

Paradise R3 Assembly and troubleshooting guide

Version 5, 28.05.2013

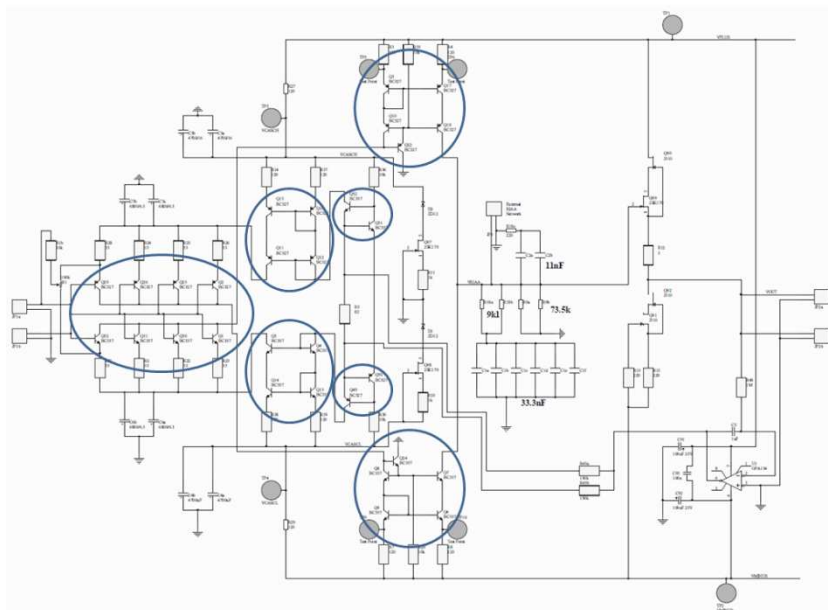
Welcome to the Paradise R3 phono stage, which arguably is one of the finer phono preamplifiers out there. The circuit concept is from Joachim Gerhard, circuit design and simulation from Michael Borresen, the power supplies from Frans de Wit, and the layout from Alfred Hesener. This assembly and troubleshooting guide is intended to help you assembling the components onto the PCBs, and power them up safely.

Assembly

The recommendation for the assembly is to start with the low-height components (resistors and LEDs), then continue with the larger devices. The large electrolytic capacitors should go in after all of that, and the heatsink with the power devices should be last. If you want to put the SMD decoupling caps on the bottom side, do that before the heatsink assembly but it doesn't matter much.

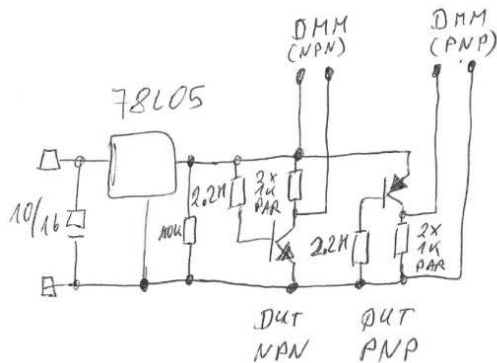
Do not put wires into the power supply jumpers (between the shunt regulator and the amplifier section) at this point.

A lot has been said about the BC327 and BC337 transistors, the important thing is that they should be matched for current gain, especially the transistors in the input stage. Here, the PNPs and NPNs should have as identical current gain (10% or better) as possible. The gain should be high (use the -40 grade). Various beta builders have reported that the PNPs have higher gain than the NPNs, you may have to get a batch of 100...200pcs to select from and get the proper mix. If identical gain is not possible, try to have identical gain at least for the input stage, and then have identical gain for the transistors in each current source, but the matching of current source to current source is not so critical, according to my experience.



This picture has the groups of transistors circled that should be grouped as good as possible, again with the 8 input transistors being the most critical.

If you cannot use a DMM with current gain measurement function, or a curve tracer, here is another proposal how to match the bipolars:



How does it work: The base resistor determines the base current, and the transistor will amplify it with its current gain. This will lead to a voltage across the collector resistor, and the values are so that the voltage you can measure there corresponds to the gain (e.g. 400mV == hfe of 400). It is not super-precise, but good enough for matching.

On the input stages, the absolute BEST would be to have 8 PNPs and 8 NPNs that have matched gain, collector-base leakage current and no noise issues. Unfortunately that will require expert equipment, so we can make do with just matching 8N and 8P with current gain, that is good enough in 99.9999% of the cases. To match the current gain between the left and right channel input stages will help to remove gain differences between the channels, helping imaging. If that cannot be done, at least the NPNs and PNPs of the input stage should match between each other (within N and within P separately, that is).

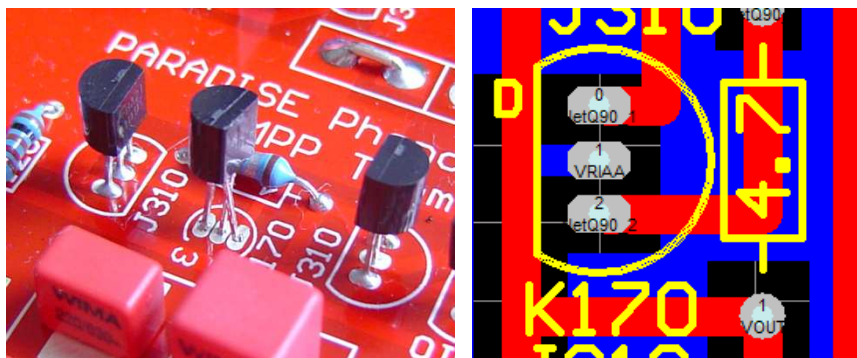
This input stage is the key piece of the amplifier, so spending time to make the match as good as possible is definitely worth it. While you are at it, you may want to match the 330ohm resistors as well, their influence is less (and 1% resistors are already pretty good), so that may be borderline to overdoing it. So, how good is good enough? If you can get the matching down to +/-10% that is good. More precise is better of course but obviously tends to be a little difficult. My recommendation would be to try and achieve closest possible matching for the input stages, and then match every other subcircuit within itself.

For the other subcircuits of the amplifier, take another look at the schematic further up in the assembly guide, it has circles around sections of the circuit in which the transistors should match to each other. They do not necessarily have to match to the other devices. This is because the transistors in these subcircuits work closely with

each other, and outside of those the absolute parameters don't matter much for the function of that subcircuit, thanks to the great design that MiiB did.

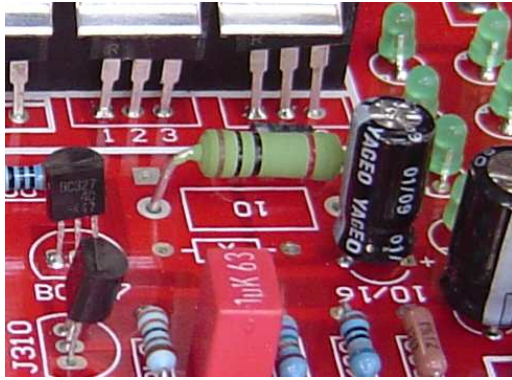
On the absolute numbers, the circuit will work with pretty much all the values in the -40 grade, so that's not an issue. The team has a feeling that if the current gain gets very high (above 550 or so), the amplifier may have a tendency to oscillate, but we did not perform all the tests to make sure that that REALLY is the reason. It is clear though that with lower values it should not oscillate. My recommendation would be to use values around 400 for both N and P for the input stage, whereas the other parts of the circuit can use higher gains (but please make sure you have those decoupling caps in place on the bottom side of the PCB).

The K170 JFETs can be replaced with J107, but please note these have a different footprint. See the picture below how that can be accomplished.



Don't worry if your PCB does not look like the picture, this was taken from the R2 version. Pin spacing is shown in the second picture, where the Drain is marked with a "D", and the sequence is D-G-S, whereas for a J107 the sequence is D-S-G (in both cases, looking at the flat surface of the package). Please note that this introduces offset in the output buffer, which the servo will correct.

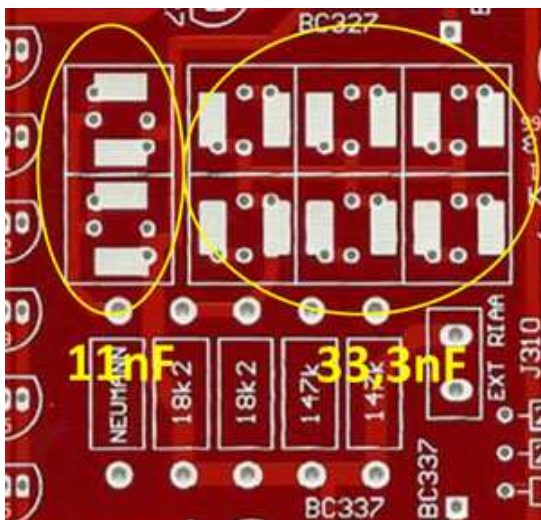
The recommended resistors are 0.6W metal film, pin spacing 10mm. Selection is not really critical, but opinions may differ (for these as for any other component on this board). The LEDs have little markers inside the footprint circle, that indicate where the "long" wire (+, or anode) should go. All the electrolytic capacitor footprints have a "+" marking for correct polarity. The power resistors are best mounted with some mm of spacing above the PCB, to improve the airflow around them and not put their heat into the PCB.



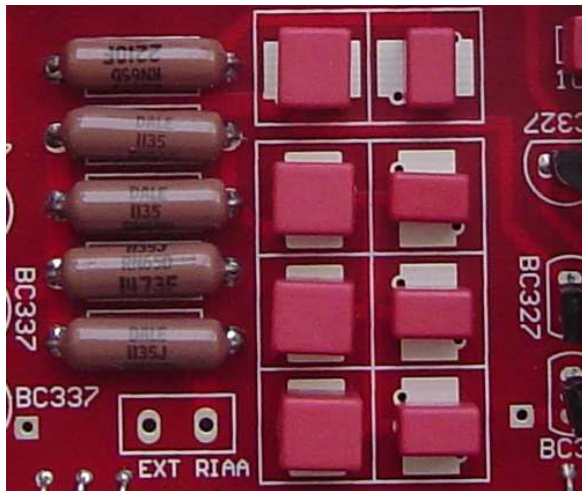
Power resistor mounting

In the RIAA section, a wider spacing is provided to make room for Dale RN65 resistors, which appear to be a good choice. The resistor values are based on the original values as given in the diyaudio thread (9.1k resp. 73.5k), e.g. by using two available values in parallel, but you are of course free to implement differently. It is recommended to measure and match the resistors between the two boards for stereo, to make sure you have the same gain and frequency curve for both channels, which will improve imaging.

The footprint for the capacitors in the RIAA section will accept various capacitors, including LCR caps, Rifa SMD caps, and Wima caps (5mm pin spacing). Enough footprints are provided in parallel, so that you can put caps in parallel to achieve the desired values. Again, the value given in the schematic is the total value, so in order to have the correct value (33.3nF resp. 11nF) you should add up as many caps as needed, the footprints are all connected in parallel. To have some spare 100pF and 220pF caps probably is a good idea. Again, measuring and matching the values between the two channels is a good idea too. The following picture illustrates once more how the footprints add up to the desired values.



Use the connector to put an external RIAA network e.g. if you want to make it variable (in that case, no RIAA components on the PCB of course), but provide shielding of the network connected to the input GND connector please.



RIAA section with connector for external RIAA network

One resistor is labelled “Neumann” (upper left corner in above picture), this refers to the theory where in the process of cutting the record the high frequency content was supposedly filtered, so that the cutter head would not burn with high frequency content (remember the records are cut with inverse-RIAA, that rises to high frequency). Put either 220 Ohm (for implementing that additional time constant) or a wire bridge.

The values given above and also in the schematic will give you the groovy Joachim-Gerhard –Michael-Borresen house curve. Alternatives include:

- Ricardo Cruz: 11nF, 32,1nF 9.9K and 73.7K
- Salas: 12nF, 34.7nF, 9.1k, 67.8k

To assemble SMD caps, first put some solder on one side of the footprint (but not the other), then put the cap in place and heat up the solder, so that the device is fixed on that side. You can now solder the other side, and reheat the first side just to be sure there is no mechanical tension left in the component.



Step 1



Step 2



Step 3

The power devices are best put on the heatsink first, remember the sequence as indicated on the PCB. Place the device in position (remember to put isolation if needed), then put the clip on top and press in. It does not have to be enormously precise, as the position of the heatsink on the PCB is not critical. Then, slip in the heatsink with all devices into their mounting holes on the PCB (that step may require a little wiggling of the devices back and forth), until you can press them all in nicely and the heatsink touches the PCB. Then, fix the heatsink with the screws (center screw first) before soldering the power devices.



Alignment of the devices



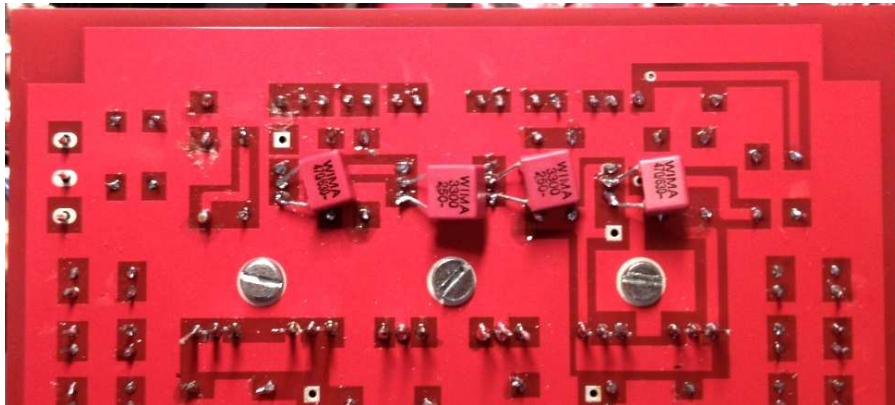
Mounting the power transistors

Powering up

At this point, the jumper in the middle of the PCB, between the shunt regulator section and the amplifier section, still should not be bridged. You will need the external power supply, or a lab supply with two outputs, and a multimeter (DMM).

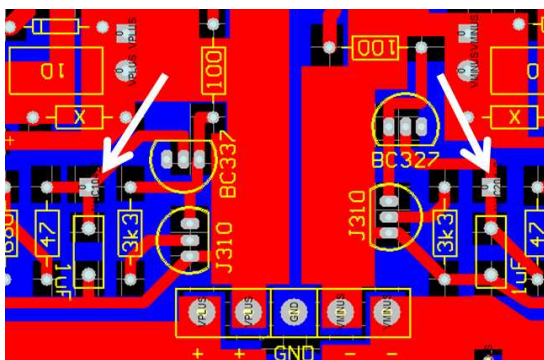
Best way to powering it up is to use a regulated lab power supply with current limit (set to 200mA), both the positive and negative voltage tracking to each other. Turn up to +/-30V and see what happens. Alternatively, you can use a regular power supply with +/- 28V to 35V output, in which case a couple "safety" resistors (100 Ohm/10W) should be put in series with the rails. If the voltages come up fine (or, the lab power supply does not run into current limit), you can remove the resistors.

then the capacitors that are at the output voltages (C105 / C205, as well as the ceramic blocking caps in the amplifier) and the shunt regulator (which is not limited in its performance by these changes, as Frans pointed out) will have to work just a touch harder. As for the caps, a good foil cap will do (e.g. Wima MKS, FKP or similar). The following picture will show how that can be implemented on the PCB. The transistors to look for are under the heatsink.

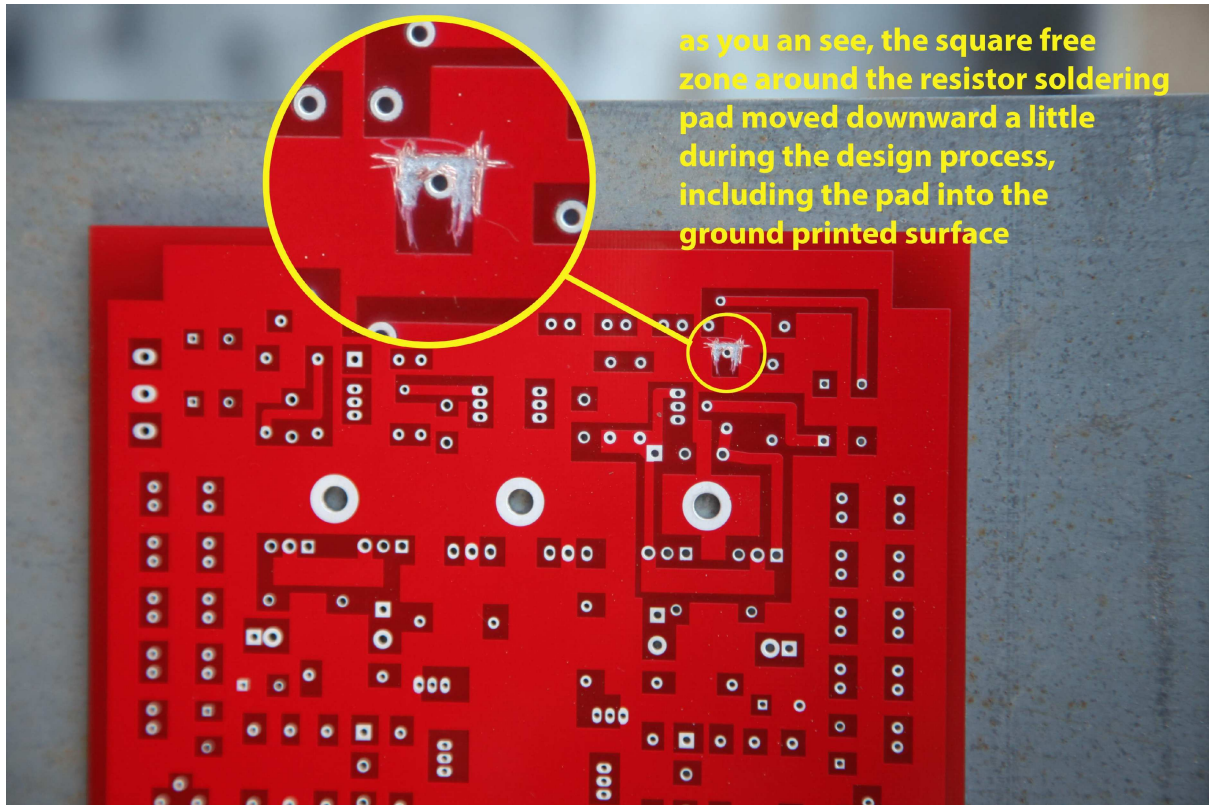


The two outer caps are the 470pF caps, the two inner caps are the two 2.7nF caps. You will notice on the picture that there are 3.3nF caps – that is simply because that is what I had on hand. The PCB version R31 has these capacitors included in the layout on the topside.

LEDs all shine but output voltage is high or very low – Shunt regulator does not work, check the two test points as per the picture below. These should be at +/- 0.6V to GND. If not, check the transistors (also in the picture).

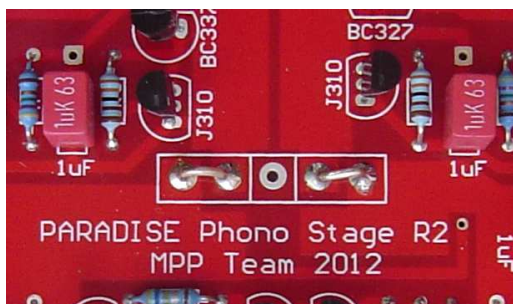


If that's OK, measure the Gate-Source voltage of the PMOS/NMOS, they should be around +/- 2.5V to GND. If not, check the power transistors. MOSFETs are rather sensitive to handling, please make sure you do not expose them to electrostatic charges. You can measure the resistance from Gate to Source (should be very high), and use the "diode testing" function between Drain and Source (should be around 0.65V in one direction, high-ohmic in the other direction).



What is next?

Now the regulators should be working fine and stable. Next, turn the power off, and connect the main amplifier section to the regulators, as per below picture.



Don't worry if your PCB does not look like this one, the picture is taken from the R2 version but the jumpering is the same. Put in good-sized wires and make sure you have a good solder joint on both sides of the PCB.

Put a resistor of 10KOhm across the input of the phono stage. Now you can turn on the power again. The LEDs should all be shining. Check the supply voltages, they should be at +/- 18V. Then check the output voltage, should be at 0V or very close.

You should be up and running at this point ;-)

If that is the case, measure the voltage at the input connector and adjust the “offset” trimmer so that the input voltage of the amplifier goes to zero. This voltage will be very small, so it may be difficult to measure. You can start doing this with open input (just the DMM), and try to adjust it down to <100mV or so, this will be a little tricky but doable. Then, put a 10KOhm resistor in parallel to the input and adjust it down to 0V. This is a very important adjustment for the sound, hugely impacts clarity and transparency!!

At the same time, the output voltage of the servo opamp (pin 6 of the IC) should go to zero – if not, there is some asymmetry most probably in the input stage. If the servo output voltage is lowish (a couple of volts, either polarity) that doesn't necessarily mean there is a problem. If it is close to the rail voltage of the opamp, the asymmetry is pretty strong and you should recheck the transistors. Recheck after 15min when the amplifier has warmed up. You may have to increase the resistor from input to GND to actually see something. Some cases have been mentioned where the electrolytic caps in the input stage may take a couple days of being powered up to get rid of excessive leakage current, which may cause imbalance and output voltage fluctuations.

As subsequent experience shows, 10kohm is quite high, as it will see leakage current variations of the elcaps and the BJTs. Once the input offset is adjusted, try putting 10Ohm across the input, and check what the output does. It should be fine, meaning, no more significant voltage fluctuations. In fact, while the elcaps are "forming" (and that can take a couple weeks of constant operation), you may continue to see output voltage fluctuations, and it may pay off to re-adjust the input offset after that time. No need to worry, though.

The servo having a large voltage (more than +/-2...3V) on its output (pin 6 of the opamp) means it has to compensate a certain offset of the input stage. Remember the servo's impact is pretty weak, and deliberately so, so that voltage can reach several volts both ways. It should stay within +/-12 V though, in order to not leave the linear output range of that opamp. 5V is therefore OK. Best would be 0V, but that only happens with perfectly matched input stage, current mirrors, and output buffer with no offset.

While the mirrors's design is very clever so their impact on offset is negligible, the other two sources of offset may be the cause. On the output buffer, if the JFET Q99 has a high IDSS, the voltage on its gate will have to be negative so it operates at the current dictated by Q91+R15. So in order to have zero output offset, the voltage at its gate will have to be negative, so the servo must steer it that way. In my case, the voltage there is -1.5V (as documented in the annotated schematic in the assembly guide). Now, you could run the output buffer at higher current, by reducing R15, and thereby reducing this offset, but that's not needed for performance or sound.

On the input stage, if the current gains are not matched well enough between the NPNs and the PNPs, you may see that type of offset as well. In fact, it should be different for different values of input load resistance (you would probably have to look at values of 10K or higher to see something). It may also be that there is one transistor that exhibits base-collector leakage current, this is VERY rare but sometimes individual transistors escape the test program at manufacturing. You may want to try input resistors of 10K and 20K just to see if the offset changes. If it doesn't then just live with it, you should be fine. If it does, you may want to check if you can come up with another set of input stage transistors.

What can go wrong?

One or both supply voltages stuck at zero – this probably indicates a shortcut. Check your soldering, especially around the SMD caps, to make sure nothing connects the wrong way.

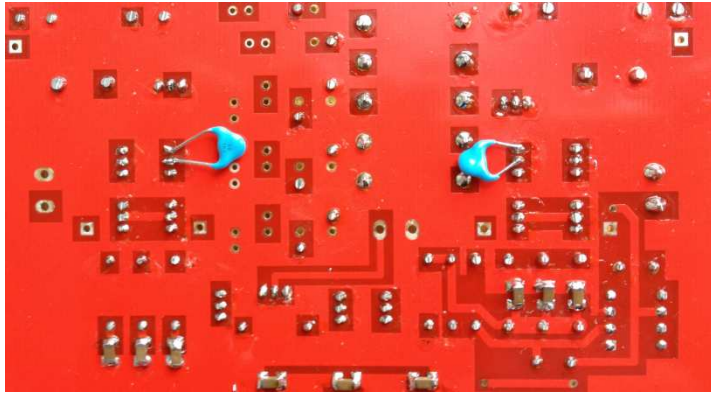
One or both supply voltages stuck at 2-3V – this probably indicates that you put a BC327 in place of a BC337 or vice versa. Check your transistors to make sure they are all correct.

Output voltage stuck at plus or minus 15-18V – check the transistors in the output buffer. Measure the voltage across the "Ext RIAA" connector, it should be zero or very close. If not, one of the current sources is not working correctly, check those.

If there is a voltage across the "Ext RIAA" connector, and the output voltage of the amplifier is (more or less) the same, then the output buffer is working fine. If the output voltage is dramatically different, then the output buffer has a problem. Some DC across the "Ext RIAA" connector really is no problem, it indicates the DC offset of the output buffer that the servo is taking care of. Ideally, it should be zero. I had -1.5V there, and it still sounded (mighty) fine.

Output voltage stuck at some in-between level – Check the output voltage of the servo opamp (pin 6). If this is at +/-18V, the servo is trying to fix the offset but cannot. This indicates that there is more asymmetry in the circuit than the servo is set up to handle. You may consider reducing R43a and R43b, to give the servo more impact. But the real problem probably is a mismatch in current gain between the PNPs and NPNs.

High frequency oscillations (70MHz...100MHz) – In extremely rare cases (one verified so far), the amplifier can oscillate at really high frequencies. This is probably due to the physical size of the amplifier, the parasitic inductances, and transistors with high current gain in the mirrors. The way to fix this is to introduce two capacitors in parallel to the Collector and Base of Q18 and Q7.



This picture shows where the compensation caps should go, this is right under the RIAA capacitor / resistor area. The value of the capacitors needs to be tested, 33pF / 47pF / 68pF should be fine.

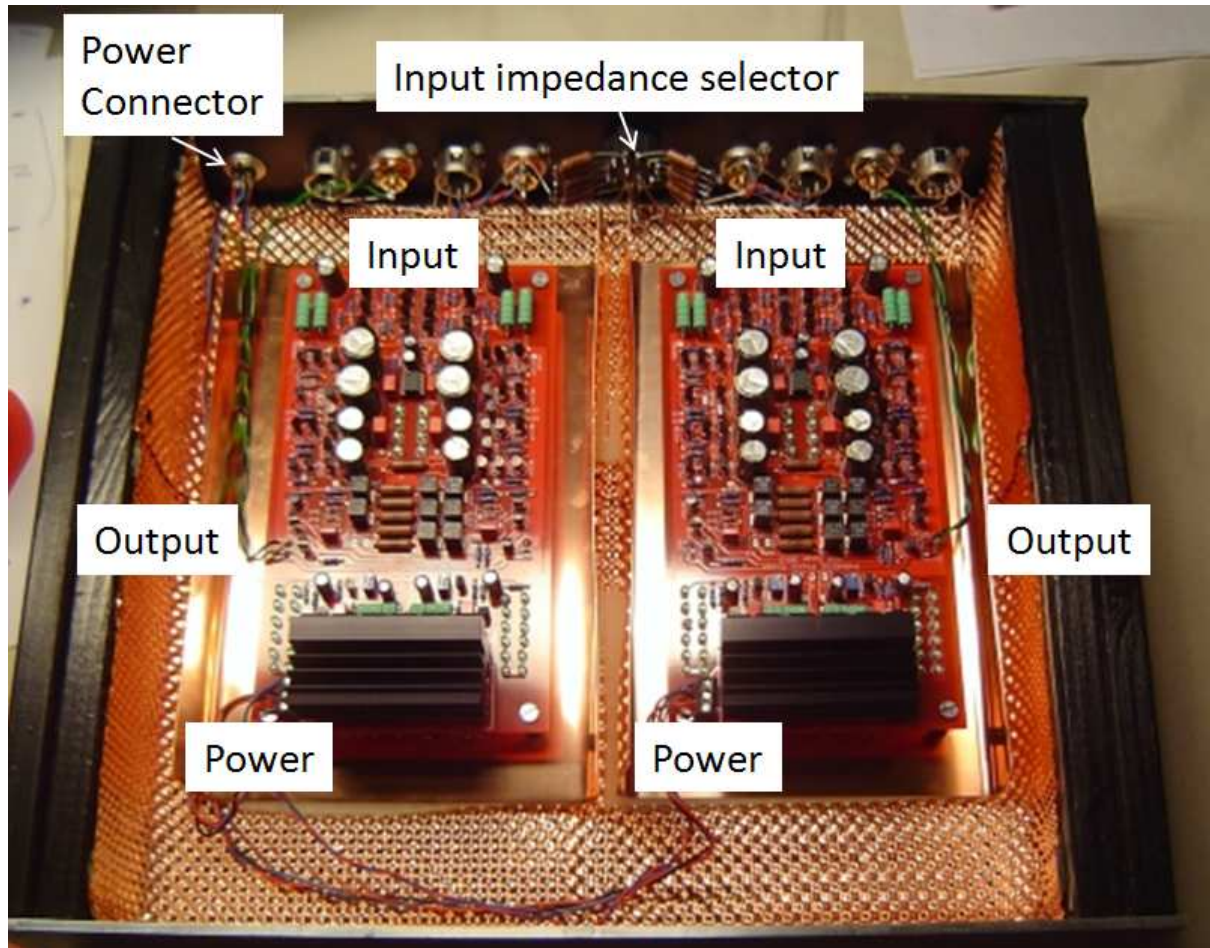
For everything else, a thread has been started under this address:

<http://www.diyaudio.com/forums/analogue-source/218625-paradise-builders.html>

There is a lot of additional information available there, and questions will be answered.

Installation

The PCB has output connectors on both sides, so that the wiring of the output connectors does not have to go over the PCB and potentially cause crosstalk. The picture below illustrates how I did it, worked fine for me:

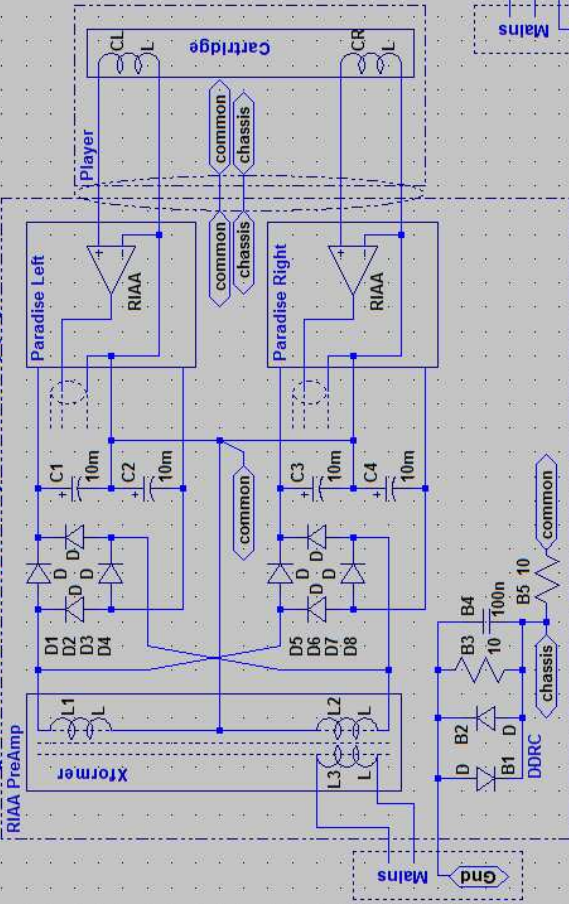


The power connections are at the far corners from the inputs, and can be routed around the PCBs to the power connector, again to not cause any crosstalk.

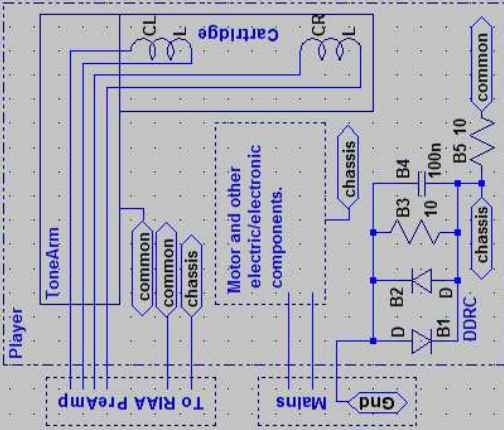
In the picture above, the grounding scheme is two-fold. The case is clad with copper foil, and the two boards are sitting in copper "tubs" that are isolated from the copper foil. A star ground connection is implemented in the middle at the upper edge of the picture, and the copper foil, the copper tubs, and the ground connections from the PCB input connectors are all centrally connected there. I have used a 4mm banana socket for the grounding cable of the turntable, which serves as the star ground and the grounding of the case backplane. The input connectors are right next to this, then the output connectors, all in symmetrical fashion. The power supply ground is NOT connected there, but routed through the PCBs. The power cables are routed outside of the copper "tubs".

In the “MPP” thread on diyaudio, Frans has contributed this schematic that shows in great detail how to achieve the best noise isolation for the phono stage, and how to wire it all up:

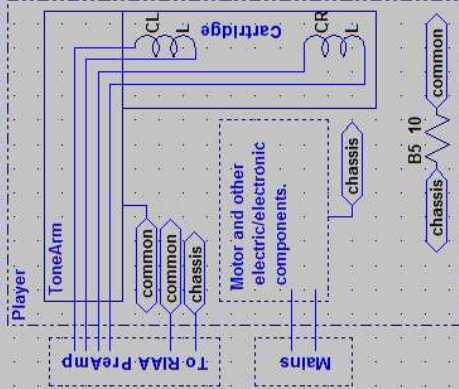
Using a one transformer power supply.



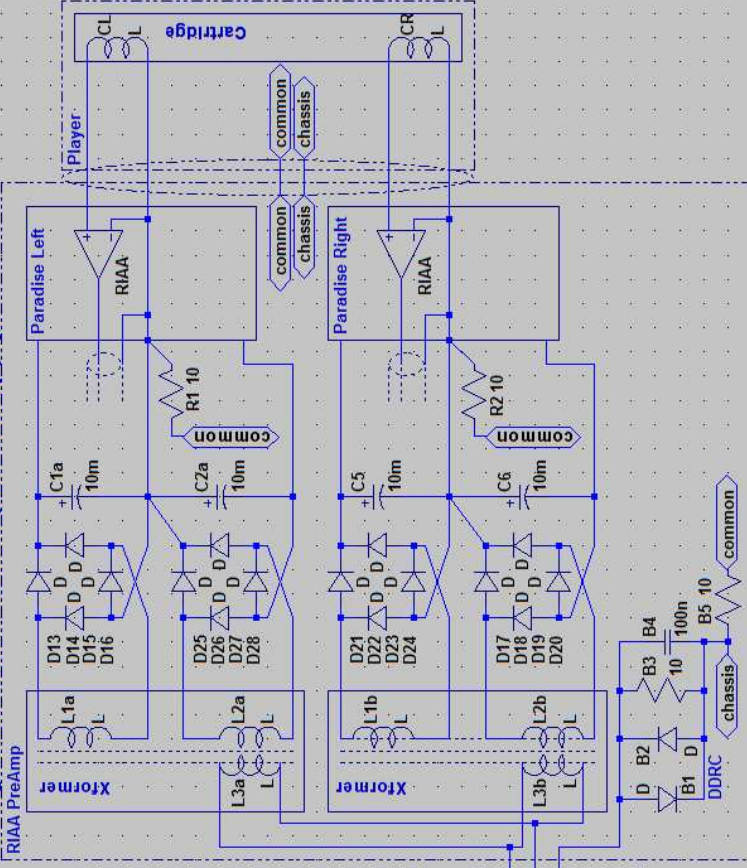
3 Wire mains connection



2 Wire mains connection



Using a two transformer power supply.

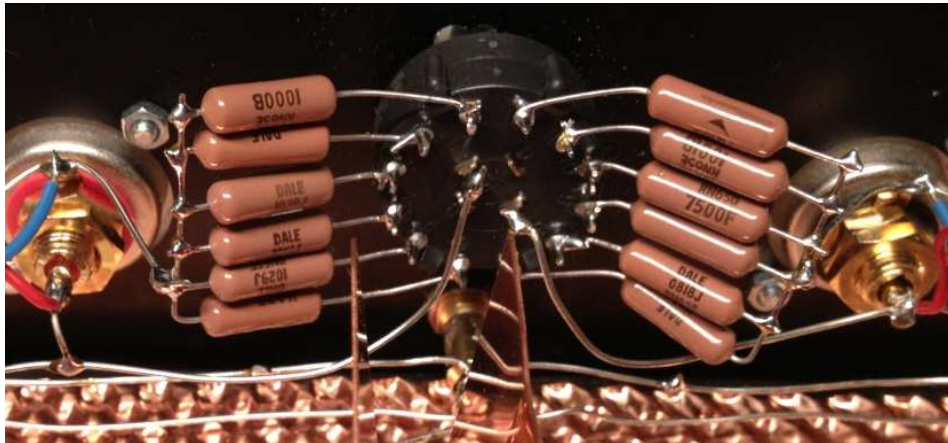


If the record player has a mains ground wire connected to it, then it should be connected to the player <chassis> using a DDR-Circuit. Make a 6-wire connection to the RIAA as drawn. You have to install it yourself, commercial players will not include the DDR-Circuit. If the record player has a two wire mains connection then use a 10 Ohm resistor to connect the tone arm to the <chassis> and use the 6-wire connection as drawn.

The tone arm should be connected to <common> and not to <chassis>. Using the DDR-Circuit is safe and breaks any mains-ground-loop that may exist, all equipment in the system should be <chassis> connected, but only if a DDR-Circuit is installed. Resistors R1 and R2 break galvanic ground connection between the left and right RIAA power supplies, this makes that the only galvanic (0 Ohm) connection between those is at the preamplifier. This prevents any ground loop!

For resistor B5 it may be beneficial to select a wire wound resistor, the small inductance that is has may prevent high frequency noise to enter <common> (from <chassis>). This needs more testing and/or discussion.

The input impedance selector switch can be realised like in the following picture:



The switch in the middle is a rotary switch with 6 positions, 2 contacts. The resistors here are Dale RN65 (10 / 100 / 220 / 470 / 1000 / 2200 Ohm), and the input connector is connected in parallel to the rotary switch and the input at the PCB, while the other ends of the resistors go to GND. All the GND lines are fed back to the star ground which is visible just below the switch.

Another way of doing this would be to use little DIP switches and solder resistors and capacitors to these. For the resistors, similar values should be fine. For the capacitors, some selection up to e.g. 4nF should be appropriate, but for most systems that may not be required. The best way to find out is to use a test record that allows you to measure the frequency response of the cartridge. Here, you may see a peak at 10...20kHz, which the resistor and capacitor are supposed to control. Underdamping it will add some brightness to the sound, whereas overdamping will make it sound dull. Typically the cartridge datasheet will give some recommendations, which are a good starting point. Please remember that phono cables will add between 50pF...150pF to the input capacitance.