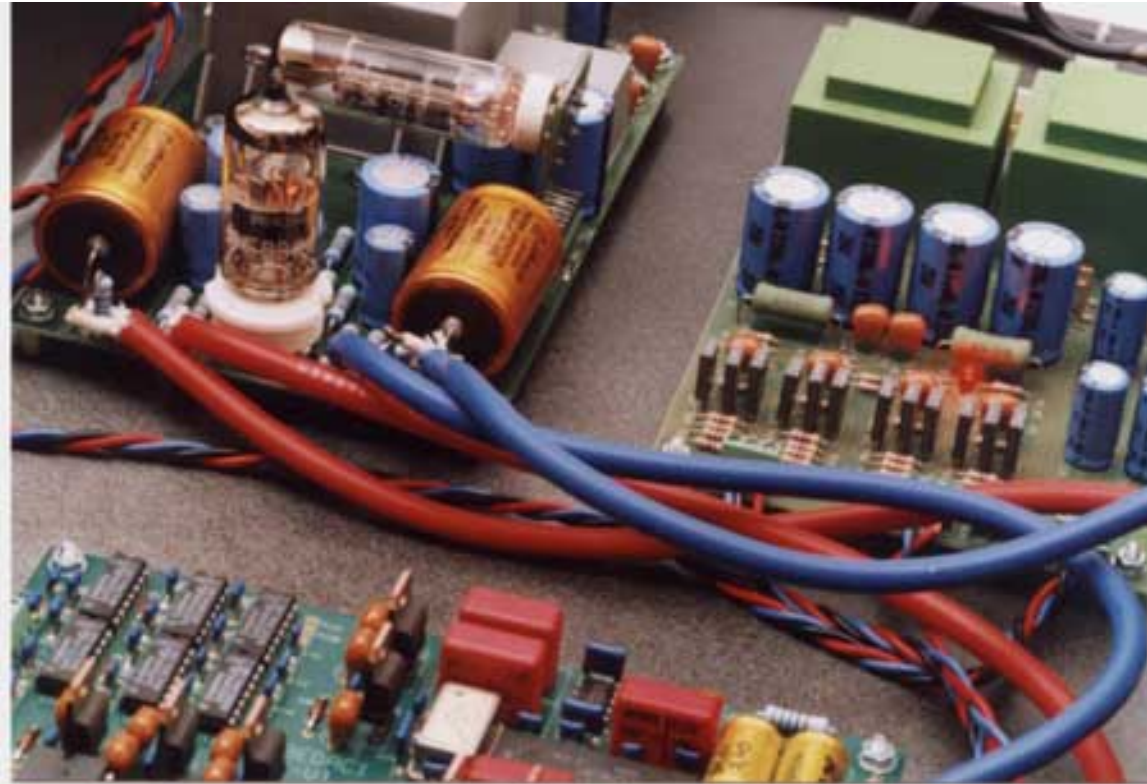


DAC assembly guide

v. 2.0



03-09-2000

by

Jos van Eijndhoven
Marc Heijligers
Guido Tent
Korneel Wijnands

General guidelines

Introduction

Building the DAC takes 4 main steps:

- Building and testing the analogue supply PCB.
- Building and testing the tube PCB.
- Building and testing the digital PCB.
- Assemble the PCBs, and build them in a cabinet.

This manual will explain all these steps in detail. This chapter first provides some introductory information about building electronics. We do emphasise to read all text, and to follow all instruction carefully.

Tools

Use appropriate tools, e.g. a tweezers, a cutter, a proper soldering iron (you may borrow one), a resistor bow bridge to bow the terminals of components, etcetera. Buy proper solder like Biliton or what is locally supplied. Make sure there is "harskern" inside. To unsolder components, litze may be helpful, but not strictly necessary. A multimeter finally is needed in order to check some voltages for correct function of the PCBs.

Soldering

NEVER use solder paste, solder water or whatever that might "help" you. Make appropriate junctions with not too much solder.

Component guide

Appendix 1 contains drawings of most components. These drawings contain:

- The symbol as used in the circuit.
- The symbol as printed on the PCB.
- The component as it looks in real life.

Construction order

Start building the PCB with the smallest components first. In that case it is much easier to turn around the PCB for soldering. A feasible order is given in this guideline.

Component attaching

Attach the components as close to the PCB as possible, unless otherwise stated.

Construct the PCB carefully

Attach and solder only one component at a time. It's very easy to forget to solder a component.

Bowing

When bowing the terminals of components, make sure the text at the component will stay visible once attached to the board. This makes verifying more easily!

Directional components

Some components are directional. It is VERY important to connect these components in the right direction! If not connected appropriate, damage may occur.

Don't hurry, be happy

Don't build in a hassle. Better spend a couple of hours (over a few days), with ample of time of checking, instead of having to measure, detect errors, replace defective components etc. This takes you more time after all.

Building instructions

- Check, check, and check everything you do. One feasible way could be:
- Select the component; make sure it is the right one.
- Tick it in the component list.
- Find it in the schematics, and circle it.
- Bow it (if needed), and make sure the text is visible when connected to the PCB.
- Make sure in which direction it must be attached to the PCB.
- Attach it; re-check that the place at the PCB is the same as in the schematic and in this building manual. Attach it short by pressing it close to the PCB.
- Recheck the direction!
- Cut the terminals. Cut them short, but leave about 1 mm to solder, no more.
- Solder. Both the terminal and the 'island' must be heated, and the solder should fluid among both places.
- Check again that it is connected at the right position.
- Re-find the symbol in the schematic, and tick it. This should be the circled component (of-course).
- Photos are added as a guide. It is not possible to build the PCB from these photos, so please refer to the text for exact guidelines. Use the photo only to check when you are uncertain about something

Warning

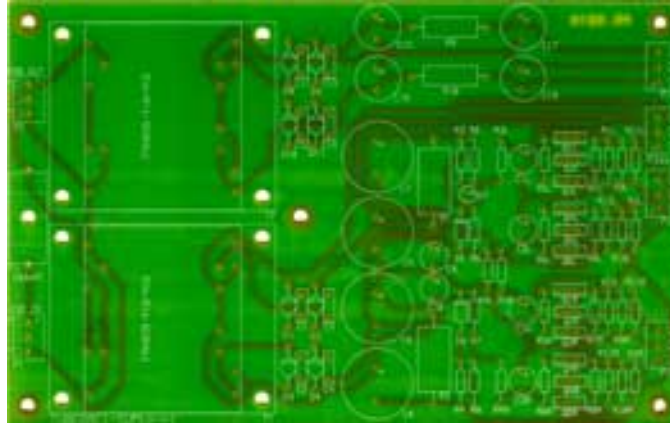
We are NOT responsible for damage caused by the builder, especially when he makes errors during construction. We do our best to prevent errors from this building manual, but there could be an error left inside. If you find such an error, please contact us as soon as possible. In that case, we can inform all the other builders.

The analogue power supply (SUPLV)

First phase (check)

Sort out all the components, and make sure you recognise them all. Check the values and number of all components.

The PCB looks as follows (look at the enclosed photo for more details):



The components look as follows (look at the enclosed photo for more details):



If components are missing (or you have too many components, and hence someone else has missing components), call your contact person. Make sure you understand the building guidelines and the schematic. Read them in total, before you start building. Correcting errors afterwards will cause much frustration and needs much more time! Ask your contact person when something is not clear!

Second phase (building)

Connect the components in the order below, and make sure you understand the guidelines. Take the main PCB and you may start building!

- **Diodes 1N5819 (D1, D2, D3, D4, D8, D9, D10, D11) DIRECTIONAL!**

Watch the direction (the bar at the component should be directed such that it matches the direction of the bar denoted on the PCB). Solder the diodes one for one! The terminals are rather thick, so you should use a pair of tweezers to pull the terminals CAREFULLY. Solder, and check, check, and check!

- **Zener 8V2 (D5, D6) DIRECTIONAL!**

Also take care that they are connected in the right direction, just like the diodes.

- **Small resistors (all R, except R1, R2, R9 and R10)**

Connect them all in the same direction (tolerance ring below, gold), so it is easier to read the values when they are connected. Connect them one by one in the following order:

390 Ω	(R5)	orange white brown gold
560 Ω	(R8)	green blue brown gold
820 Ω	(R3)	grey red brown gold
1k5	(R4)	brown green red gold
6k8	(R6, R7)	blue grey red gold
10k	(R3R, R4R, R3L, R4L)	brown black orange gold
47 Ω	(R1L, R1R, R2L, R2R) (NOT R11L, R11R, R12R, R12L!!!)	yellow purple black gold
100 Ω	(R5L, R5R, R6L, R6R)	brown black brown gold
470 Ω	(R7L, R9L, R11L, R7R, R9R, R11R, R8L, R10L, R12L, R8R, R10R, R12R)	yellow purple brown gold

- **Big resistors**

Same comment as in 3. Remind to keep the text visible when bowing and mounting.

4.7 Ω	(R9)	yellow purple black silver brown
10 Ω /1W	(R10)	brown black black gold brown
100 Ω /5W	(R1, R2)	no colour code

Mount resistor R1 and R2 1 cm above the PCB, because they get a little bit warm.

- **LED (D 7)**

The long terminal is the positive side, so watch the direction. Look out with soldering that it doesn't make a shortcut with the surrounding terminals (these are closely)! Try to solder fast.

- **Connectors (P1, P2, P1D, P1L, P1R, P3)**

Make sure that the wire inlets are directed to the outer side of the PCB.

- **Capacitors**

470pF	(C1, C2, C3, C4, C11, C12, C13, C14).	n47, orange stripes, some are round and brown.
-------	---------------------------------------	--

- **Solid aluminium capacitor, DIRECTIONAL!!!**

Watch the direction!!! A small '+' is denoted on the capacitor, and on the top a black bar is drawn (the '-' side). After soldering, the '-' side of all elcos should be in the same direction

10uF/25V (C9, C10, C1L, C2L,
C1R, C2R) orange coloured.

- **Transistors (watch the difference between the 139 and 140!!!)**

BD139 (Q1L, Q2L, Q3L, Q1R, Q2R, Q3R)

BD140 (Q4L, Q5L, Q6L, Q4R, Q5R, Q6R)

Connect such that the backside of the transistor matches the dashed line on the PCB.

After soldering the text on the transistors is difficult to read. Looking at the transistors after soldering gives a four times 'back front back' view of them.

- **Fuseholder (F1)**

Immediately attach the fuse itself after soldering the holder, so you don't lose it afterwards.

- **Electrolytic capacitor (Elco). DIRECTIONAL!!!**

Watch the direction!!!

1000uF (C15, C16, C17, C18)

2200uF (C5, C6, C7, C8)

After soldering, the '-' side of all elcos should direct to the same side.

- **Transformers. DIRECTIONAL!!!**

Watch the difference between T1 and T2. Carefully mount them (it's a bit puzzling to get the terminals connected, but they only fit one way), press to the PCB and solder

T1 18/18V

T2 8/8V

Check afterwards if all junctions are soldered at the backside. Again, make sure all components are at the right place.

Third phase (testing)

Now all is finished, it's testing time ! This is the last opportunity to check the PCB and the component placing. It may be a good idea to ask someone else, not involved, to have a look as well.

The PCB should look as follows (look at the enclosed photo for more details):



The test of the PCB is not too difficult.

- Connect the multimeter across the output terminal P1R, pin 1 and 3, denoted by the + and – signs on the PCB. Set the meter in the range "DC", and select a voltage in-between some 15 V to 50 V.
- Take care that the PCB is on an insulating surface. Check if no metal parts or solder rests are beneath.
- Connect a suitable mains lead to the power input, connector P1, pins 1 and 3, denoted by the ~ signs on the PCB.
- Take care that the test pins do not make a short circuit when measuring !

Warning

Open mains voltage is at the circuit. Take care that no-one (included animals) can touch the circuit under the circumstances of testing. We are NOT responsible for damage caused by the builder, especially when he made errors during construction or testing.

- Plug in the lead. At first watch the LED at the PCB. This should start burning immediately. Then watch the readout of the multimeter. This should read some 9 V initially, slowly rising to 10 V.
- If the LED does not work, or any other value appears on the meter, pull out the mains plug immediately !
- If the reading is correct, disconnect the mains, and connect the "-" connection of the multimeter to pin 2 of connector P1D (denoted o at the PCB), and release the "+" connection of the meter (which also was at P1R).
- Now plug in the mains again, and measure the following points:

Point	should read	tolerance
P1D, pin 3(+).	+ 13 V	± 1 V
P1D, pin 1(-).	- 13 V	± 1 V

- If these readings are correct, disconnect the mains again, and connect the "-" connection of the multimeter to pin 2 (denoted 0) of connector P1R, and release the "+" connection of the meter.

- Now plug in the mains again, and measure the following points:

Point	should read	tolerance
P1R, pin 3(+).	+ 5 V	$\pm 0,2$ V
P1R, pin 1(-).	- 5 V	$\pm 0,2$ V
P1L, pin 3(+).	+ 5 V	$\pm 0,2$ V
P1L, pin 1(-).	- 5 V	$\pm 0,2$ V
P3, pin 3(+).	+ 24 V	± 2 V
P3, pin 1(-).	- 24 V	± 2 V

- If all voltages are OK, the measurement under point 9 may be repeated after the supply has been switched on of an hour to stabilise the temperature effects. The voltages shall not exceed 5,2 V positive as well as negative. All other tolerances are due to components and mains variations and are acceptable.

Congratulations, you now have finished your first DAC-PCB.

The tube output stage (SUPHV)

First phase (check)

Sort out all the components, and make sure you recognise them all. Check the values and number of all components.

The PCB looks as follows (look at the enclosed photo for more details):



The components look as follows (look at the enclosed photo for more details):



If components are missing (or you have too many components, and hence someone else has missing components), call your contact person. Make sure you understand the building guidelines and the schematic. Read them in total, before you start building. Correcting errors afterwards will cause much frustration and needs much more time! Ask your contact person when something is not clear!!!

Second phase (building)

Connect the components in the order stated below, and make sure you understand the guidelines.

1. Transformer (T1)

First of all, to get the big transformer T1 into its holes is a bit puzzling. Therefore it is first a good idea to carefully fit, and eventually bend the terminals slightly. Watch out for scratching pads and tracks on the PCB with the terminals, and don't put too much force on the transformer when pushing it inside the holes! Only one orientation is possible (one side of the transformer has less pins than the other one). Eventually use a tweezers to carefully bend the terminals.

Once all terminals fit in their holes, the transformer and the main PCB may be laid upside-down on the table. Now all terminals (10) appearing through the holes shall be visible. Press the main PCB downwards such that it is as close as possible to the transformer. Now attach screws and nuts (4 times M4 screw and nut). The nuts shall be at the solder side. Again, do not use excessive force to mount them. Since de-soldering is a terrible job, again inspect so turn the PCB around and watch if the transformers is mounted correctly. Finally all terminals can be soldered.

2. Small resistors, 1/4 Watt (R1, R3, R4)

Connect them all in the same direction, so it is easier to read the values once they are connected. Connect them one by one in the following order (the colour codes on these resistors are not very clear):

887 ohm	(R1, R4)	grey	grey	purple	black	brown
221 ohm	(R3)	red	red	brown	black	brown

3. Diodes 1N5819 (D1-D5) DIRECTIONAL!!!

By mistake these diodes are designated 1N5817 or 1N5818 in the circuit diagram and components list. Diode 1N5819 however is fully equivalent and shall be used here. Connect those in the right orientation. The terminals of the diode just fit in the holes, so you might need to pull a bit with some pliers (carefully!!!). Because the terminals of the diodes are relatively thick, soldering needs more heat (time) than usual.

4. Somewhat bigger resistors, 1 Watt (R2, R10, R20, R11, R12, R21, R22, R13, R23, R14, R24, R15, R25)

220 k	(R2, R14, R24)	red	red	black	orange	brown
4k7	(R10, R20)	yellow	purple	black	brown	brown
47k	(R11, R12, R21, R22)	yellow	purple	black	red	brown
330 ohm	(R13, R23)	orange	orange	black	black	brown
1M	(R15, R25)	brown	black	black	yellow	brown or brown black green gold

5. Power regulator LM317T (U1)

Some additional components out of the smallest plastic bag are needed here. The LM317T must be connected horizontally. It must be electrically isolated from the PCB, hence a mica insulator must be connected underneath the LM317T. At first we will bend the terminals of the LM317T. Place it flat on the PCB at the position of U1, such that the holes in both LM317 and the PCB are in equal position. Now it is clear how and where the three terminals need bending (over 90°) such that they fit in the PCB. After bending

place the insulator (from the small bag) on top of the PCB at U1 and place the LM317T on top of the mica, while taking care that the terminals fit in the holes PCB.

Take care that the mica insulator remains under the LM317T, and that the hole of the LM317T is still positioned above the hole in the PCB.

Now add the plastic pre-formed ring in the hole of the LM317T (from the bag again). It fits only in one way. Now take the separately delivered screw (the one in the plastic is too long), and fix it through the plastic ring, the LM317T, the mica and the PCB. Take the isolating (transparent) ring (inside the other plastic bag), and put in underneath the screw at the PCB solder-side. Finally attach the nut and tighten the combination. Watch correct position of the mica, and eventually correct! At last solder all three terminals.

6. Capacitor (C6)

Place it close to the PCB and solder.

100nF (C6) “small capacitor in blue colour”

7. Solid aluminium capacitor, DIRECTIONAL!!! (C4, C5)

Watch the direction! A small '+' is denoted on the capacitor, and on the top a black bar is drawn (the '-' side). After soldering, the '-' side of all elcos on this PCB should point in the same direction

10uF/25V (C4, C5) orange coloured.

8. Connectors ‘ASYMMETRIC’ (P1, PX)

3-cons (P1, PX) green coloured

Watch the direction in which these are attached. The open ends should direct to the outside of the PCB. No 3-cons will be placed at P3-P6! These will be hard-wired with coaxial cable later on (cabinet phase).

9. Relays ‘ASYMMETRIC (REL1) small block, 8 pins

It can only be connected in one way. Push tight to the PCB and solder.

10. Tube socket (V1) ceramic, round, 9 pins

The tube socket might not fit immediately, so you might need to carefully bend (tweezers) the terminals little inwards so that it fits into the PCB. Use enough solder such that the holes around the pins are filled completely. (The tube attached to socket V1 contains two triodes. Hence, in the schematics you'll find two references to V1, whereas on the PCB you'll only find one).

11. Electrolytic capacitors (Elco). DIRECTIONAL!!! (C3, C11, C21)

1000uF (C3, C11, C21) blue, round.

After soldering, the '-' side of all elcos shall direct the same side. Watch the direction!

12. Choke (L1), ‘ASYMMETRIC

Mount the choke as close to the PCB surface as possible, and solder the four terminals.

10H/15mA (L1) grey, block.

13. Electrolytic capacitors (Elco). DIRECTIONAL!!! (C1, C2, C10, C20)

Watch the direction!!!

47 μ F (C1, C2, C10, C20) blue, round

After soldering, the '-' side of all elcos shall direct the same side.

14. Paper-in-oil capacitor (C12, C22)

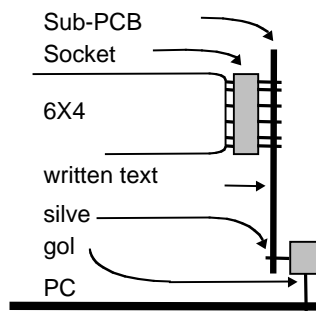
By incident, 0.33 μ F is mentioned in the circuit diagram and component list. The leads of the paper-in-oil capacitors are also relatively thick w.r.t. the diameter of the hole. Bend both wires directly where they come out the capacitor, and carefully pull them through the holes. You'll regret if a lead will break, because this is an expensive capacitor. Make sure to heat the leads enough when soldering.

0.47 μ F/400V C12, C22 yellow, round.

All paper-in-oil capacitors have been matched as close as possible for sonic reasons! This might result in two different coloured PIOs in your package. Don't worry, it's beneficial!

15. Small PCB (B1, PB1 / PB2)

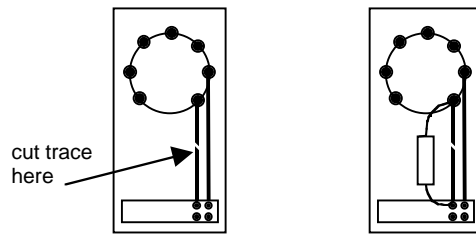
This set contains a small PCB, from now on called sub-PCB. A side view of the final construction is shown below.



Firstly connect tube socket B1 to the sub-PCB. It should be placed at the side where the written text resides, and soldered at the others side (no text). Apply enough solder, all holes shall be filled.

Then attach the remaining connector to PB1 at sub-PCB. It shall be connected with the silver-coloured terminals in sub-PCB. It shall be at the side where no text resides, and soldered at the side where text resides. The gold-coloured terminals shall direct downwards. Solder the silver coloured terminals one by one, and take care no short circuit appears in-between.

Secondly, apply a patch to the small PCB with a 1kOhm resistor as drawn in the following picture:



Back side view of sub PCB for EZ90 (part of SUPHV)

Finally, attach the small PCB to the main PCB at PB2. The tube socket shall direct towards the centre of the main PCB, i.e. the tube will finally hang above the main PCB. Make sure that the small PCB is vertical, because otherwise it will 'hang' after a tube is fitted in the tube socket. It might even be worthwhile to attach it with a very small angle, to compensate for the weight of the tube.

Don't press the golden terminals all the way down. You should just see the golden terminals appearing ($< 1\text{mm}$), and solder it like that. In that case, the rectifier tube will hang as high as possible, imposing less heat to the surrounding components.

Take a look to the photograph (see phase 3 below) and the drawing above to see if all went all right.

16. Tubes (B1, V1)

No tubes shall be attached in their sockets yet. That will come later, during and after the test procedure.

Third phase (testing)

You have built a quite impressive looking print with transformer and tubes, requiring a careful step-by-step test. Be aware that a dangerous open mains voltage and an even more dangerous secondary high DC supply voltage for the tubes are present on the PCB while testing.

These high supply voltages will remain there quite a while after the power has been switched off! After removing the mains cord, 5 minutes shall be taken before touching any part of the assembled PCB.

Please remind that the PCB and parts for this are very expensive. This means that if you make mistakes (which are hard to test for this phase), you may damage components and possibly also the PCB. We don't have any spare parts, so re-ordering will be much more expensive than the costs you've paid so-far. The general message is: make no mistakes, and double, triple, quadruple check your work!!!!!!!!!!!!!!

First check if all junctions are soldered at the backside. Again, make sure all components are at the right place. See the photo for details.



You need a multimeter, a mains cord, supply and brains to build and test suphv.

First test (no mains voltage applied)

Initial tests are passive only; that is with no mains power cord attached: this is to reduce the risk of blowing components in case of any assembly error.

1. Once again, inspect proper placement (orientation!) of diodes, capacitors, regulator, and check for correct values (color ring codes) of all resistors.
2. Check if all soldering joints at the bottom (solder) side of the print are carried out carefully.
3. Use a multimeter to check some connections. The meter shall be in 'resistance' mode: You may choose a scale of 100 ohms or next larger range. When the two probes touch each other, the needle swings to full reading (equals about 0 ohm), or the display indicates about 0 ohms.
4. Perform the tests with neither tube in their sockets. To follow the orientation of the descriptions below, place the print in front of you, component side above, the transformer left and the on-print tube socket at the right.

Check for an (almost) 0 ohm connection between all following pairs of points:

- A 'ground' point (such as one of the outer (out of three) soldering points of one of P3, P4, P5 or P6), versus the center lead of the regulator (LM317T).
- A 'ground' point, versus pin 4 of the socket of the standing tube (V1, ECC88, on the main PCB). (With a top view, pin 1 is next to the spacing between the pins,

near P3. Now count counterclockwise until pin 4. That will be somewhere between R11 and R12.

- Pin 5 of this tube, versus R4 (the most upper left small resistor) terminal left (near the print edge).
- Pin 3 of the other tube socket (B1, on sub-PCB, count the way described with V1 socket) versus R1 (small resistor behind the sub-PCB right terminal).
- Pin 7 of the sub-PCB tube socket, versus R2 (large resistor next to the regulator) terminal towards you (next to the transformer). OK, this should give at least some confidence.

Second test (connected to SUPLV)

Suphv will now be set-up together with low-voltage supply print (suplv). Take that previously assembled print. You also need:

- A mains cord (which you probably used before to test that print)
- a short stretch of 2-lead mains power cord (lets say a few decimetres)
- a short stretch of 2-lead low-voltage wire, with different colours.

For all these wire ends: take the wires apart for a few centimetres, strip their isolation for the last half centimetre, twist the strands, and solder these to allow a firm and safe mounting in the connectors with screws.

Place both prints in front of you on an insulating surface (yesterday's paper). Be sure that there are no loose wires, solder stretches, screws etc. beneath them.

Warn people around you and take utmost care of animals. Feed them and send them to another room.

1. If not yet in place, fix the power cord (not yet in mains!) into the connector P1, pins 1 and 3, of the low-voltage power supply print. Connect the low-voltage wire with the (lets say) black wire to the centre pin of P1R (denoted 0, indicating ground), and the (lets say) red wire to the '+' pin of P1R.

Connect the other end of the 'black' wire to the new tube print, connector PX (at the side towards you), at pin 3 (right, next to the relay).

Watch out, a wrongly connected wire will apply reverse voltage, and will damage some (cheap) components when powered!

Now leave the other ('red') wire end near the tube-print, unconnected until now! Also don't yet connect the mains power to the tube-print! You now may connect mains power to the low voltage supply print; its LED will light up red.

2. Select your multimeter for DC voltage measurement, 10 V or nearest higher scale. Connect its' ground (reference, black) probe to the screw in the tube print, holding the black wire next to the relay. Check the voltage at the loose ('red') end of the wire near the tube print, (with the other probe), to see that it reads + (yes, PLUS!) 5 volts.
3. Put the meter probes aside and take the 'red' end of the loose wire. Touch momentarily with its stripped/soldered end the inside of pin 1 (left side near the screw hole) of the same PX connector. You should hear a 'click' of the internal switching of the relay.

Congratulations! If you like applause, make a few more clicks

If you don't hear the relay, don't keep the red wire connected any longer: you might blow your power supply and or other components due to wrong orientation! If you don't see any mistake, call for help.

Test the tube print under power

Disconnect the mains power from the low-voltage print. Connect the 'red' wire to pin 1 of the PX connector, tube print (where you tested the relay before).

Connect a mains double wire, between P1 on the tube print (pins 1 and 3) and P2 (pins 1 and 3) of the low-voltage print (denoted '230 out') (Orientation is not important for this wire.)

Place the ECC88 tube into its socket (V1, standing straight up on the main print), leave the other tube aside for a moment. (Press it in gently, wiggle a little, and provide back pressure behind the socket if necessary.) Now connect the mains power to the low-voltage print.

Watch out, highly dangerous open voltages are everywhere! The DAC design group does not accept any responsibilities for damage or injury! If you do not want to take risks contact your contact person in order to look for alternatives.

The red LED burns again at the power supply print; you hear a click of the relay. After a few seconds the ECC88 tube starts glowing internally, with a yellow-orange colour. SUCCESS!

Take the multimeter at hand, with its (black) reference probe connected to a 'ground' point of the tube print (see above), prepared for DC voltage measurement

1. With the meter in a 10 V DC scale and its black probe at ground. Check the voltage at R4, upper left edge of the print, left terminal: it shall read -6.3 volts (+/- 0.2 V). It shall be a negative voltage, if needed exchange your probes to get a positive value.
2. Disconnect the mains power cable, attached to the low voltage supply.
3. Wait for at least 1 minute.
4. Take out the ECC88. Give some re-active force to the main PCB.
5. Plug the '6X4'-rectifier tube into the sub-PCB socket. (Press it in gently, wiggle a little, and provide back pressure behind the socket).
6. Connect the mains voltage again; now the 6X4 tubes should start glowing after few seconds, if not disconnect mains immediately!
7. Disconnect the mains lead and wait for 1 minute.
8. Plug in the ECC88 again.
9. Switch the meter to 100V or nearest higher DC scale, black probe to ground again.
10. Connect the mains lead again, watch the tubes and give them some 15 seconds to heat up and stabilise.
11. Measure the voltage at the left side (transformer side) of each of the large capacitors C22 and C21 (yellow or orange colored). You should measure a voltage at each of those of about 88 volts (+/- 8 volts) to ground.

Congratulations: You successfully built your tube output stage! If you like you may watch the glow of the tubes and eventually switch off all external light sources. After that:

Disconnect the wires from the connectors, and store everything aside for the upcoming greatest challenge: the digital PCB.

The digital PCB (DIG)

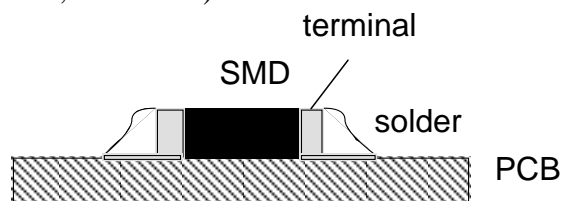
First phase (check)

We'd like to give you a very important warning first. Desoldering components from DIG is really a hassle. So, please, please, please prevent making errors!!! You'll regret it, and you probably have to re-order the components you have to desolder.

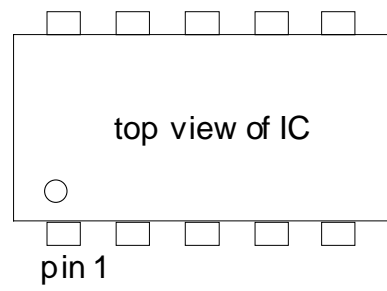
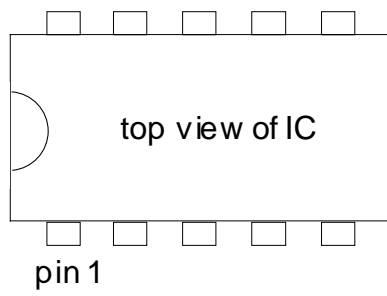
The digital PCB has to be assembled on both sides. First the bottom side (side without ICs) will be assembled, then the upper side (side with the ICs).

Soldering DIG takes quite some time. Don't think you can do the job within a day. The dimensions of DIG are quite small, so you need good light, and patience. Again, better spend 2 or 3 days without any errors, than trying to speed up, and end up with errors, expenses and a lot of time needed to debug the whole design.

There are some new components, which have not been discussed in the SUPLV and SUPHV part. On the bottom SMDs (Surface Mount Device) must be attached, on the upper side 'ordinary' components must be attached. An SMD is small, and therefore needs some special attention. The idea is to first apply (!) very little (!) solder (e.g. use 0.1mm - 0.3mm solder, and a solder iron with a very fine tip) on one island of the PCB where the SMD must be attached. Then take the SMD with a pair of tweezers, hold it above its intended position, and gently shift the SMD to its place on the PCB while heating the pre-soldered side. Make sure that the SMD is mounted close to the surface, so no air gap is visible between the PCB and the SMD. Then, apply solder to the other side. Use little solder, the connection should look hollow (see the picture). Finally, apply some additional solder to the first island to make a good final connection. If the connection fails, desolder the SMD, remove the solder from the PCB, take the spare component that we have supplied (not for all values!!), and start again. Do not bend the PCB after soldering, because your SMD artwork may fly through the room if you do so (which might look very artistic too, but well...).

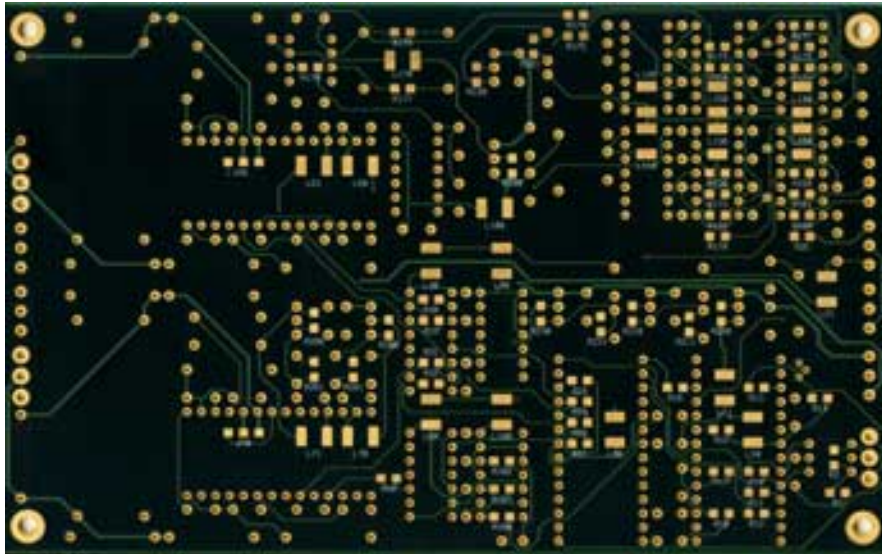


The other class of components are chips. Chips look symmetric from the outside, but internally are directional, and therefore need to be attached in the right direction. The standard way of determining the orientation of a chip is to locate pin number 1. The figure below gives an idea where pin number 1 w.r.t. the top view (terminals be directed below) is located. On the top of the IC text about its type number is printed.

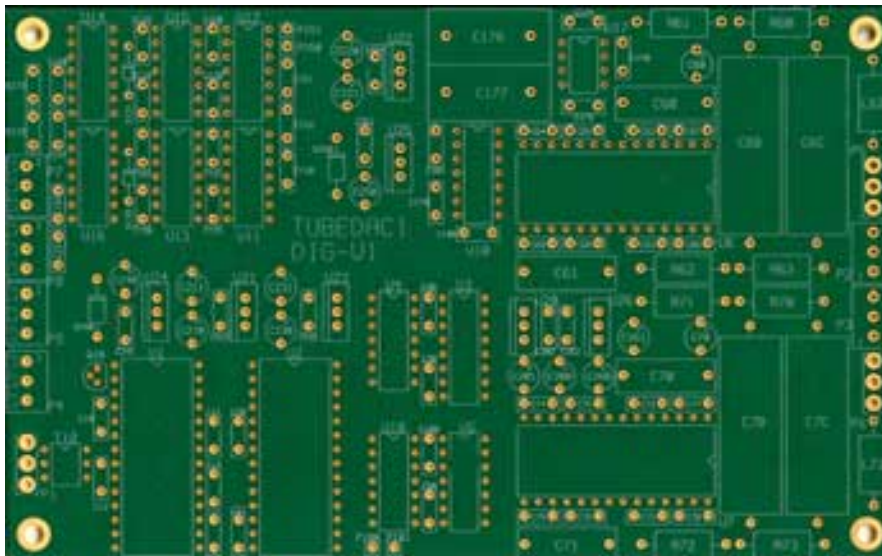


The PCB of DIG consists of two sides, as shown below:

Bottom side:



Upper side:



The assembly will be split into 2 parts.

Second phase (building the bottom side)

1. SMD resistors

For some values we did provide an additional SMD resistor in case something goes wrong. Don't sneeze when SMD is lying on your table, you'll regret it. Also look out for wire movement of your soldering iron, and the sleeves of your sweater. Take care that the text on the SMD remains visible after soldering. For reading convenience, orient all text in the same direction.

R2	150 R	150Ω
R182	75 R0	75Ω
R10	604R	604Ω (plastic case)
R162	2204	2M2
R174, R175	4703	470k
R130, R131	2201	2k2
R132, R133	4704	4M7
R240, R250	33R0	33Ω
R176, R177, R13	4702	47k
R160, R161, R25	470R	470Ω
R171, R172, R173	1004	1M
R170	1104	1.1M
R50, R200, R210, R220, R230, R260	330R	330Ω
R14, R15, R1, R150	1002	10k
R30, R31	100R	100Ω
R178	4702	4k7
R11, R12, R16, R17, R18, R21, R22, R23		
R24, R32, R33, R180, R181, R201, R211, R221, R231, R261	1001	1k

2. SMD beads

The beads are the square small black boxes packed in the plastic strip. There are 20 beads, they all have the same value:

L10, L11, L20, L30, L40, L50, L60, L61, L70, L71, L100, L110, L120, L130, L140, L150, L160, L170, L180, L21

Third phase (building the upper side)

The following components are mounted on the top side, and soldered at the back side of the PCB.

3. Diodes

Mind the direction.

D130, D131	1N4148
D240, D250	BZX85

4. Chokes

They look like resistors.

L62, L72	brown black red silver	1mH
----------	------------------------	-----

5. Resistors

R60, R63, R70, R73	brown green black black brown	150Ω
R62, R72	blue gray black black brown	680Ω
R61 and R71	not used!	

6. IC socket

2 IC sockets have been supplied for chips which might be updated by their manufacturer with new and better versions. These ICs are situated in U1 and U2. On the PCB text side a little triangle is drawn just below the designator U1 and U2. The IC sockets have a little gap at almost the same place. Position the gap of the IC socket at the same side, so that you have the same reference when plugging in the chip later on. Place the IC socket as close to the PCB as possible.

U1, U2	IC socket
--------	-----------

7. Chips

Align the triangle denoted in the chip mark on the PCB to the mark on the chip. Six chips are close to another (U14, U16, U15, U13, U12, U11). Mount these components first before soldering!

U16	74HC32
U15	74HC00
U13, U14	74HC74 (NOT the 74AC74)
U11, U12	74HC393
U4	74AC74 (NOT the 74HC74)
U3	74HC175
U18	74HC04
U5	74HC02
U6, U7	PCM63
U17	TLC271
T10	T4-6T (some are black, some are white)

8. Capacitors

This concerns the small blue-coloured capacitors. Mount them in such a way that you can still read the text. This means that capacitors are relatively mounted in different directions.

C151	101J	100pF
C21	331J	330pF
C12	103J	10nF
C13	473k	47nF
C171	683k	68nF
C178	335M	3.3uF
C131, C132	22 J	22pF
C172, C173	682J (not 684k!)	6.8nF
C174, C175	684k	680nF
C14, C62, C63, C72, C73	102J	1nF

All the other small capacitors are 100nF (104k)

C10, C11, C20, C30, C40, C50, C64, C65, C74, C75, C202, C212, C222, C232, C241, C251, C262, C69, C6A, C79, C7A, C66, C67, C76, C77, C100, C110, C120, C130, C140, C150, C160, C170, C180, C152, C133, C134, C101, C161, C22.

9. Transistor

The geometry of the transistor sign on the PCB should match the way the transistor is mounted. The pads are very small, so take care!

Q10	C547B	BC547
-----	-------	-------

10. Connectors

There are six connectors. Direct the inlets to the outer side of the PCB.

P4, P9, P8, P7, P2, P3	CON3
P1, P5, P6	no connection

11. Elcos

These components are directional (bar denotes minus). Mount in the right direction (positive side is marked on the PCB)!

C201, C211, C221, C231, C240, C250, C261, C200, C210, C220, C230, C260, C68, C78	10 μ /25V	10 μ F, 25V
--	---------------	-----------------

12. Regulators

All regulators are mounted in the same orientation. The metallic back is denoted on the PCB, and hence the regulator should be attached as such. Take care that the LM337 is at the right place!

U26	LM337
U20, U21, U22, U23, U24, U25	LM317

When cutting the terminals of the regulators, save two pieces of terminal for making junctions in a later step.

13. Capacitors

C60, C61, C70, C71	0,22 MKP	0,22 μ F
C176, C177	4,7 μ F MKS	4,7 μ F
C6B, C6C, C7B, C7C	0,033 μ F paper-in-oil (yellow)	

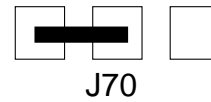
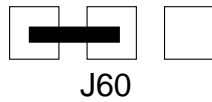
14. VCXO

Pin 1 is indicated with a '1' in a black box. It should be situated at the side of the triangle on the drawing of the PCB.

U10	Raltron 11.2896MHz
-----	--------------------

15. Junctions

Two junctions (J60, J70) need to be connected as in the following figure (the middle pad with the left pad, take the text underneath as a reference). Use a piece of terminal cut from a regulator.



16. Chips

Put the remaining chips in their sockets. Watch the direction.

U1

CS8412

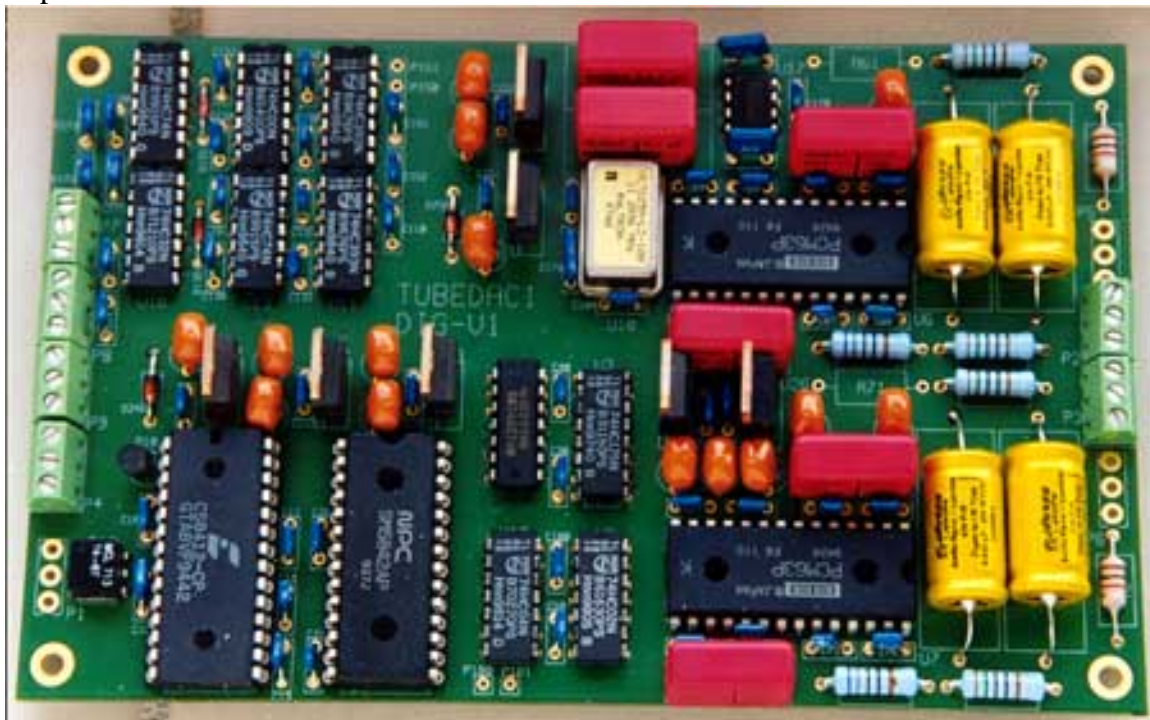
U2

NPC SM 5842AP

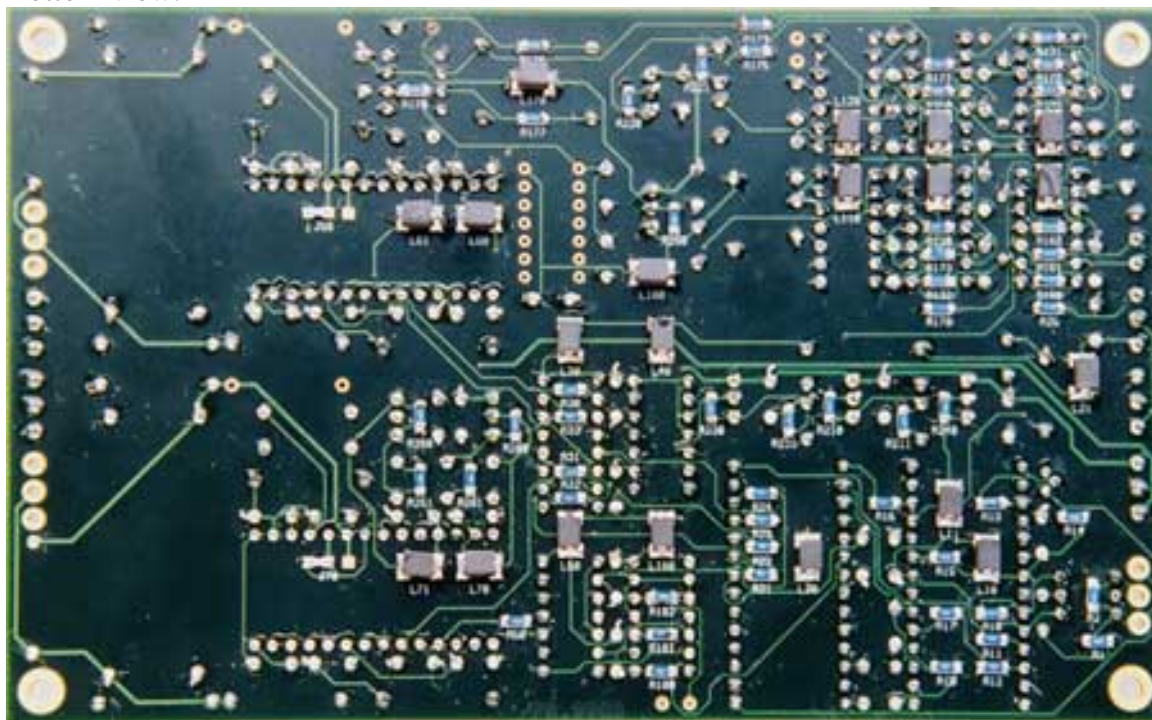
Fourth phase (checking and testing)

After building, the PCB should look as follows.

Top view:



Bottom view:



This print contains a quite large number of components, and some mounting errors can easily destroy them. So although not too much can be tested without actually connecting the power supply, we will do what we can. For the description below we assume an orientation of the print in front of you, with the IC's and capacitors on the upper side, the four connectors at the left, and the four cylindrical capacitors at the right hand side. For all tests described below it is recommended that the IC sockets remain empty.

1. By inspection.

You will once more carefully check the print for errors, which might have catastrophic effects later on. In particular you should check:

- The orientation of all IC's should be checked, and match with the orientation on the photograph.
- All regulator IC's (U20 to U26) have identical orientation, with the metal backside at the left (assuming that the PCB is oriented such that you can read the text). Only the right most one (U26) is of type 'LM337'.
- At the bottom side of the print, the SMD resistors R200, R210, R220, R230, and R260 are 330 ohm. The resistors R201, R211, R221, R231, R261 are 1k ohm.

2. By measurement without power.

With your multi-meter adjusted to measure low resistance values, we will check for a few proper connections. (The print is again with the IC side upwards.) These are power supply lines, measured through the small beads at the bottom side. You should measure a short-circuit (resistance less than 1 ohm) between: U6 pin 13 and U7 pin 13.

Also between the pins in the sets:

- U6 pin 11, U7 pin 11

- U3 pin 16, U4 pin 14, U5 pin 14
- U11 pin 14, U12 pin 14, U13 pin 14, U14 pin 14, U15 pin 14, U16 pin 14

3. By measurement with power on.

To bring power to this print, you need to connect it with the analogue power supply 'suplv': the print you had built first. The connections you have to make between these prints are only from digital print connectors P2, P3 and P4 to suplv connectors P1D, P1L, P1R. For details on these wires see the next chapter on chassis mounting. With these two prints in place, some low-voltage wires between them, and 220V mains power connected to 'suplv', you should check the power supply voltage of all IC's. (Note that the IC sockets are easily and safely measured with their IC's not in place!)

About +5 volts (a deviation of 0.2 volts is still OK) at the pins:

U1 pin 7, U1 pin 22, U2 pin 22, U3 pin 16, U4 pin 14, U5 pin 14, U6 pin 2, U6 pin 13, U7 pin 2, U7 pin 13, U11 pin 14, U12 pin 14, U13 pin 14, U14 pin 14, U15 pin 14, U16 pin 14, U18 pin 14.

About +6.3 volts at U17 pin 7.

About -5 volts at:

U6 pin 11, U6 pin 28, U7 pin 11, U7 pin 28.

With these values confirmed, you have soldered many components correctly.

The cabinet

Introduction

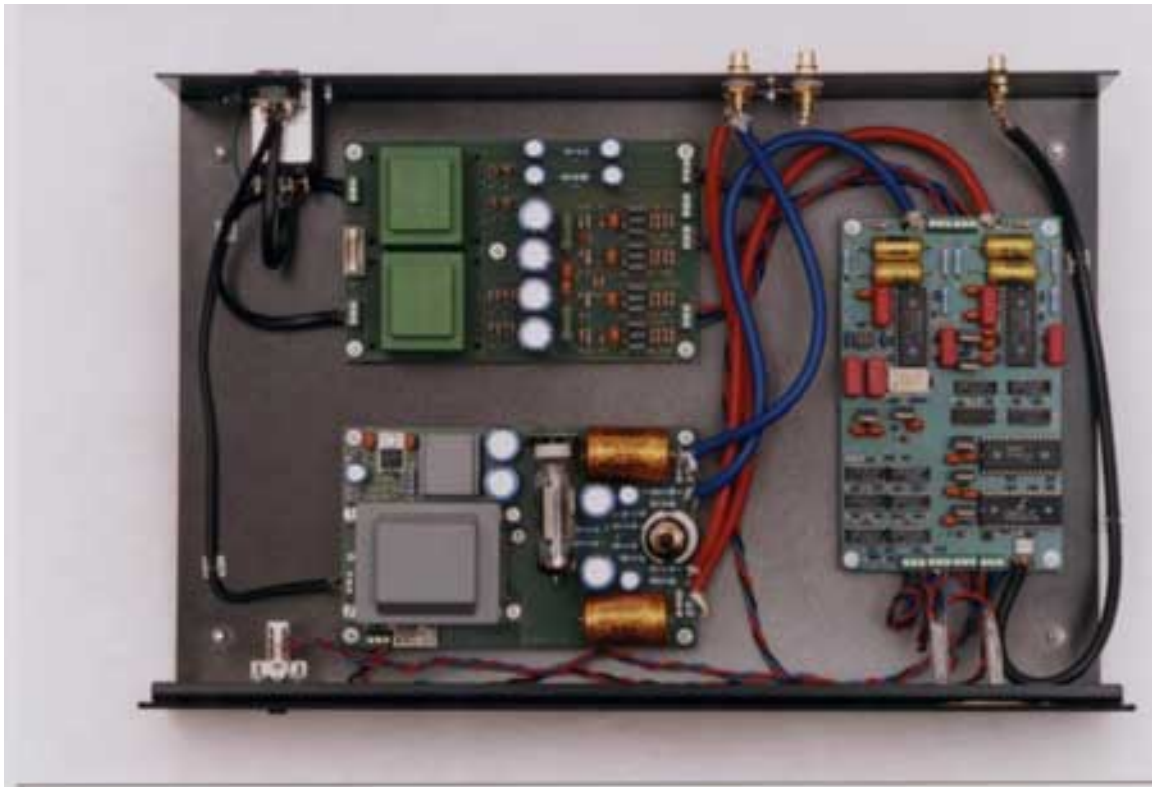
The cabinet consists of three parts:

1. Bottom part (in which all PCB's, switches, LEDs and connectors are mounted)
2. Cover
3. Front plate (which is glued to the front of the bottom part).

Mounting the cabinet consists of two phases:

1. Mechanical
2. Electrical

Please take the photograph as supplied as an indicator; lots of details are more clear just by looking at it.



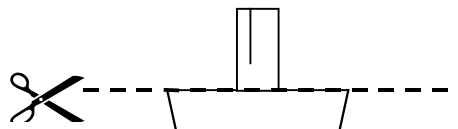
Mechanical

Please refer to the electrical wiring diagram, as included in this manual. This diagram shows the orientation of PCBs and connectors.

1. Bottom

We will first mount the 4 rubber feet in the chassis bottom. This is performed to protect the cabinet during assembly, and you'll have to remove them later on (so don't mount them too tight).

- The center black plastic cylinder is to be pulled out, and can be thrown away. The upward pointing soft rubber cylinder is to be cut off. Now four M4 'VK' bolts can be used to put the feet onto the pre-mounted nuts in the chassis.



- Mount 14 bolts M3*16 CK in the holes (upwards), intended for mounting the 3 PCBs. Use the nuts with height 7,5 mm (don't use any washers here). As a result 14 mounting points become available. Later on 3 PCBs will be mounted upon them.
- Take the mains entrance/filter and push that in the lower square hole at the back. Scratch the lacquer of the outer hole, inside the cabinet. Then a proper electrical ground-connection can be made later-on. Fix the mains entrance using 2 bolts M3*10 VK. Take two nuts. Mount one at the right bolt. At the left bolt, at first a toothed lockwasher shall be applied, then a solder lug follows, then a normal M3 rivet where after the second nut shall be added.
- Now take the switch. Push it in the other square hole, with the "I" sign pointing upwards.
- After that, 4 cinch connectors can be mounted. Please note the orientation:
 - Red at the lower two positions
 - Black at the upper two positions

These connectors are supplied with:

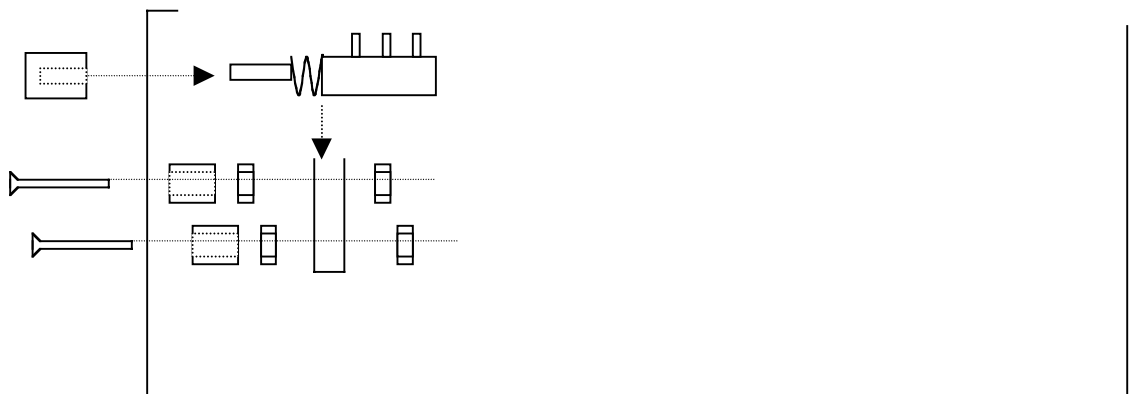
- 2 plastic rings (one of these is flat, the other not).
- 1 nut.
- 1 rivet with solder lug.

Note that 1 of the 5 connectors is supplied with only 1 plastic ring. This connector shall be used later-on.

Scratch the lacquer (inside, just around the hole) of the thin hole centered in-between the 4 holes.

Take the first of the 4 connectors and release all loose parts. Look for the plastic ring that corresponds with the colour of the connector (either black or red). Add this ring to the connector. Push it through the cabinet wall. Add the other plastic ring, the rivet with lug and the nut. Mount it loosely but such that the non-flat ring falls in the hole in the cabinet. Do the same with the other 3.

- Scratch the inner side around the 5th hole, just around it such that the 5th cinch connector after mounting contacts the cabinet. Contrary to the above 4 connectors, the second plastic ring shall not be used (not present so do not look for it).
- Take a multi-meter, set to "ohms" (lowest range, 200W or so) and check for a proper connection between the solder lug of the 5th cinch connector, and the solder lug of the mains entrance.
- Now take an M3*10 CK, 2 solder lugs, an M3 toothed lockwasher, a normal M3 rivet and a nut. Insert the bolt through the outside of the cabinet, add a toothed lockwasher, two lugs, the normal rivet and tight the whole with a nut.
- Check for proper connection between one of the before mentioned lugs, using the same test method.
- The high-voltage power-switch will be mounted in the front side of the chassis. For this you need the switch itself, its mounting rail, two little M2 screws, two plastic distance rings, and 4 nuts. The two screws are first mounted in the chassis with the plastic rings at the inner side and a first nut. The switch should be pressed into its mounting rail, and slid on the screws. This is fixed by the other two nuts. Finally the black plastic cover is pressed from the outside onto the switch.



- The remaining two holes are intended for the two LEDs, but these will be mounted later on.
- The three PCBs can now be placed on the previously inserted screws. Fix them with 14 rings and 14 nuts M3.
- Unfortunately there is a small problem in the chassis construction: The two right-most bolts conflict at the bottom side with the cover, when you try to close the chassis. Possible solutions are:
 - leave out these two bolts, use shorter ones (5mm) to screw the 7.5mm nuts to the PCB, and glue these nuts to the chassis.
 - Use two other 'VK' type bolts, and drill the two holes accordingly.

- Ignore the problem, and just mount. The cover will bend a bit, it doesn't look very nice at the bottom, but you won't notice the problem at all when you position the cabinet normally.

2. Front plate

The front plate must be glued to the bottom chassis, but you'd better first attach the electrical wires, described below. One suggested solution is to use 'montagekit'. Put sufficient montagekit to the cover side of the bottom chassis, put the front plate on it, and make sure it is positioned right. Take care that the switch on the front can be operated.

3. Cover

The cover slides over the bottom chassis. The holes underneath are directional, so it will fit only in one way. Take care that you don't damage the tubes or other parts while sliding the cover. And, of course, first remove the plastic cylinders before sliding the cover.

Also the LEDs must be mounted in the cover (see below).



Use glue to mount the LEDs in the holes. Use glue which does not dissolve plastic, and sticks to aluminium.

Electrical

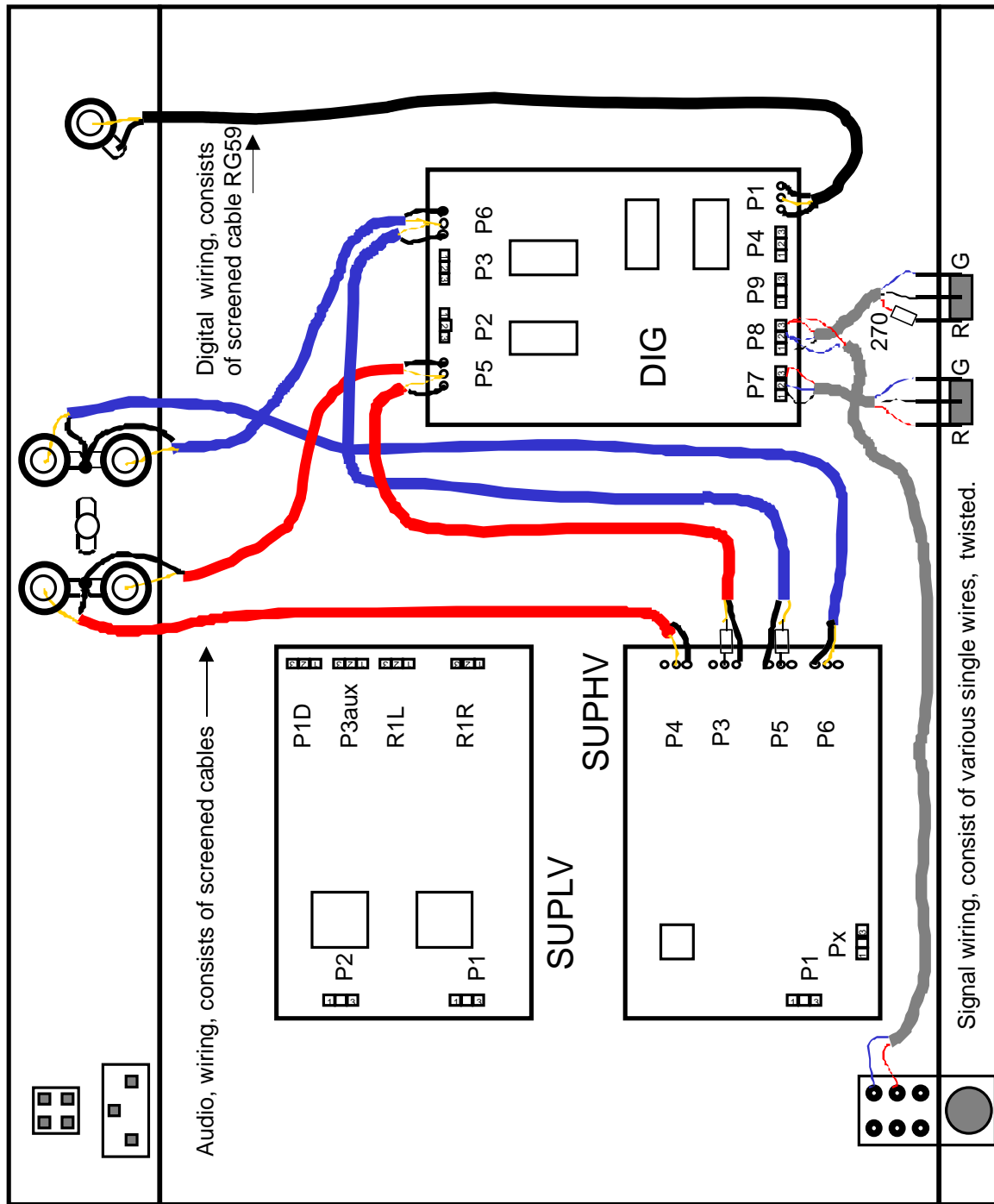
We propose to apply the wires one by one, with for each wire:

- measure
- cut
- strip
- solder / fix in connector

Wiring length

Please take note of the diagram and the following wiring schemes.





As a guideline, the following lead lengths are advised. Please take note that definitive lengths are depending upon the wiring as applied in the cabinet. As an advise, apply the complete cable at the correct position, inside the cabinet, just as if it is in its' correct place. Then decide the correct length. Take some 30 mm extra to be sure (!)

- Signal wiring (single wires, to be twisted)

LED P7	100 mm (Red/Blue/Black)
LED P8	100 mm (Red/Blue/Black)
Switch P8	400 mm (Red/Blue)
Relais PX	350 mm (Red/Blue)
P1D/P4	350 mm (Red/Blue/Black)
Va P2	200 mm (Red/Blue/Black)
Va P3	200 mm (Red/Blue/Black)

- Audio wiring (shielded, either Red (right channel) or Blue (Left channel)). Numbers refer to SUPHV

P5 to cinch	100 mm
P6 to cinch	150 mm

P3 to dig	300 mm
P5 to dig	300 mm

P4 to out	200 mm
P6 to out	250 mm

- Digital wiring (Shielded cable, black, noted RG59)

P1 to cinch	300 mm
-------------	--------

- Mains wiring (black, double insulated)

P1 to P2	300 mm
P1 to switch	150 mm
switch to entrance	100 mm

- Earth wire (green) 100 mm

Stripping / soldering wires advice

- General

Some insulation needs to be stripped from the wiring. The necessary length depends on the application. Take care not to damage to inner structure (either copper or another insulator) when stripping. After stripping, twist the various strands of wire.

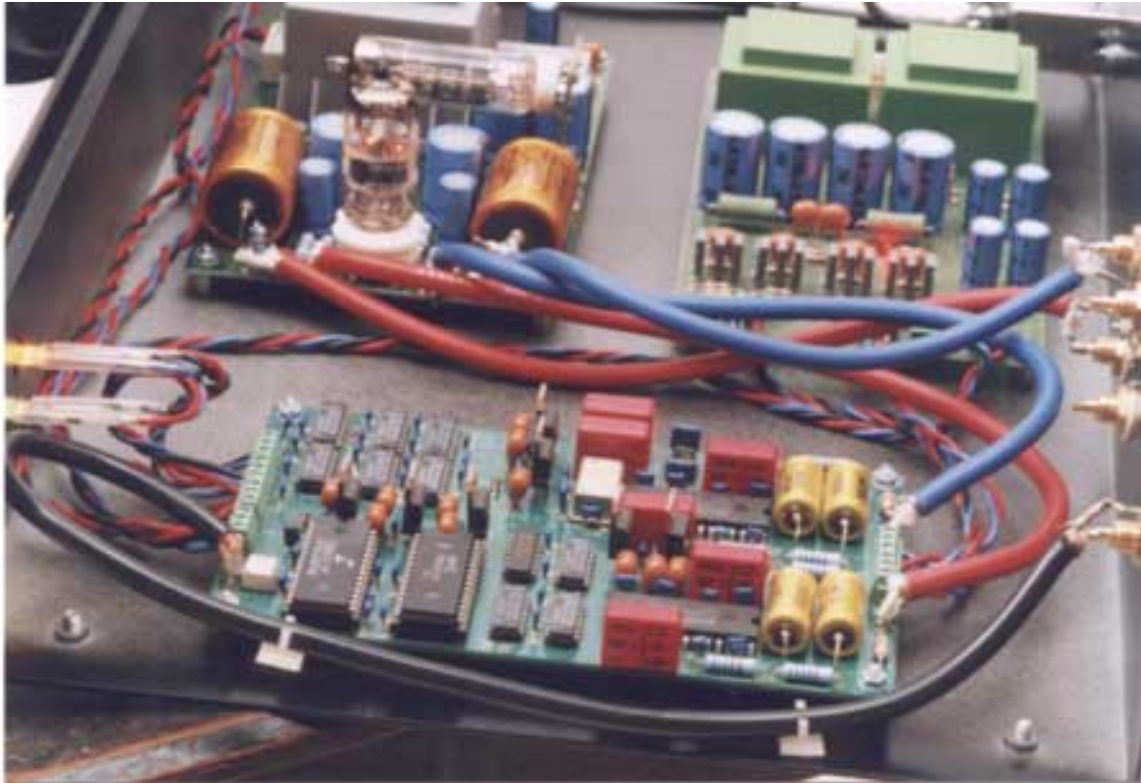
- Mains (black double insulated)

Strip the outer insulation. About 40 mm will do. The easiest way is just to cut (exactly) in-between the inner two wires. Now the outer insulation can be peeled of, as if it where a banana. Now strip the inner two wires at about 5 mm. Take care, do not damage the copper strands.

The mains cable shall be soldered at the mains connector / switch (both at the back). The other ends shall be screwed at connectors. Do not use solder here!

- Shielded cables – digital (black round coax)

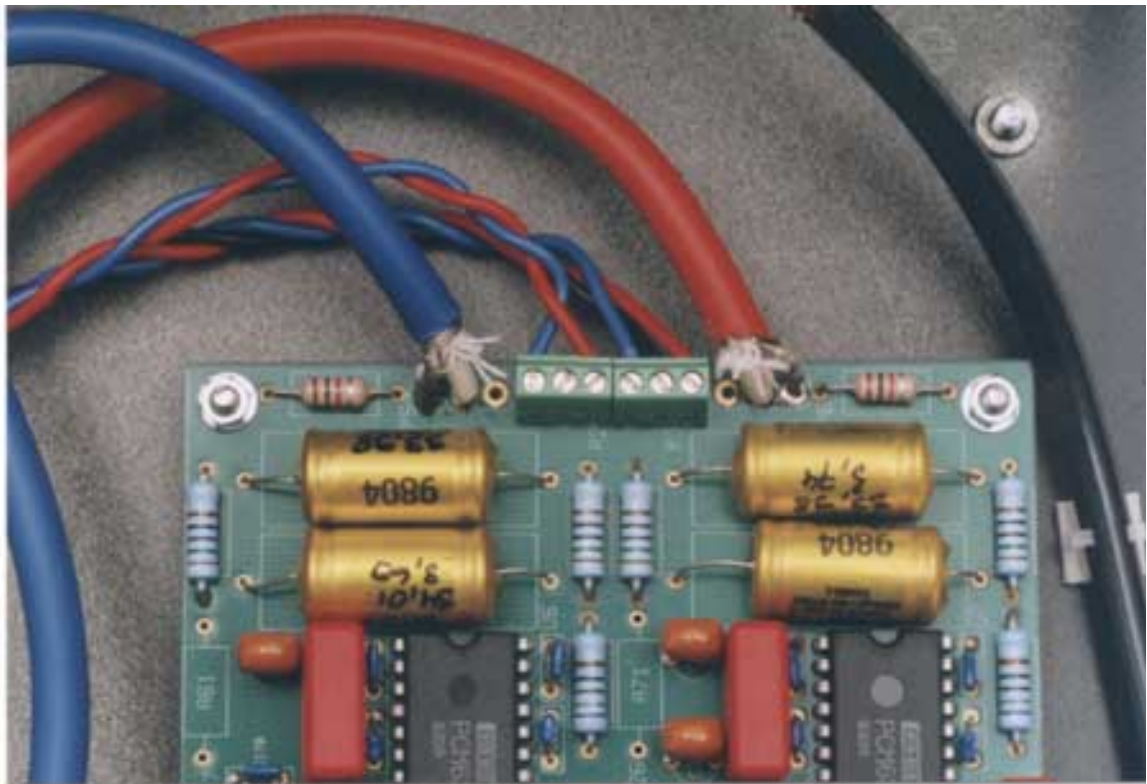
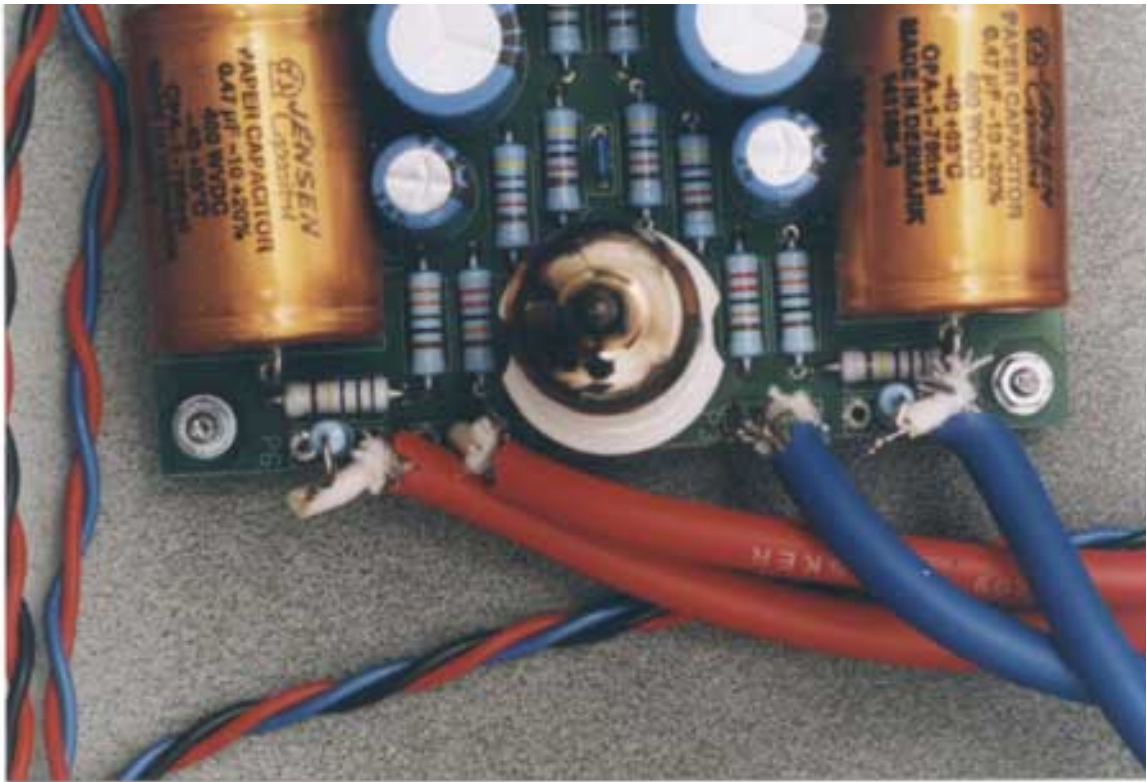
Strip about 30 mm from the outer insulator. Do use a sharp (Stanley) knife. Be very careful as it is easy to damage the screen (Practice at a loose wire). Then release the braided screen and twist these strands to one wire. Apply some solder such that they stay together. Now strip, about 5 mm, from the inner isolator, again use the knife. Twist and apply some solder again.



- Shielded cables – audio (red and blue coax)

Before cutting the wires in pieces, be careful that everything fits: the provided wire length is only just sufficient. Strip about 30 mm from the outer insulator. Do use a sharp (Stanley) knife. Be very careful, as it is easy to damage the screen (Practice at a loose wire). Cut away the cotton inner wires (dirty job). Then release the braided screen and twist these strands to one wire. Apply some solder such that they stay together. Now strip, about 5 mm, from the inner isolator, again use the knife. Twist and apply some solder again. The center lead of each wire, should be soldered into the middle of the three holes in each PCB.

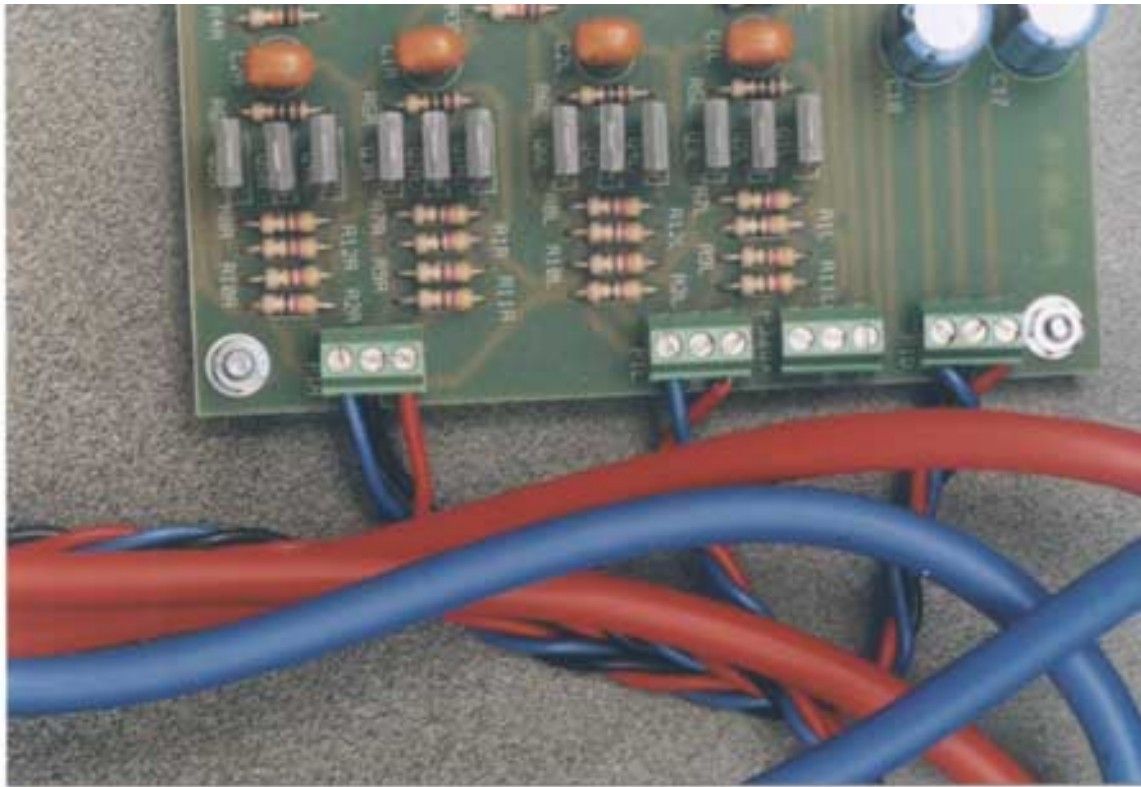
Please note that two resistors (100 ohm) need to be added in series with the inner wire, at P3 and P5, as shown in the diagram. Prevent a short circuit between resistor wires and screen of cables (not dangerous, but the music will be killed).



- Single cables (applied in 2- or 3-wire bundles, drawn gray in the picture.)

Strip 5 mm. Twist these, do not apply solder. If all single wires are cut, 2 or 3 various colors may be twisted together.

In general Red is used for "+" or signal (often "1" at the connectors), Black for ground or "0" (often "2" at the connectors) and Blue for "-" (often "3" at the connectors, but not for the low voltage supply).



Procedure

If all wiring is finished please add the two capacitors (1 nF, yellow/brown) between the cinch connectors and the chassis. Add the green earth wire nearby the mains entrance as well.

The LEDs can show two colors. If the LED is put flat on the table, with the leads pointing towards you, and the middle lead bent down, the left lead refers to the green (G) color, the right to the red (R). As second check, the 'red' lead is the shortest of the three. Please note, that a resistor is added, in series with one 'red' LED connection, see wiring scheme. Shorten the leads of the LEDs to about 15 mm, as well as those of the 220 ohms resistor (red-purple-brown-gold). Solder the resistor to one of the LEDs, and solder the 6 wires to the LEDs. Now shift 6 small pieces of the (clear, colorless) isolation tubes across the bare wires and solder joints, to protect them from shortages. The resistor will be fully inside one of these tubes. Finally heat the plastic tubes, by keeping them for a short while narrowly above your soldering iron. This will shrink the tubes to fix them. Finally twist

the 3 wires of each LED together. After fixing the other ends of the wires to the DIG PCB, you leave the LEDs lying around: they will be fixed in the front later.

If all wiring is in place, the unit can be checked (preferably by another), connected in the system, and switched on. At this point, you might to follow the test procedure of the digital print, if this is the first time it is powered. If everything works fine, the intended behavior of the LEDs is as follows:

- The LED connected to P8 (the right one):

If the power-switch at the back is turned 'on', always some light should come from this LED. If the front switch is turned 'off' then red, if turned 'on' then green.

(The front switch will turn off the tube stage only, thus silencing the active/amplified audio outputs.)

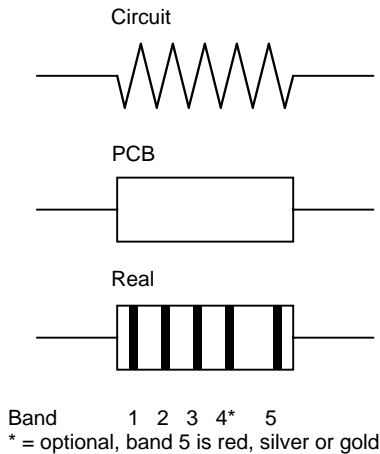
- The LED connected to P7 (the left one):

If the DAC works correctly, this LED does not light. (Actually, any light here causes the audio output to go silent.) If the digital input doesn't detect a proper digital (SPDIF) signal, this LED shows yellow (green and red both on). If it shows either red or green light, the DAC is trying to get in lock with your CD player. Such light should end after a few seconds, otherwise you have a serious compatibility problem.

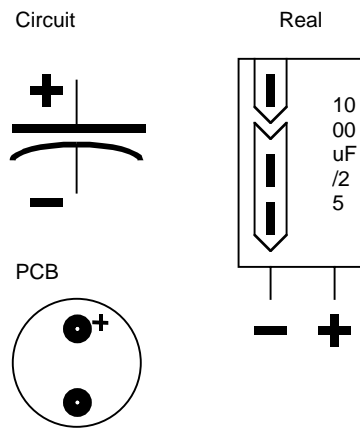
If all works fine the wiring can be fixed using some of the supplied studs (to be glued to the bottom).

Appendix 1: Component Guide

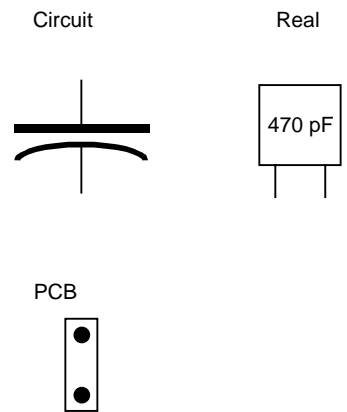
RESISTOR "R"



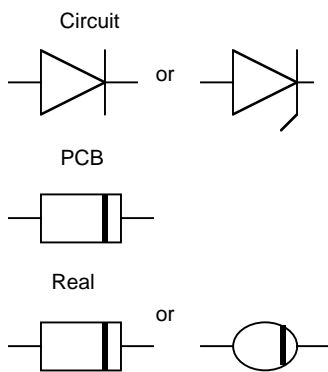
ELECTROLYTIC CAPACITOR "C"



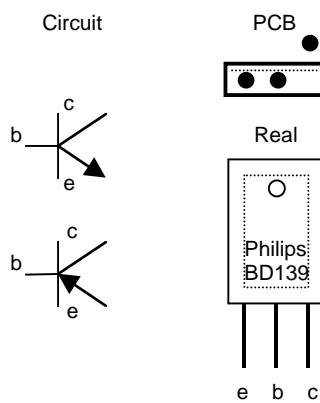
CERAMIC CAPACITOR "C"



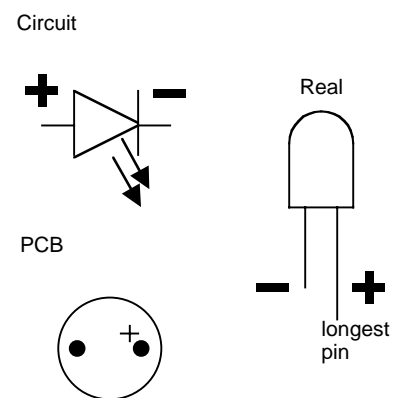
DIODE zener or normal "D"



TRANSISTOR "T"

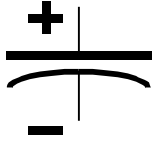


LED "D"

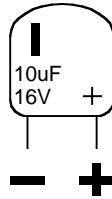


ALUMINIUM CAPACITOR "C"

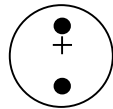
Circuit



Real

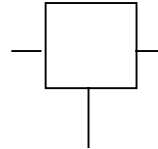


PCB



Regulator "IC"

Circuit



PCB



Real

