

Paradise R3 Assembly and troubleshooting guide

01.09.2012

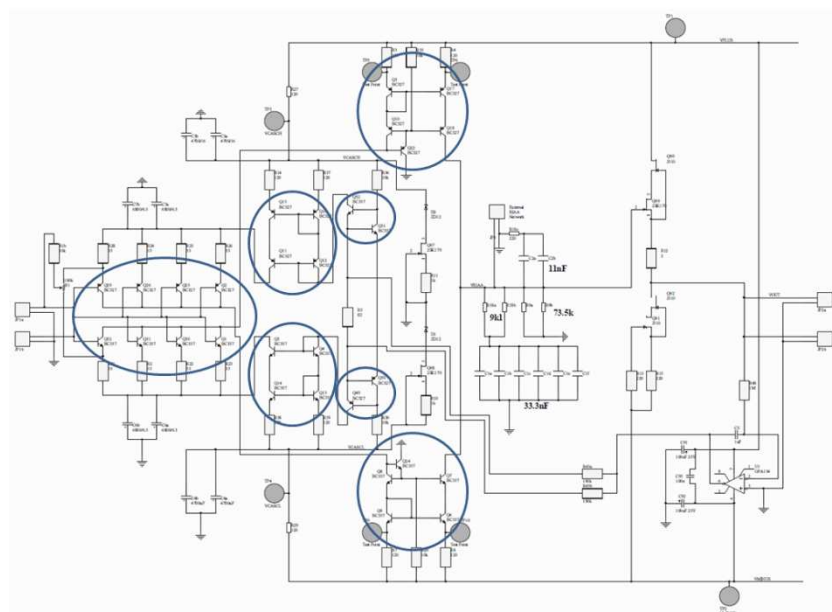
Welcome to the Paradise R3 phono stage, which arguably is one of the finer phono preamplifiers out there. The circuit concept is from Joachim Gerhard, circuit design and simulation from Michael Borresen, the power supplies from Frans de Wit, and the layout from Alfred Hesener. This assembly and troubleshooting 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.

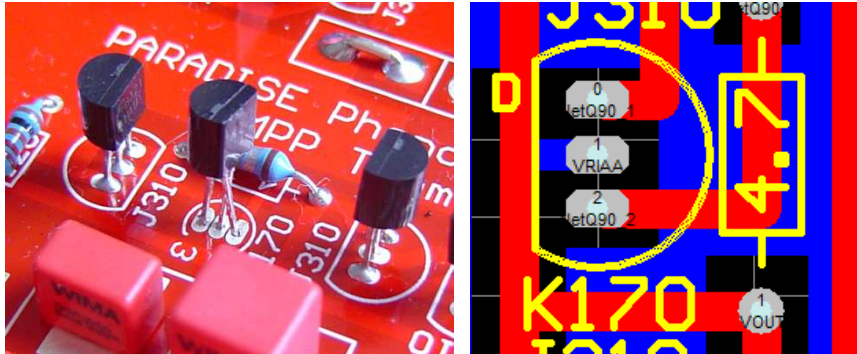
Do not put wires into the power supply jumpers (between the shunt regulator and the amplifier section) at this point.

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.



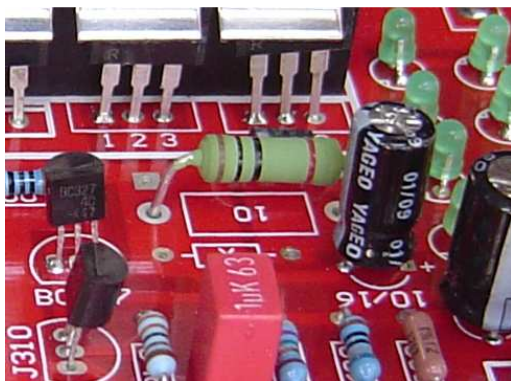
This picture has the groups of transistors circled that should be grouped as good as possible, again with the 8 input transistors being the most critical.

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).

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.

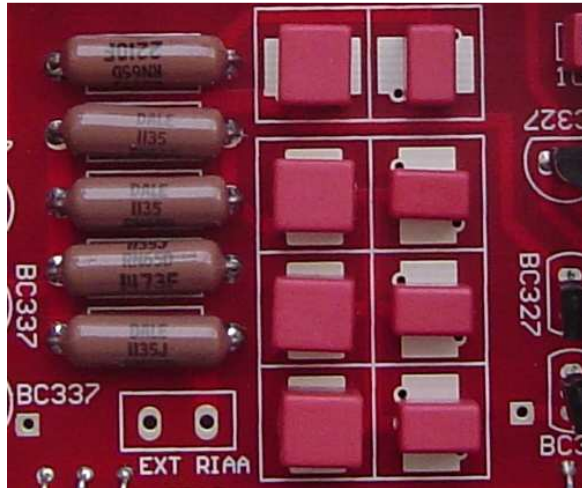


Power resistor mounting

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, 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. 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. 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.

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.

Next, check the output voltages, they should be around $\pm 18\text{V}$ and adjust them using the trimmers for each polarity. The trim range is about $\pm 0.5\text{V}$. Recheck the voltages after 15min or so, when the regulator is warmed up.

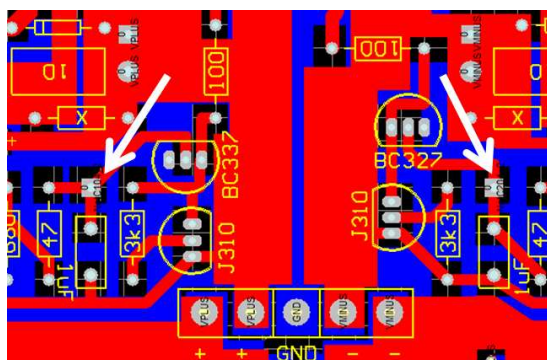
What can go wrong?

Input voltage clamped at 18V-20V, high current – Check if the power diodes are inserted correctly. Check the LEDs behind the heatsink. Check the Collector voltage of the outer NPN/PNP transistors (edge of the heatsink), these should be ~3V higher than the output voltage (+/-21V). Check the 10 Ohm power resistor for correct value. Check the power transistors for shorts.

LEDs do not shine – Most probably one of the LEDs is inserted the wrong way, recheck the polarities. If they are all OK, measure the voltage across them, on a good green LED this should be 1.8V. High voltage indicates the LED is broken. Make sure the LEDs behind the heatsink are also fully on.

Voltage is lower than 18V but can be adjusted – One of the LEDs is probably shortened

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).

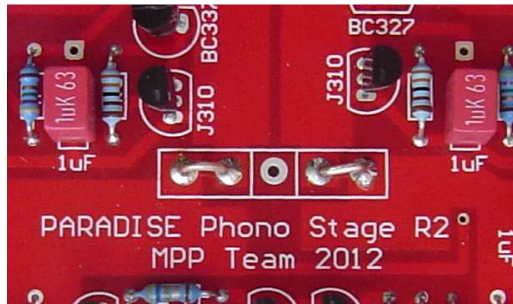


If that's OK, measure the Gate-Source voltage of the PMOS/NMOS, they should be around $\pm 2.5V$ to GND. If not, check the power transistors. MOSFETs are rather sensitive to handling, please make sure you do not expose them to electrostatic charges. You can measure the resistance from Gate to Source (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).

LEDs shine but output voltage wanders around, strange readings – You may experience oscillations. With this shunt regulator design, that is quite unlikely. If you have an oscilloscope, connect it to the output voltage (AC coupling), and try to find the oscillations. If that is confirmed, try and decrease the 1uF caps (see picture above).

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 output voltage of the servo opamp (pin 6 of the IC) goes to zero. Recheck after 15min when the amplifier has warmed up. You may have to increase the resistor from input to GND to actually see something.

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.

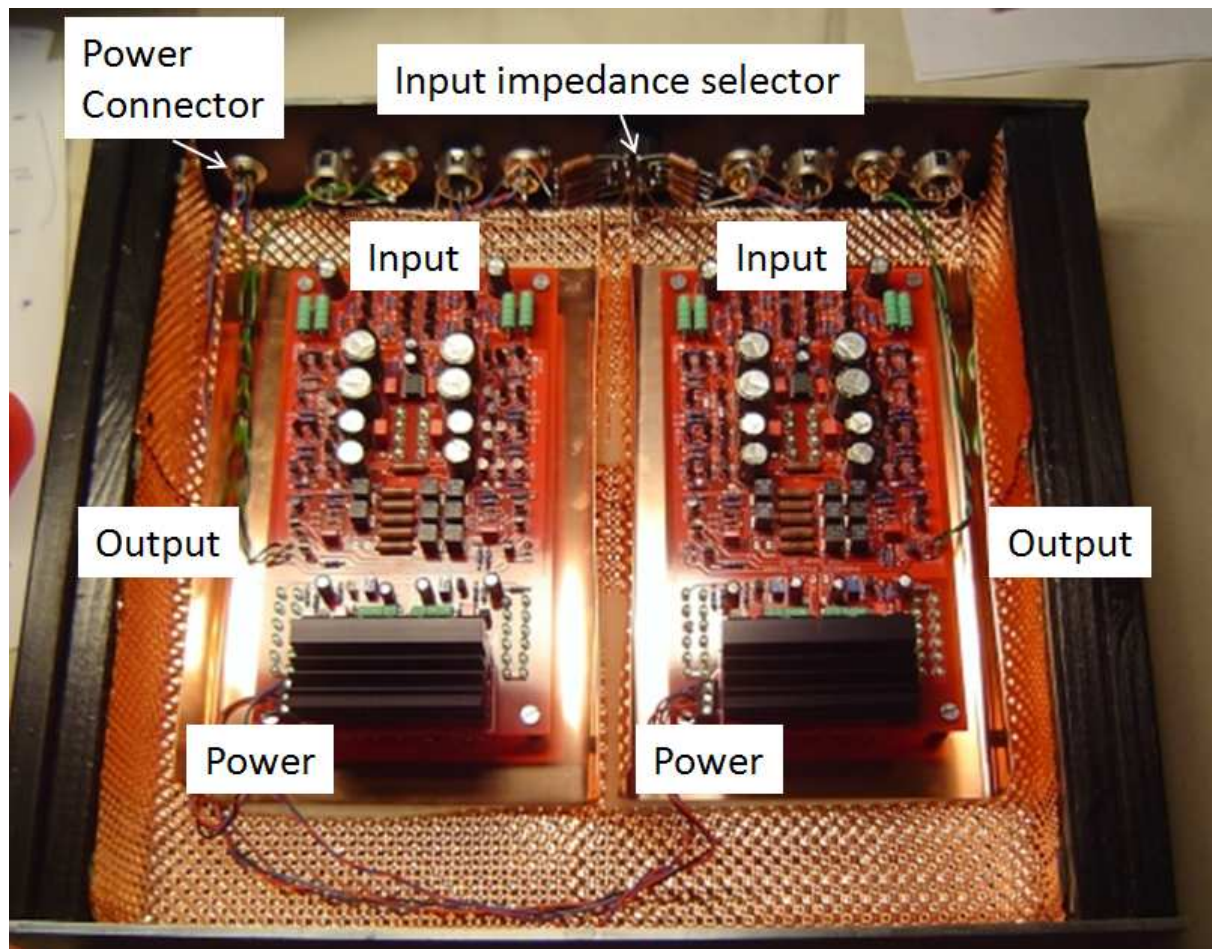
Output voltage stuck at plus or minus 15-18V – check the transistors in the output buffer. Measure the voltage across the "Ext RIAA" connector, it should be zero or very close. If not, one of the current sources is not working correctly, check those.

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.

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.