handles for about the same price.
j. A pair of flush cut pliers. These are useful for cutting off the excess part leads under the PC board. Flush cutters differ from diagonal wire cutters in that the blades bypass each other resulting in a flush “shear” cut. The more common diagonal wire cutters squeeze the wire and essentially pinch it off, leaving a diagonal cone sticking up. I use a pair of Xuron Maxi-Shear #2175 cutters, available on Amazon.com for $11 or so plus shipping. The blades on the 2175 are much heavier and thicker (and hardened) than some other flush cutters I’ve used and resist knicking by larger and thicker wires.. I’ve had a lighter duty pair of Xcelite flush cutters be damaged (nicked) by the second cut they did.

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. I’ve had good luck with “VETUS ESD-15” angled tweezers available from various eBay vendors (just search eBay for that name) for just $1.60 or so each plus shipping. 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

- <u>**Voltage! There are relatively high DC voltages present at various points in the circuit, such as +/-37Vdc at the output of the main filter capacitors when using a 24Vac wall adaptor. Between those would be 2x 37Vdc = 74Vdc, enough to cause a serious shock, as could the 37Vdc to ground. Between the power supply rails in the higher voltage position is 2x 16Vdc = 32Vdc, also high enough to cause a serious shock. Never touch the circuitry or solder connections on the PC board with your hands with the case removed while the power is plugged in! The filter capacitors can also store a charge for long periods, so before touching any parts after the unit had been powered be sure to short the capacitor leads together with a screwdriver or insulated test lead (with the power plug removed to the amplifier of course) on all 4 1000uF and 820uF capacitors to discharge them.**</u>
- 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. **One of the 3A 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 22 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.

- 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 then 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 which is very hard to see. Instead look through the lens to see the metal square in the center. Then follow that lead down to an end, and that is the end that matches up with the three dots on the PC board.
- 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 J2 (ground) and J3 (left, right channels).
- Related to the above, 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. Once a source is plugged in the only noise will be whatever the source device produces. There are no resistors to remove to use the rear external input pcb hole. JP1 is ground, JP4 is one channel and JP5 is the other. These can also be used for the ODAC or a rear panel external input jack.
- Remember to check the position of the input select switch when first testing. “in” on S2 on the front panel selects the rear RCA input jacks (or JP1/JP4/JP5 if used for external input) while “out” selects the front panel 3.5mm jack (or JP2/JP3 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.
- R88 and R89, the optional damping factor resistors need to be jumpered with 22AWG wire if not being used, as mentioned above in the options section, 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, R86, R87, C60, C61 will be left off In most builds. They are there to use if oscillations occur, but to date I’ve tried all sorts of reactive (normal and heavy) loads with the ODA and it is unconditionally stable. No need for the Zobel network that I’ve found yet.
- Both the 3.5mm and ¼ inch output jacks are active, along with the output tap-off JP22, and all can be used at the same time. All three are after the relay, so there will be a 5 second turn on delay for all.
- Remember that the relay has a 5 second delay to turn on! Turn-off is immediate since the coil loops through the on/off power switch.

- If using high impedance (600R) low-sensitivity headphones that need maximum voltage swing, don't use the attenuation resistors, as mentioned in the "options" section above. Just jumper them across with wires. The attenuation resistors would reduce the voltage swing but you need all the swing you can get for high impedance phones. For example, with a 1K attenuation resistor and 1K pot you divide the output of your gain stage in half, so the maximum 14Vdc (peak) swing gets reduced to a maximum of 7Vdc (peak), which then gets passed along to the output stage. Jumpering those two attenuation resistors so you just have the gain stage feeding the 1K pot directly allows the full 14Vdc swing to be passed along to the output stage.
- Make sure you have installed the nut on the 3.5mm input jack, as this single-point grounds the entire chassis. None of the other front panel jacks or controls use a nut, even if they ship with one, with the exception of a plastic nut on the ¼ output jack (which would be just for looks).
- Make sure all 4 front panel and rear panel panel-to-case screws are installed and tightened down. Two sets of screws ship with the B4-080 box. I recommend using the "thread forming" set that has the torx heads. These can be inserted and removed many times and leave fewer metal filings inside the case. The other set of screws, the "self taping" are more meant to fewer insertions and do leave enough filing s in the case that it needs to be shaken out after first insertion. One of these screws on each panel grounds the panel to the case.

1. Solder in the power jack, switch, diodes and filter capacitors, load resistors, output balance resistors, relay circuit and related parts first. Test the output voltage, relay voltage and relay function. This is the "CRC" (capacitor – resistor – capacitor) filter section in the first set of build photos.

First solder the surface mount resistors under the PC board in this section. These are R13, R18, R19, R20, R21, R71, R78, R82, C5, C9.

Next solder the diodes an since they are harder to get to later with large capacitors installed These are D1, D2, D15, and D16. Be careful to get the polarity of the diodes and electrolytic capacitors correct. The diodes are mounted on end ("tombstoned", as it is called). The banded end on D2, D15, & D16 goes down toward the board, while the band on D1 goes up, as marked on the PCB.

The rest of the relay circuit parts go in next, but not the relay yet since the parts are easier to solder with it out. Also not the mosfets yet (Q2 & Q3). They are static sensitive and you will get best results soldering the parts around them in first. So these are IC11, R70, R79, R83, R84, R85, C49, C50, C61.   R70 is end mounted just like the diodes, but it has no polarity. Either end can go down toward the board.

Next solder in the two mosfets, Q2 and Q3. It is best to use static precautions here – a grounded anti-static mat, a anti-static wrist strap, and a grounded soldering iron tip.

Next solder in the large parts that are left: C7, C8, C11, C12, S1, J3, and relay K1. Only solder in one lead of the J3 power jack and S1 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 jack and switch leads. Be sure to check the polarity on the electrolytic capacitors C7, C8, C11, C12, and C49 to make sure the negative stripe marked on the capacitor is opposite the hole marked "+" for each capacitor.

Finally in this build section solder in the 0.5 ohm output balance resistors R65 – R68, R72 – R77, R80 and R81.   These resistors have nothing to do with the power input, relay, or CRC power filter functions. Now is just a good time to solder them in while nothing taller is soldered in yet.

<u>Testing</u>.

<u>VOLTAGE WARNING!! There are relatively high DC voltages present at various points in the circuit, such as +/-37Vdc at the output of the main filter capacitors when using a 24Vac wall adaptor.   Between those would be 2x 37Vdc = 74Vdc, enough to cause a serious shock, as could the 37Vdc to ground.   Between the power supply rails in the higher voltage position is 2x 16Vdc = 32Vdc, also high enough to cause a serious shock. Never touch the circuitry or solder connections on the PC board with your hands with the case removed while the power is plugged in! The filter capacitors can also store a charge for long periods, so before touching any parts after the unit had been powered be sure to short the capacitor leads together with a screwdriver or insulated test lead (with the power plug removed to the amplifier of course) on all 4 1000uF and 820uF capacitors to discharge them.</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 100Vdc. Place the negative meter lead on either one of the top two holes in the 4 holes marked "GND" (JP6) on the PCB. Put the positive meter lead on JP8. Turn on the power switch and verify that you read around +34Vdc +/-3Vdc if using a 24Vac transformer or 28Vdc +/-3Vdc if using a 20Vac 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. 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 from the outside of the plug's barrel to the inside of the barrel.

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 one of the four JP6 GND PCB pins, and then putting the positive DMM lead on JP9. You should measure around -34Vdc +/-3Vdc if using a 24Vac transformer and around -28Vdc +/-3Vdc if using a 20Vac transformer. If either of these readings are significantly off then troubleshoot the parts are per the above.

Finally test the relay voltage regulator voltage and relay function. Connect your negative lead to the negative rail -V at JP9, then the positive rail to the lead of C50 closest to IC11, or to JP23 if your board has one by IC11. The relay circuit voltage should be 48Vdc +/-2Vdc.

Turn the on/off switch off for 30 seconds. Then flip it on and listen for the relay clip in 5 to 6 seconds to test the relay function. You can also feel the relay close by putting a finger on the top of it.

<u>Theory.</u> A "24Vac" transformer will output around 25 to 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 = [(25Vac) * (1.414)] – 0.7Vdc = 34Vdc.

Note that JP8 and JP9 are where two long jumper wires will go later to hook the CRC filter into the pre-regulator chips, JP10 and JP11.

2. Solder in the rest of the power supply parts, except the 4 voltage regulators, and rear RCA jack parts. This is the second "power supply" set of build photos.

Solder the surface mount parts underneath the PC board first again, then the through-hole parts. The 4 voltage regulators won't be soldered in now, nor the rear RCA jack, since they stick up quite a bit. Easiest to solder those tall parts in last.

The surface mount parts under the board are all resistors. Solder in R3, R4, R26, R27, R29, R30, R33, R34, R35, R36, R37, R38. There is no polarity here and no order to soldering in the resistors. Just start at one end and work your way across. Many of the surface mount resistors in this section have the same value used in two places, like R26 and R27 are both 3.40K resistors. One is used in the positive regulator and one in the negative regulator. For these it is easiest just to do both resistors of the same value at once, while that particular part bag from Mouser is open.

Now move on to the through-hole parts on the top of the PC board in this section. Solder in the diodes D3 – D6 next, noting that they are "tombstoned" on end again, with markings on the board for "band up" and band down". 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. Also solder in the Schottky diodes D12 and D13, making sure you align the bands with the markings on the board (the two are in different directions).   It couldn't hurt to go back and double check all the diode band directions one more time when you are done.

Next move on to the through-hole capacitors in this section. There are several different types and it is very important to get the polarity right! On the 4 yellow tantalum capacitors the positive lead is marked with a tiny, hard-to-see (use a magnifying glass here) "+" on the capacitor, which lines up with a similar "+" on the PC board. Solder in C35 and C36, the solid organic polyment 22uF caps. The half-cicle band on the top of these is the negative side, which goes opposite the "+" lead marking on the board. The ceramic capacitors C1, C2, C22, and C23 go in next. These don't have a polarity, so either end is OK to go to either side.

At this point let's review what the various "jumper" ("JP") holes in this section are for. JP10 and JP11 are the input leads to the voltage pre-regulator chips.   Later on a wire jumper will go from JP8 to JP10 to connect the positive voltage coming out of the power supply CRC filter to this positive pre-regulator input.   Similarly a jumper will connect JP9 to JP11 for the negative half of the power supply.   JP8 and JP10 are also the places to test the positive and negative voltage output of the CRC filter section, as in the last build section.

JP16 and JP17 are tests points on the output of the voltage pre-regulators, going into the final LDO voltage regulators. These are just places you can put your positive DMM meter lead, with the negative meter lead on ground (JP6) to test the pre-regulator voltage. JP16 should be +18.5Vdc +/-1Vdc. JP17 should be -18.5Vdc +/-1Vdc.   Voltage testing in this section will be done at the very end of this build, once the rear panel heat sink is attached.

JP18 and JP19 are the main power output points for the power supply section. These are set up as "disconnects" for the power supply, where one half (two of the holes) in each set of 4 go to the power supply, while the other two go out to all the parts on the board. JP18 is the positive +16Vdc voltage output while JP19 is the -16Vdc output. Later on both voltage will be checked, then these pads are jumpered across to connect the power supply to the rest of the board. JP18 jumpers up and down (any of the two holes on the bottom to any of the 2 on the top) while

JP19 jumpers sideways. The extra holes can be used to power other things in the case, like a board in the top case slot.

3. Solder the clipping indicator surface mount parts and jumper wires next. This is the third set of build photos.

The clipping comparator section around IC5 is next since it gets harder to solder with parts around it later. There are no parts under the PC board in this section. Solder in the clipping comparator chip IC5, LED current limit resistor R28, and the two window comparator reference LEDs LED1 and LED2. The build photos show IC5 being soldered on in detail. Solder one corner lead first, then the order corner, then the rest of the leads.

The two LEDs have a micro miniature sized bit of green paint on one end to show the end that lines up with the three white dots on the PC board. I had to use a 6x magnifier just an inch away to even see it. Luckily there is an easier way. Look through the LED lens. There is a fairly large metal square in the center where the LED mounts. Follow the lead inside the lens from that square down to one of the pads, and that is the pad that lines up with the three PC board dots.

Once the surface mount parts are done solder in the diode logic, D7-D10.

4. Solder the rest of the clipping indicator parts, volume control, power indicator LEDs, gain switch and gain resistors, input select switch and 3.5mm input jack and RF filter parts next. This is the 4th set of build photos.

In the build photos I have one that has a red circle around the parts related to the clipping indicator and blue circles around parts related to the input jacks and gain stage. Once again solder in all the surface mount parts under the PC board in these sections first. That would be R1, R2, R5, R6, R7 – R12, R14, R15, and R44. If you are adding the option bass boost then R22 and R23 also. There is no order to soldering in the parts. Under the gain switch the same gain resistor will often be used in 2 places (both channels) so might as well solder both of the same value in while that particular bag from Mouser is open.

R1 and R2 need some special attention. These are the zero ohm jumpers that connect the switch inside the 3.5mm input jack to ground, to ground that input when nothing is plugged in, similar to what the O2 headamp does. And like the O2 which requires those traces be cut if you are using the O2's external input holes, here you would leave R1 and R2 out if you plan to use the JP2 (ground) and JP3 (left and right channel) external input holes to wire in an external jack somewhere. These are very small 0603 sized resistors. You can either solder the resistors in or alternatively just take a piece of tinned wire and solder across the two pads in each location to jump them. Tin the wire end with solder, tin the two pads with solder, then just use your iron to melt the sold and stick in the end of the wire. Then cut off the excess wire.

Next start soldering in the through-hole parts on the top of the board in these sections. Solder in the resistors R16, R17, R31, R32, R39, R40, R41, R60, R61, R31, and R32 have a special type of PC board pad that allows two different size of resistors to be used. If you use the resistors in the BOM (special low current noise resistors) then just ignore the center hole that winds up under the middle of each resistor. Then solder in the capacitors in this section, which are C3, C4, C24, and C37. Make sure you get the polarity of C37 lined up correctly with the PC board markings. The other 3 capacitors don't have a polarity. Then solder in the transistor Q1, matching its orientation up with the board marking.

Finally solder in the volume pot VR1, the two power indicator LEDS LED4 & LED5, the gain switch S3, the clipping indicator LED LED3, the input select switch S2, the 3.5mm input jack J2.

There are more holes in this section to take note of! In addition to the external input jack holes already discussed (JP2 & JP3), JP7 is the output of the left and right channels coming out of the input select switch S2. The JP3 and JP7 may also be used later with the pre-amp stage and pre-amp RCA output jack as one of the wiring options. You can wire the pre-amp output jack J5 (later on in the build) to JP3 to pass the 3.5mm input straight out to the pre-amp out. You can use the similar holes JP4 & JP5 near the rear RCA input jack to pass those signals directly to the pre-amp out as a pass-through. Or you can use those JP7 holes to simply pass whichever jack is selected by the S2 input select switch on to the Pre-Amp out. In that pre-amp build section there is also the option of running any of this through the pre-amp chip with no gain (just a current buffer) or with voltage gain. Lots of options for flexibility!

If you should use a board in the top case slot that does additional things, like parametric bass boost or crossfeed, these same holes would be useful to take signals from the OCA input and pass along to that board for processing.

5. Solder the two gain op-amps and the parts around them (including bass boost if desired), the two jumpers, and the 560uF capacitors next. This is the 5th set of build photos.

The parts around the two gain stage op-amps IC1 and IC2 where not installed in the step above to leave more room for the soldering iron when installing these chips. Now is the time to break out all the anti-static soldering stuff again. ☺ Anti-static mat that is grounded, anti-static wrist strap, and grounded soldering iron tip. The go ahead and solder in IC1 and IC2. The dots on these two chips should both be pointed toward the nearest edge of the PC board. The photo build pictures for this section that I've posted show installing these chips in detail. Once again solder the lead on one corner first, check chip placement over all the pads, then solder the leg on the diagonal opposite corner to hold the chip. The solder all the other legs. Make sure you look at the soldering job through a magnifying glass, and looks sideways in addition to straight down from the top to check for solder bridges.

With the two op amps in place solder in the rest of the through-hole parts in this section. R24, R25, C9, C10, C13, C14, C19. If you are using the optional bass boost then solder in those through-hole parts too, C15, and C16. Note that the bass boost resistors under the board, R22 and R23, were installed in a previous step if using bass boost. Also remember from the BOM that if you use bass boost the resistor values change in a few places. Without boost you just have R24 and R25 at 1.5K ohms. With bass boost R22, R23, R24, and R25 are now all 3K ohms. Bass boost works by putting two of the 3K resistors in series at higher frequencies, which then reduces back to 1.5K (two 3Ks in parallel are 1.5K). R24 and R25 are similar to R31 and R32 in the previous build section, with an unused hole in the middle under them.

Fiinally solder in wire jumpers JPR1 and JPR2, along with 560uF capacitors C38 and C39. The jumpers MUST be insulated wire since they sit over traces on the top of the board. 22 gauge solid insulated wire is recommended. C38 and C39 are solid polymer caps that have the negative end marked again with the dark half-circle on the top. Make sure that is opposite the lead marked "+" on the board.

6. Solder the DC output offset null parts and twelve 4.7uf 63V films capacitors in next. This is the 6th set of build photos.

Next up is soldering in all the DC output offset null adjust parts. There are several surface mount resistors in this section which should be soldering in first again. These are R42, R43, R45, R47, R48, R50, R63, R64.

Then solder in the through-hole parts for the DC output offset null section, R51, R54, R58, R59, D11 and D14. The two diodes are "tombstoned" again on end. The cathode band on both D11 and D14 should be down toward the PC board.

The trimmer pots R46 and R53 should be placed in the center first by measuring the center lead to the outer leads with an ohmmeter and turning the trimpot wiper. They should measure about 5.7 ohms or so from the center leads to the end when centered. Once centered go ahead and solder them in with the metal adjustment screw towards the volume pot.

Now solder in all the red 4.7uf WIMA film coupling caps and DC offset null filters, C25 – C29, C30 – C34, C40 and C42. The leads on the caps sometimes are slightly offset so feel free to turn them 180 degress to make them match up as well as possible with the PC board markings. These caps do not have any polarity. There is plenty of space around the caps either way, it just looks nice to have the spacing between them all uniform.

7. Solder the 6 output buffer chips and related bypass film caps and 560uF resistors next. This is the 7th set of build photos.

Solder in the decoupling caps first. Try to keep them as straight up and down as possible, and centered over the PC board markings, so they won't interfere with installing the output chips. Solder in C51, C52, C53, C54, C55, C56, and C59.

Then solder in the output chips. The gold bar marking on one end of the chips goes toward the bottom (power supply) part of the PC board. These are IC9, IC10, IC12, IC13, IC14, and IC15. I show in the photo build pictures how they solder in. Solder just the pin on one end first, then flip the board around and make sure the chip is straight up and down and flat against the board. Bend as needed. Then solder the pin on the other end of the chip and flip the board again to double check placement. If all looks good then solder the rest of the chip's pins.

Finally for this section solder in the two 560uF caps C57 and C58, making sure that half-moon negative side marking on the top is opposite the "+" marking on the pc board again.

8. Solder the output ¼" jack, 3.5mm jack, the two RCA jacks, Zobel network, and damper resistor jumpers next. This is the 8th set of build photos.

The PC board allows resistors to be placed in series with each output, R88 and R89, to change the damping factor as an option. These same holes can instead be wired to an external rotary switch to provide a selection of several damping factors. Most of the time though these two resistor positions will simply be jumpered across with some bare wire into the holes to allow for the lowest possible output resistance from the amplifier. Go ahead and solder in these jumpers, to the out holes in each of the R88 and R89 positions (the middle hole is not used), or damper resistors if you are using those.

Places have been left for an optional Zobel network, R86, R87, C60, and C61. A Zobel network can be used with certain loads that are causing oscillation to help prevent the oscillation by maintaining a constant load impedance over a wide frequency range. I have not found a need for these in any of my testing. The amp so far is unconditionally stable, as was the O2 headphone amplifier which used the same output chips in the smaller CIP8 package. So you should leave these 4 parts unpopulated.

Now it is finally time to solder in all the remaining tall stuff on the board! ☺ These parts are the 3.5mm output jack J4, the ¼" output jack J6, the pre-amp output jack J5, and the rear RCA input jack J1. With all of these jacks it is best to just solder one lead in first, then flip the board and make sure the jack is oritented correctly to match the marking on the board, and that the jack is sitting flat against the board, or there can be trouble later when the front panel is installed. Just heating up the one solder lead and pressing on the jack with your thumb will push it flat against the PB board.

9. Solder the pre-amp parts and chip next, if used. This is the 9th set of build photos.

10. Attach the rear panel to the voltage regulators for heat sinking. Be sure to use the mica insulating washers and nylon screw insulting washer in the TO-220 mounting kit (specified in the BOM) for each of the 4 voltage regulators.. This is the 10th set of build photos.

11. 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, including jumpers for R88 and R89 if damping resistors are not used.

Compare them all to the polarities in the layout.

TESTING

NOTE! The voltage regulators MUST be bolted onto the rear panel, with mica insulating washers, for proper heatsinking before powering up and testing with a load. With no load on the amplifier output, as tested here, it is OK to run it for about 10 seconds at a time with no heatsinks.

1. Test the power supply output voltage.

Turn on the amp's power switch and measure the DC voltage between ground, JP6 (any of the 4 holes), and the V+ output of the power supply which is either of **the two lower holes in JP18**. You should read around +16Vdc. Then do the same for the negative power supply rail. Measure the DC voltage between ground on JP6 and either of **the two rightmost holes on JP19**. You should read around -16Vdc.

If either the positive or negative output voltage is wrong, measure the pre-regulator voltage for that rail. Ground (JP6) to JP16 will give you the positive pre-regulator voltage, which should be around +18.5Vdc. If this voltage is correct but the +16Vdc output isn't, then the problem is with IC6 or the parts around it. For the negative rail a similar thing applies. The negative pre-regulator voltage is measured between ground and JP17, which should be around -18.5Vdc. If this is good but the -16Vdc isn't then the problem is IC7 or the parts around it.

If either the +18.5Vdc at JP16 or the -18.5 at JP17 is wrong, go back one step further to the voltages feeding the pre-regulators. For the positive rail that is between ground and JP8, which will vary between 22 and 28Vdc depending on what power transformer you are using (20Vac or 24Vac). Similarly on the negative rail the voltage feeding the negative pre-regulator is on JP9 and will be -22Vdc to -28Vdc depending upon transformer used. If either of these voltage are wrong at JP8 and JP9 then troubleshoot your CRC filter the 820uF and 1000uF capacitors and the parts around them. Make sure your solder connections on the resistors R18-R21 under the board are OK.

For problems with any of these power supply sections, the first thing to check is make sure you have all the parts in the correct locations, especially the four voltage regulator chips. It is easy to get the regulator chips swapped. Then check the polarities of all the capacitors and diodes to make sure they are correct. The next thing to do is check all your solder connections with a magnifying glass. If any connection looks at all questionable don't hesitate to re-heat it with your iron for a couple of second and add more solder if needed. Finally the thing to do is trace the circuit voltages with your DMM all the way from the AC input, through the diodes (which is now DC), through the filter capacitors, through the pre-regulator stage, and on to the LDO stage. The schematic has voltage readings at various points.

2. Measure the resistance from the power input pins to ground.

This step checks to make sure there is not a direct short on one of the power rails on the board before hooking the power supply output to the rail. Set you DMM on a low ohm range, if not automatic, and measure the resistance between the **top two** pins of JP18 to ground (JP6). The reading may move one way or the other over the next few seconds as various capacitors charge up. But the reading should be fairly high, like 5K or 50K ohms. It should not be nearly a dead short, like 1 ohm or 3 ohms. If it does read a short then you need to go back and look at the solder connections on the board carefully with a magnifying glass to find the solder bridge. Also look carefully at the pins of all the surface mount ICs to make sure pins are not solder bridged.

Then do the same with the negative supply rail by measuring the resistance from the **left two** pins of JP19 to ground, JP6. Again you should not read anything near a dead short, like 1 ohm, but something in the thousands or tens of thousands (or higher) ohms. Again, if you read a short, check those solder connections with a magnifying glass.

3. Connect the power supply disconnects and re-measure the power supply voltages.

Once you have verified in the previous step that you have no power rail shorts, connect the two power rails to the power supply output. TURN THE oda POWER SWITCH OFF AND WAIT 15 SECONDS FOR THE CAPACITORS TO BLEED DOWN. Connect the positive power supply by jumpering one hole from the top two on JP18 to one hole on the bottom two. Then connect up the negative power supply by jumpering one hole from the two on the left side of JP19 to either one on the right side of JP19. Once done, then turn the ODA power switch back on and once again measure the DC voltage between ground and JP18 to make sure it is still +16Vdc. Then the same for the negative rail, JP19 to ground, which should measure -16Vdc.

Congratulations – you have just successfully powered your ODA! If you have any of the capacitors or IC in backwards you will know it pretty soon as they will likely start smoking. If that happens switch the power off, unplug the AC transformer, and go about unsoldering the part and replacing it with one in with the correct polarity.

4. Test the output DC offset voltage and adjust for a minimum amount

This measurement tests the amount of DC voltage present on each channel at the output jacks. **The headphone relay takes about 5 seconds to turn on to allow the amplifier to stabilize so you will need to wait 5 seconds for the relay to close before you will see this measurement.** Using a low DC voltage range (if not autoranging) on the DMM check for around 500uV(that is MICRO volts!) 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 simply say zero. Any DC reading above 1.6mV is too much and troubleshooting should be done to find the problem.

If you meter is capable of measuring down to 10 or 100uV, go ahead and turn the trimpot (R46 and R53) for that channel to minimize the DC offset. These are 25 turn trimpots, so you have 12 turns available in each direction from center. You can get it all the way down to the 30uV or so range. If you meter is not capable of measuring below 1mV, and you are reading zero, then just leave the trimpots as they are. Since you mechanically centered them during the installation they will already be close to a null.

A good way to connect your meter test leads to the amplifier output for these tests is get a 3.5mm or ¼" plug and unscrew the shell to expose the solder or screw contacts inside that would normally connect to an audio cable. You can separate those a bit and clip on your meter ground to the shield contact, then the other test lead to each channel's signal out contact one at a time.

Once you have the DC offset minimized near zero on the output of both amplifier channels, go on to the next step.

5. 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 +/-16Vdc.   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 5 SECOND DELAY! WHEN YOU FIRST TRY IT WITH HEADPHONES YOU WON'T HEAR ANYTHING UNTIL 5 SECONDS HAVE GONE BY SINCE POWER UP. YOU CAN FEEL THE RELAY ENGAGE BY LIGHTLY KEEPING A FINGER ON IT.

Mounting the PCB in the case and single point grounding

The PC board is designed to fit in a B4-080 case from Box Enclosures. The board slides in the second slot up from the bottom, not the bottom slot.   The case has an extrusion running the length of the case at the bottom for a center screw. Using the 2nd slot up provides plenty of space between that extrusion and the bottom of the circuit board.

The back panel should already be attached to the PC board from building up the power supply in build step #3 above.   Slide the PC board into that second slot from the back. The rear panel should fit well and the 4 holes in the panel corners should line up with the bolt slots tapped into the case. Go ahead and screw in the rear panel bolts – they come with the case. Note that the lower left hole has a countersink around it that cuts through the blue anodization and exposes the aluminum to make a good electrical ground contact between the panel and the case.   Using a small dab of aluminum no-ox compound used by electricians, such as Ideal Industries 30-024, available at Home Depot in the electrical section or at Amazon.com, helps insure that an aluminum oxide film won't develop and keep a good connection.

The front panel just fits right on the front of the box. All the jacks and control shafts should fit through properly.   The lower right corner of the front panel is also countersunk the same way as the back to make good electrical contact with the bolt to the panel and case.   The 3.5mm input jack is the single-point ground for the entire case. It has a countersink around it to slice through the anodization and allow the nut for the jack to make good electrical contact with the panel. Thread the jack's nut on and carefully tighten it down firmly but not extremely tight using fine tip pliers.   The front panel holes on all the other metal jacks and control shafts are large enough to insure they don't touch the panel. This insures that the entire case is grounded by just the one connection to the 3.5mm input jack and no ground loops form around the case.

Install the rest of the front panel screws (that came with the case). Install the knobs on the volume and gain switch shafts. If using the Alps gain switch with the T18 shaft just push these two knobs on, after aligning the mark where you want it.   If using the Alpha gain switch a setscrew type knob will be needed.   Line the setscrew up with the flat part of the shaft and tighten it down. The knobs should not rub against the front of the panel. If so, pull them out slightly to create proper clearance. If you are using a metal knob that does not have a plastic insulator between the shaft and the knob, it is especially important that the knob does not touch or run against the front panel, otherwise you can form a ground loop and inject noise.

None of the jacks or controls use nuts (that they came come with) except the 3.5mm input jack to ground the panel.

If desired some rubber feet can be attached to the bottom of the case. In the BOM I've specified some feet that have adhesive but also have a center section that would allow a screw to be used.   If using the screw (a number 8 metal screw), drill an appropriate hole first with the PC board out of the box (so metal filings don't short out the PCB parts), mount the feet, clean out at filings, then put the PCB back in the case.

ADVANCED BUILDS FOR EXPERIENCED BUILDERS

These builds require more building experience with electronics and more mechanical board and case work experience.

1. Using a B4-160 case and 2 PC boards to mount an ODAC

A special type of build can be done using 2 of this amp's PC boards end-to-end in a longer B4-160 case. The rear board only has the power supply parts populated while the front board has everything BUT the power supply parts populated.   Three 22 gauge stranded wires are twisted together and run from the V+, GND, and V- PC board holes in the rear board to carry the power supply current to the similar holes in the front PC board. The audio out of the ODA can go to the PC holes either by the rear RCA jack or by the front 3.5mm jack on the front board. The input select switch on the front panel will select accordingly.

The two main benefits of this build are (1) ease of mounting an ODAC board on top of the rear PC board where the front 2/3 of the circuitry would normally be and (2) more separation between the power supply and the amplifier circuit. This build essentially keeps the noisy stuff like the input AC power and digital USB input on the ODAC in the back, while the quiet analog parts are on the front PC board.

2. Using the top slot of the B4-080 case to mount an ODAC

Enough space exists in the B4-080 case to slide a PC board into the top slot of the B4-080 case, then mount an ODA between the top of this board and the top of the B4-080 case. The case does not have a center screw extrusion on the top like it does at the bottom, so this board can go into the very topmost slot. The board may have to be lifted slightly to clear the 820uF capacitors. If a blank perboard or solid copper clad board is used there is space in the front section to mount an ODAC under the top board. The audio out of the ODA can go to the PC holes either by the rear RCA jack or by the front 3.5mm jack. The input select switch on the front panel will select accordingly.

DAILY OPERATION

Here are the day to day operating instructions once your Desktop headphone amplifier is built.

Input voltage and current. The amp can accept an AC adapter in the 20Vac – 24Vac range. 24Vac is extremely common, although the 20Vac will warm up the chassis less. As a general rule of thumb, the output current of the amplifier per channel will be about 1/6 of the secondary RMS current rating of the transformer, due to the half wave rectifier (1/3) and two channels (another ½). For example, a common 500mA secondary 24Vac power transformer will yield about (1/3)(500mA) = 167mA, then divide by two for about 80mA per channel. To get the full rated 320mA per channel with all 6 output buffer chips installed you will need the 1.8A (1800mA) 24Vac Triad wall transformer or equivalent. They also make a 20Vac 2A transformer that is preferable, but also about $30 at Mouser..The power jack is the 2.1mm pin / 5.5mm outside variety and is rated up to 5 amps.

Turn the volume down initially! This is always good advice for any amplifier. The first time you use the amp, until you get familiar with the volume and gain settings, turn the volume pot all the way down (rotate all the way counter clockwise) to keep from accidentally blasting out the headphones or IEMs. Then turn the gain switch to the lowest setting (rotate counterclockwise again) and then slowly turn up the pot. If not loud enough (or to get the pot more centered) turn up the gain switch to the next level, etc. **And remember, the headphone protection relay takes a full 5 seconds to come on after power up. So wait 5 seconds before turning up the pot or the gain switch the first time.**

Clipping indicator. When you start seeing the red clipping indicator light by the volume pot flash, your maximum signal swing out of the gain stage is within 2.4V of the power rails. Not clipping yet, but will be soon (clips at about 2V). Lower your volume pot and/or reduce the gain switch to lower the gain stage amplification and eliminate clipping.

Input jacks. The input selector switch switches between the rear panel RCA jacks and the front panel 3.5mm jack. The 3.5mm jacks are grounded (no noise) when nothing is plugged in, but the RCA jacks are not. So on full volume and full gain you should not hear any background hiss or noise with the 3.5mm selected and nothing plugged in, but you might with the RCA jacks selected and nothing plugged in. If you hear hiss and background noise while using either jack try unplugging the connector going to your source. You may find the hiss and noise is actually coming from your source device since this amplifier itself is extremely quiet in terms of background noise.

Output jacks. The ¼" jack and 3.5mm jack are wired in parallel. Both can be used at the same time.

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.

Output Stage

The output stage consists of up to 3 NJM4556AL chips in parallel (6 op amps total since those are dual chips) per channel. Each op amp section can source or sink up to 70mA, for a total of around 320mA per channel. The buffers are wired up in the standard non inverting op amp current buffer configuration with the output tied back to the inverting input, causing the output voltage to follow the non-inverting input voltage.

The 0.5 ohm resistors are used to properly balance the current load amoung the paralleled chips. The 0.5R value was chosen based on an application note from a power op amp manufacturer which recommends that balancing current be around 1% of the total op amp's output current. Putting op amps in parallel like this increases the signal to noise ratio (a good thing) since the signal increases linearly but the noise adds by root mean square.

Minimum load resistors

The two 10k resistors to ground on the output of each channel are there to provide a minimum load on the output buffers. I have no proven need for these, but electrically this seems like a good idea given the multiple paralleled buffers that all have balancing currents being passed between them.

Damping factor resistors

In series with each channel on the way to the output jacks is a damping factor resistor, R70 and R71. With these resistors jumpered (zero ohms) the output impedance of each channel is just the parallel combination of 6 0.5R resistors, which is around 0.083 ohms. A lot of people these days believe that the lower the output impedance of the amplifier the better, so that maximum voltage from the amplifier reaches the headphones. Otherwise the headpone impedance and the output impedance of the amplifier form a resistive voltage divider. This ratio of headphone impedance to amplifier output impedance is called Damping Factor. A small amp output impedance gives a large damping factor.

But there is also a school of thought that a higher damping factor (a higher amp output impedance) may actually give better (best) sound with some headphones. For example, when I asked AKG tech support the question about my AKG-K550 cans they responded that the headphones are designed for an output impedance that matches the headphone, 30 ohms in this case, +/-20 ohms – not zero (or very low) ohms. I've also seen a recent "high end" headphone amplifier being sold with a damping factor switch on the front to select various damping ratios. That switch just puts various levels of resistance in series with the outputs, same as can be done here with an external rotary switch connected into the two damping resistor locations on the PC board. I've included the part number of a suitable panel-mount DP6T rotary switch in the BOM.

Zobel network

PCB holes have also been left for a Zobel network, which is a resistor and capacitor from the output to ground on each channel: R68, R69, C63, C64. A zobel network helps to maintain a constant impedance vs. the changing impedance of a speaker (or headphone) vs. frequency. The Zobel capacitor looks like an open at lower frequencies and more like a short at higher frequencies, which effect only puts the Zobel resistor to ground at higher frequencies. The impdenace of a speaker works just the opposite way with higher impedance at high frequencies. In the context of this amplifier a Zobel network would be most useful to help stop any oscillations that may occur when powering small PC speakers (under 1 watt) or any headhpones that are a problem. I have not seen any oscillatory behavior with this amp on a 100Mhz oscilloscope but I'm including the Zobel networks just for completeness. Some other well known DIY amps include an optional Zobel.

<u>3.5mm and ¼" output jacks</u>

The two types of output jacks, ¼" and 3.5mm, are wired in parallel and both can be used at the same time. Some headphones use ¼" plugs and others use 3.5mm.

<u>Output anti-thump and DC protect relay circuit</u>

The relay contacts are in series with both output channels. The relay circuit produces a 6 second delay that allows the amplifier to stabilize, come to operating temperature, and prevents any turn on thumps from making their way to the headphones. The relay coil is looped through a second set of contacts in the power switch, so that turning off the power immediately cuts the power to the relay and hence cuts off the headphones, preventing a turn-off thump. There is a reverse-EMF suppression circuit across the relay coil consisting of a reverse biased diode and a zener diode. When the relay coil magnetic field collapses at turn off a very high voltage reverse EMF spike is created. This forward biases the diode and allows the zener to clamp this voltage at a safe level, and the zener also dissipates the energy as heat. This diode + zener circuit actually allows the relay to turn off faster than the more typical simple reversed biased diode alone, and it prevents contact pitting.

The relay control circuit is a simple RC timing circuit that has been inverted to take advance of the longer "tail" on the resistor's voltage. This requires a second mosfet stage to again invert the signal and drive the relay. The power for the relay circuit is taken off the +/-18.5Vdc pre-regulator outputs and then is applied to a 48Vdc relay coil. The 48Vdc coil will not energize if a 24Vdc power adaptor is used by mistake (as opposed to a 24V AC adaptor). Using a DC adaptor only "lights up" one power supply rail, which is not enough voltage to meet the pull-down minimum for the 48V relay coil. This prevents the headphones from being connected in the event of a DC adaptor accidently being used.