



Construction manual for

RoeTest V7

professional tube testing system

(c) Helmut Weigl

much thanks for english translation to Ton Regeling, USA

As of: 01/2013

Table of contents

Introduction to this manual	3
Disclaimer:	3
Copyright:.....	4
Hardware Revision History:	5
RoeTest4 – changes compared to RoeTest3:	5
RoeTest V5 to RoeTest4:.....	5
Changes RoeTest V6 to RoeTest5:.....	6
Changes V6.2 to V6:	6
Changes V7 to V6.2.....	7
Functional description:	7
Block diagram:	8
Circuit description:	9
Boards:.....	11
Component selection:	12
PCBs:.....	14
PCB construction:.....	17
Socket box receiver – mechanical construction:.....	27
Housing/cabinet:	33
Wiring:.....	47
First time power on / test procedure:	48
Constant voltages.....	50
Variable/microcontroller controlled output voltages:	50
Calibration of the 600V board:	52
Calibration of the voltage measurement ranges:.....	52
Calibration of 600V measurement range on G3 card (from version 6 onwards):.....	52
Calibration of the heater voltage measurement:	53
Current measurement calibration:.....	55
Current limiter:	57
To test voltage regulation of the H, A and G2 boards:	57
Test check for continuity circuits:.....	57
Closing remarks:.....	58

Introduction to this manual:

Building the RoeTest tube tester is not a project for beginners. Due to the complexity and the size of the project and also the fact that this tube tester works **with high and lethal** voltages experience is required. I recommend that you only try to build this tester if you have ample knowledge of and experience with electronics.

That said, the tube tester can be built successfully as shown by the testers built by other people. You can see these on the website. The manuals appear to be complete and correct, although I would be grateful if you send me feedback if you find errors in the manuals or if you have improvement suggestions.

Disclaimer:

THE ROETEST CIRCUIT DESIGNS, HARDWARE, SOFTWARE OR ANY OTHER MATERIAL ARE PROVIDED BY ME 'AS IS' AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL I BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THE CIRCUIT DESIGNS, HARDWARE OR SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Building your own copy is at your risk. Especially software is never fault free, and can lead to a malfunction.

Lethal and high voltages:

Tubes and tube testers including the RoeTest work with high and lethal voltages. Please note that working with high voltages is dangerous. The voltages in the tube tester and also on the tube sockets pins are dangerous and can lead to injury or death. Use with care. Ensure the RoeTest is only used by adults with knowledge of vacuum tubes and electronics. Keep away from children and animals. You and anyone using the tube tester are responsible for making sure you meet all the nationwide and local safety regulations.

Charged capacitors, for instance in a power supply, will remain charged and contain high voltages for a long time after being switched off and/or disconnected from the mains, despite the use of discharge resistors. Please discharge these capacitors with a suitable resistor before working on or experimenting with the circuits!

The RoeTest is a do-it-yourself (DIY) project. The RoeTest is not UL/CE/GS/Tüv tested or listed and I cannot guarantee that use of the RoeTest is legal or permitted.

Changes can be made to the software or hardware at any time. Compatibility with earlier versions is not guaranteed, even when current versions are compatible with earlier versions.

Note related to the tube data database:

The extensive tube data database is constantly changing. Tube data is added, changed and/or deleted. Current tube data may be wrong or incomplete. No warranties are provided as to the completeness and/or correctness of the tube data. The database does contain data for tubes that the RoeTest cannot test, either because the required voltages or currents are too high or because they are for certain tubes for special applications.

Copyright:

Copyright Helmut Weigl, Heidestr. 7, 92708 Mantel, email info@roehrentest.de. I maintain all rights to the hardware, hardware designs and software. The software, hardware designs or any other material remain my intellectual property. You merely obtain a license to use the hardware designs, software or any other material for private use only.

The database is made available for private use only and it is not allowed to change the data structure and/or to remove any copyrights. It is not permitted to retrieve the data for in other applications than the RoeTest tube tester. .

Commercial use of the RoeTest is only allowed with written approval from me.

PCB designs and layouts may be changed for your own purposes and use. You are not allowed to give the PCB designs and layouts to third parties without my consent, even if you did not change anything. You may only send these to a PCB manufacturer for the purpose of ordering PCBs for own use. It is not allowed to resell these PCBs.

The firmware for the PIC-Microcontroller can only be obtained from me, in the form of an already programmed PIC Microcontroller chip. Copying and distribution of the software and firmware is not allowed. The PIC-Microcontroller has a firmware read protection that may not be disabled in any way.

When you print RoeTest test results you may not remove the copyright marks.

Many of the socket images used by the RoeTest software are provided by Mr. Franz Hamberger. Mr. Hamberger makes these images available for private use on his website. The internet address for Mr. Hamberger's website is:
<http://www.dl7avf.info/charts/roehren/index.html>.

Distribution of these images is only allowed under the conditions described on Mr. Hamberger's website.

Hardware Revision History:

RoeTest4 – changes compared to RoeTest3:

The RoeTest4 has the same functions as the RoeTest3. The circuits to supply the test voltages have changed. The following has changed:

- New circuit: electronic stabilization of the voltage supplies under load. Software compensation is no longer required.
- Simpler calibration of the voltage supplies.
- The +/- 2.5 V voltage supplies are no longer required.
- Special transformers are no longer required, low cost standard transformers can now be used.
- The main transformer has less windings.
- Maximum heater current is increased to 5 A (in the 12.75V range).
- The PCB layouts were redesigned requiring less patch wires.
- The USB interface is now on the main PCB such that just a small USB connector board is needed.
- My prototype is constructed differently – the main PCB is split in two PCBs and housed in a compact cabinet with transparent sides.

The goal was to improve the circuits, to reduce the effort to build the RoeTest and to decrease the need for special parts. Now standard, low cost and readily available parts other than the main transformer and the tube sockets are used.

Also a few more components had to be placed on the test voltage PCBs. Especially the heater voltage PCB is densely populated. A small solder iron and some patience helps when putting it together. The main PCB was laid out in such a way that it can be manufactured as one piece (40 x 20 cm) or as two pieces (2 x 20 x 20 cm).

RoeTest V5 to RoeTest4:

- The G1 and G3 boards are completely redesigned. Instead of a high voltage opamp built with discrete components a high voltage opamp IC is used.
- The G3 board has an additional circuit for measuring high voltages, that is for instance used when testing voltage stabilizers.
- Completely redesigned circuits for the plate, G2 and heater supply boards. Voltages are controlled by an integrated precision opamp, instead of an opamp built using discrete components.
- The USB power is now supplied using a relay, such that no current flows when the RoeTest is switched off. When the RoeTest is switched on power is supplied to the USB chip using the relay to enable the USB interface.
- The 600V plate voltage circuit is redesigned and is put on a separate PCB
- Small layout changes in the other PCBs.
- Use of a different PIC Microprocessor with 12 bit A/D converters (before 10 bit A/D converters were used).
- New firmware.
- The software is modified to support the new design and is compatible with earlier versions.

The new circuit designs allow for more accurate measurement and test results.

Changes RoeTest V6 to RoeTest5:

Circuit designs stayed largely the same. The goal was to simplify the construction of the RoeTest. To accomplish that, the following changes were made:

1. New double-sided PCB. The transformer PCB and connector PCBs are now integrated with the main PCB. Even more wire patches are removed. The only remaining board connections are the mains power, mains transformer, connection to the MOSFETs and to the tube socket station. This reduces the number of hours required to build the RoeTest and reduces the probability of errors.
2. The microprocessor PCB is now also double sided. All the other PCBs remain single sided. There are only a few remaining wire patches required (on the heater board).
3. A chassis is no longer required. Instead, the main PCB and the main transformer can now be screwed to the backside of a stabile aluminum front panel. The front panel can then be put into an applicable cabinet such as an aluminum suitcase or table cabinet.
4. A front panel design is now available, made with Front Panel Designer. If you are not able to or don't want to make your own front panel you can now use the design to order one from Front Panel Express, Inc (the US branch of the Schaeffer company in Germany).
5. The front panel is also used as the heat sink for the MOSFETs. A separate heat sink is no longer required. Small tubes can now be tested continuously, if you are testing larger tubes you'll have to monitor the temperature and if needed wait in between testing tubes or alternatively you can use a fan for extra cooling.

There are a few downsides: with the new large main PCB you have less flexibility in choosing or constructing a cabinet for the tube tester. The large double-sided PCB is probably too complex to make yourself. If you like to make your own PCBs or if you want more flexibility with constructing a cabinet you can still use the RoeTest 5 main PCB designs. The main PCB V5 is compatible with the other V6 PCBs.

Changes V6.2 to V6:

No changes in circuit design. A few PCB layout errors were fixed. Also a few small changes were made:

1. Banana jacks are now mounted on the main PCB. No external wiring needed.
2. The wire connections to the tube socket boxes are now soldered directly on to the bottom side of the main PCB, leading to even shorter connections.

Changes V7 to V6.2

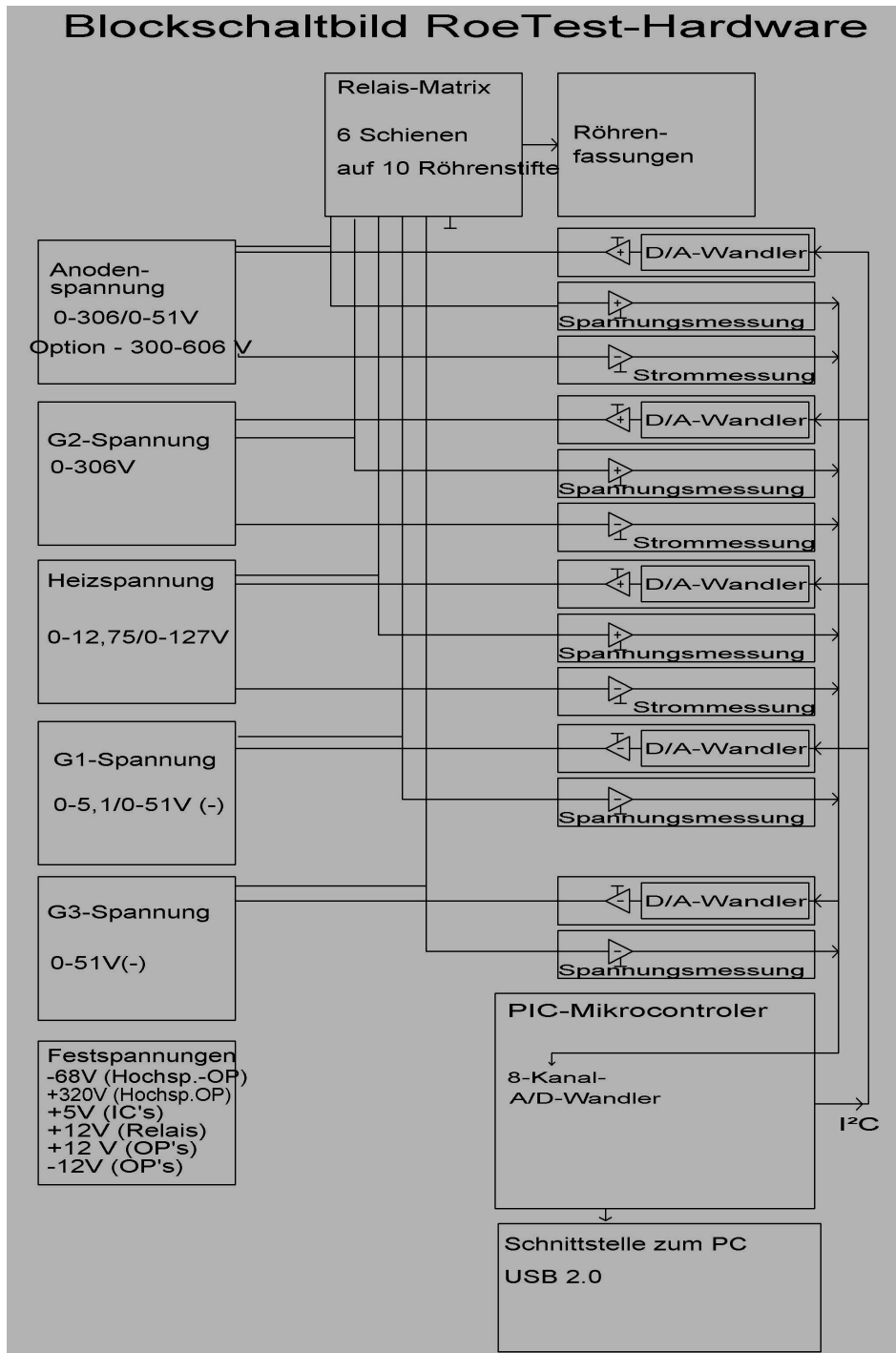
1. Change of layout such that no wire connections to the MOSFETs are needed, the MOSFETS are now soldered on the main PCB. This required a change to the main PCB and the plug-in PCBs.
2. The plug-in PCBs for the heater, anode, G2 and 600V boards now have additional contacts
3. Temperature management: you can now optionally connect a temperature sensor to the heat sink to control a fan. The software can now turn on the fan if the temperature gets too high. The heat sink can be smaller and yet there's more margin.
4. The +320V and -56V voltages are now regulated using a LR8 supertex voltage regulator such that the circuit is now simpler.
5. PCB layout improvements, especially ground connections have been overhauled.
6. New version 7 firmware, compatible with hardware version 5 and higher.

Functional description:

The RoeTest employs a USB interface to connect to a PC. A PIC microcontroller receives commands from the PC software and executes these. The PIC microcontroller controls the output voltages of the heater (H), anode a.k.a. plate (A), G1, G2 and G3 boards and a matrix of relays to connect tube pins to a voltage source. The PIC continuously measures voltages and currents and sends these back to the PC software for display and analysis. The PIC also supports functions such as checking for continuity and controlling external heater voltage relays. To be able to increase the anode voltage there is a 600V board, which is a constant 300V voltage source, connected in series with the anode voltage board.

Block diagram:

The 5 voltage sources can be recognized in the block diagram. The voltage sources and ground are connected to 6 rails. The tube pins are then connected to the rails using a matrix of relays. The relays matrix consists of 10 cards with each 6 relays. Then there are various supply voltage sources that deliver the supply voltages of +5V, +/-12V, -56V, +320V and +12V (unstabilized, for the relays) for the various circuits.



Circuit description:Supply voltages:

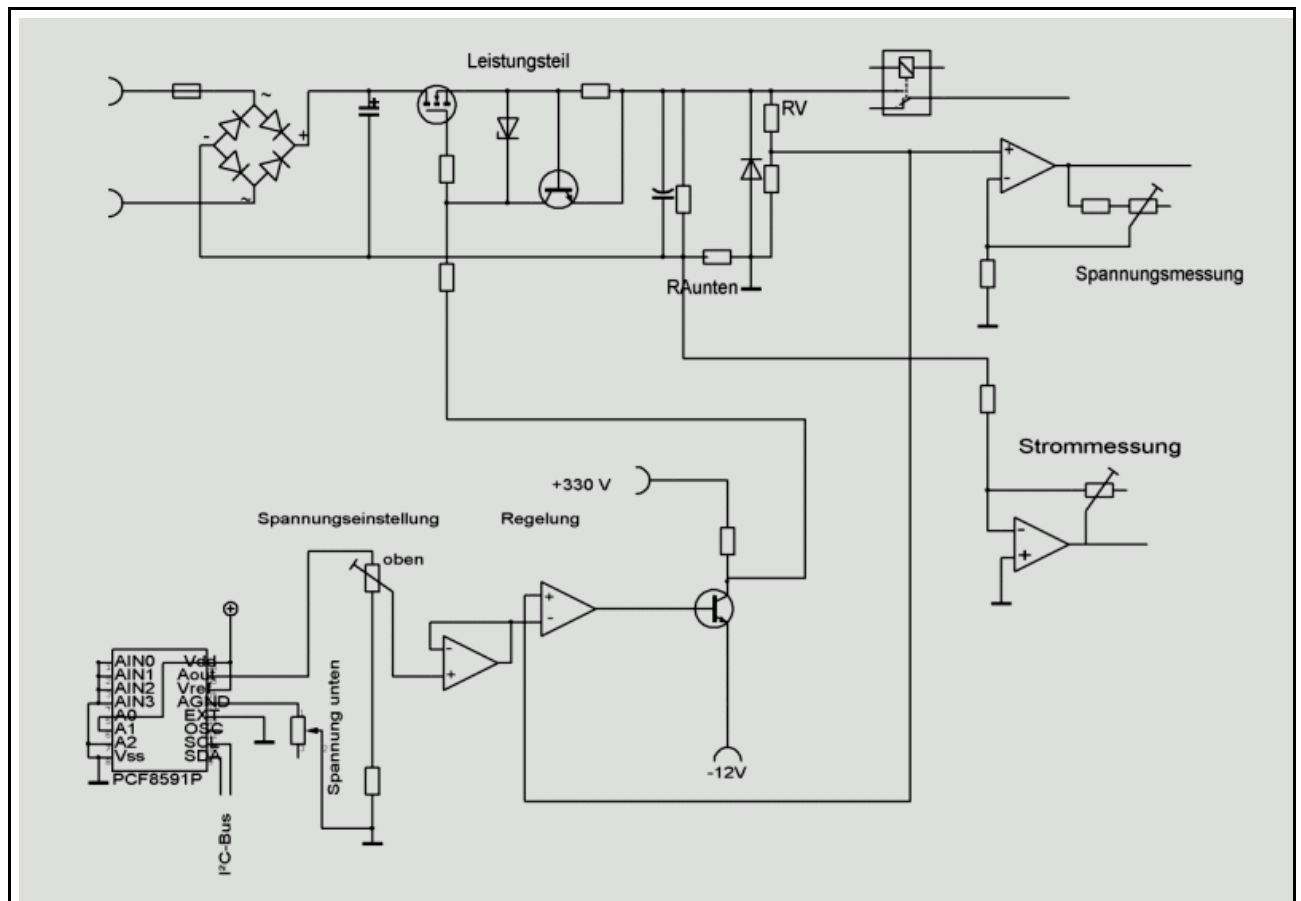
The relay voltage is not stabilized.

The +5 and +/- 12V sources need to be very stable and for that reason precision voltage regulators with low temperature drift are used. The absolute voltage output is less important than the stability since voltage variations would result in measurement errors.

The additional +320V and -56V sources are stabilized using an LR8.

Variable voltage sources for the heater, anode (plate) and G2 voltages:

These all function in a similar manner, as shown in the following simplified diagram:



A D/A converter that is connected to the I²C-Bus is controlled by the PIC and generates a 0 to 5V voltage and that voltage is amplified to over 300V using an opamp and a transistor. The output is connected to a voltage divider around RV and the voltage measured at that point is fed back to a differential amplifier. This differential amplifier compares the voltages on the + and - inputs of the differential amplifier and keeps the output voltage constant (feedback loop). To make the circuit stable many circuit updates needed to be made so the real circuit has quite a few more parts.

Additionally each PCB has a few relays to switch various ranges.

The H, A and G2 circuits use MOSFETs for increased power output.

Finally the H, A and G2 circuits have a current limiter circuit: if the voltage drop over a resistor exceeds about 0.6 volts, a transistor is switched on to decrease the MOSFET output voltage such that the output current is limited.

Here is some more detail, please refer to the actual circuit diagrams (these are on a CD-ROM can be obtained from me):

A voltage divider circuit is used to measure the voltage outputs of the voltage boards, and that voltage is fed into an opamp whose output is connected to an A/D converter input on the PIC microcontroller.

H, A and G2 output current is measured by measuring the voltage drop over resistors in the ground connections between the input voltage sources and central ground. For that, current is supplied by separate transformer circuits. Opamps are used again as amplifiers for the measured voltages and the output voltage is connected to an A/D converter input on the PIC microcontroller (see detailed circuit diagrams).

Board component dimensions and layout are different for each board. Where needed, relays are used to switch measurement ranges. This allows for high resolution even in the low ranges for voltage output control and output voltage and current measurement. For the heater circuit low-high ranges, also the transformer circuit and current limiter resistors are different.

Each board has a relay to connect the output to a relay matrix rail. The G2 board can also be connected over two 470K resistors to a rail, for testing magic eyes. The G1 board can be connected to a rail over a 1.2M resistor for tube vacuum testing. Relays are controlled by the PIC microcontroller by using relay drivers on the PIC PCB or for the relay matrix cards by relay drivers on the relay matrix PCBs connected to the I²C-Bus.

LEDs and discharge resistors are used to discharge the electrolytic capacitors when the tube tester is switched off. The LEDs indicate whether the resistors are still charged with a high voltage. For safety reasons you should only work on the tester when all electrolytic capacitors are discharged. The main PCB has various connection points that allow you to quicker discharge these capacitors e.g. using a light bulb, or that allow you to measure the voltage levels to verify the capacitors are discharged.

A high voltage opamp (OPA445A) is used for the G1 and G3 boards. MOSFETs are used to increase output power. See detailed circuit diagrams.

The G3 card also has a circuit to measure high voltages, and the circuit is active when the G3 card is not used for generating the G3 voltage. This is used e.g. when testing voltage stabilizers.

Boards:

There is a PCB for each of the 5 voltage sources. There is also a PCB for the PIC microcontroller (with relay drivers) and a PCB for the 600V range. Finally there are 10 PCBs for the relay matrix, each with relay drivers.

These PCBs are connected to and inserted in the main PCB.

The main PCB has the following circuits:

- Supply voltages
- Fuses and rectifiers, and the filter capacitors for the H, A, G2- voltage and 600 V boards
- Continuity checkers
- Relays to switch to an external heater voltage source
- USB interface
- Soft start for the toroid transformer using an NTC and relay.

Component selection:

I recommend using only first class components. If you're using second class components you should expect second class quality.

For tube sockets I recommend you use socket boxes and not to have fixed sockets.

Transformers:

For +/- 12-V, relays and other voltages normal standard transformers are used.

For the -56V supply a 2 x 24V transformer will do. Since the load is minimal, the idle (no load) voltage is high enough.

The transformer for the +320V with a 250V secondary is not commercially available. A special made transformer would be too expensive so the solution is to use 4 standard transformers in series. Also here the no load voltages are sufficient..

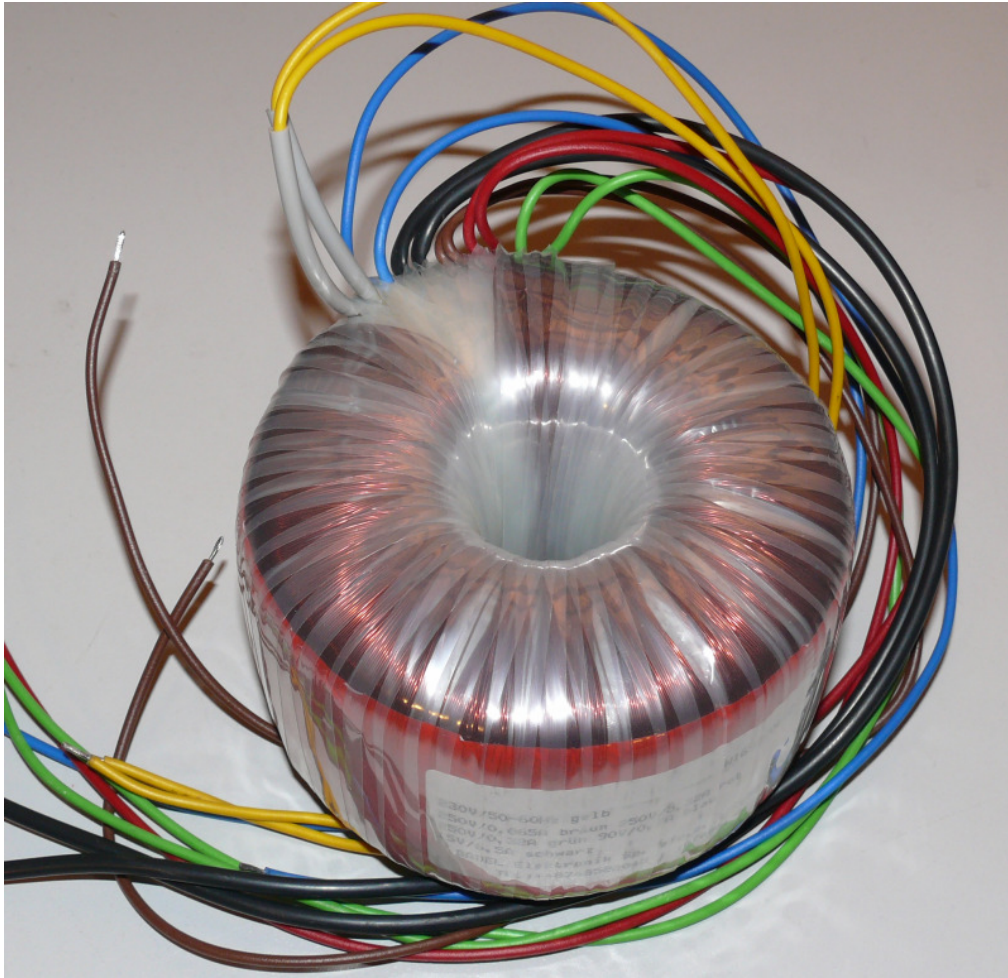
The only special made part is the main transformer.

I recommend the use of a toroid transformer. Toroid transformers are compacter, lighter and more important, the difference between the output voltage under no load and full load is less!

You'll find a table with the toroid transformer data on the CD (Trafodaten - RoeTest ... pdf).

It is important that the transformer windings are able to deliver the specified voltages under full load. The no load voltage however should not be too high, to protect the filter capacitors and to make sure power loss is not too high. The transformer must physically fit in the space that is planned for it.

Since having a transformer made to specification is very expensive I've had someone manufacture a small supply. If you're interested, contact me.



Main transformer = toroid transformer (made to specification, with long wires)

For resistors please use metal film resistors or wire wound high wattage resistors where needed.

Trimmer resistors should be good quality multi-turn trimmer resistors.

There is a **components database** application for the RoeTest that can be downloaded or that can be found on the CD-ROM (directory Bauteileliste_Components). It lists all the parts and components needed for the RoeTest with the Reichelt order number (if available). You can print lists of components to order or you can save them as a csv file that can be uploaded to Reichelt (note: that import doesn't always work properly). You are responsible for ordering parts. I cannot be held responsible for wrongly ordered parts. There is a manual for the parts database application on the CD-ROM (Bauteile – Hilfe.pdf). The database has cost price information although you will need to verify and update the cost price information since these prices are constantly changing.

You can add and delete components to/from the database, which may come in handy if you already have some components or want to make the PCBs yourself.

PCBs:

The double sided main PCB with many vias connecting different layers is very complex. It is also very large at 430 x 230 mm. This kind of PCB can hardly be made at home anymore. It will also too expensive if everyone just has to have one made with a quantity of one. For that reason and also to protect my design I no longer distribute the main PCB design files. You will find the parts layout (PCB silk screen design) on the CD-ROM, which you can use if the silk screen print becomes unreadable.

I have ordered a larger quantity of the PCB set, and you can buy them from me at a fair price.

These are professionally made PCBs:

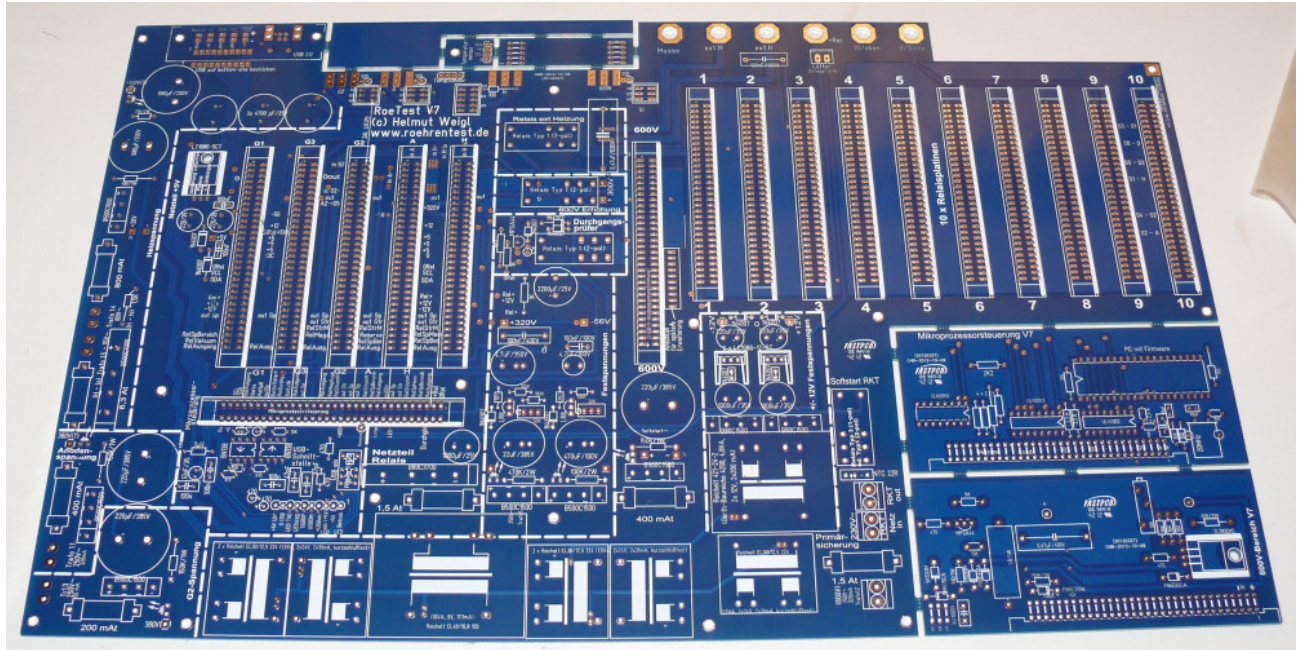
- **2 mm Epoxy FR4**
- **70 μ copper thickness**
- **Double sided with vias where needed to connect layers**
- **Solder mask on both sides**
- **Silk screen print indicating component location on both sides**
- **HAL-pre-tinned for easy soldering**

This table lists how many PCBs you need:

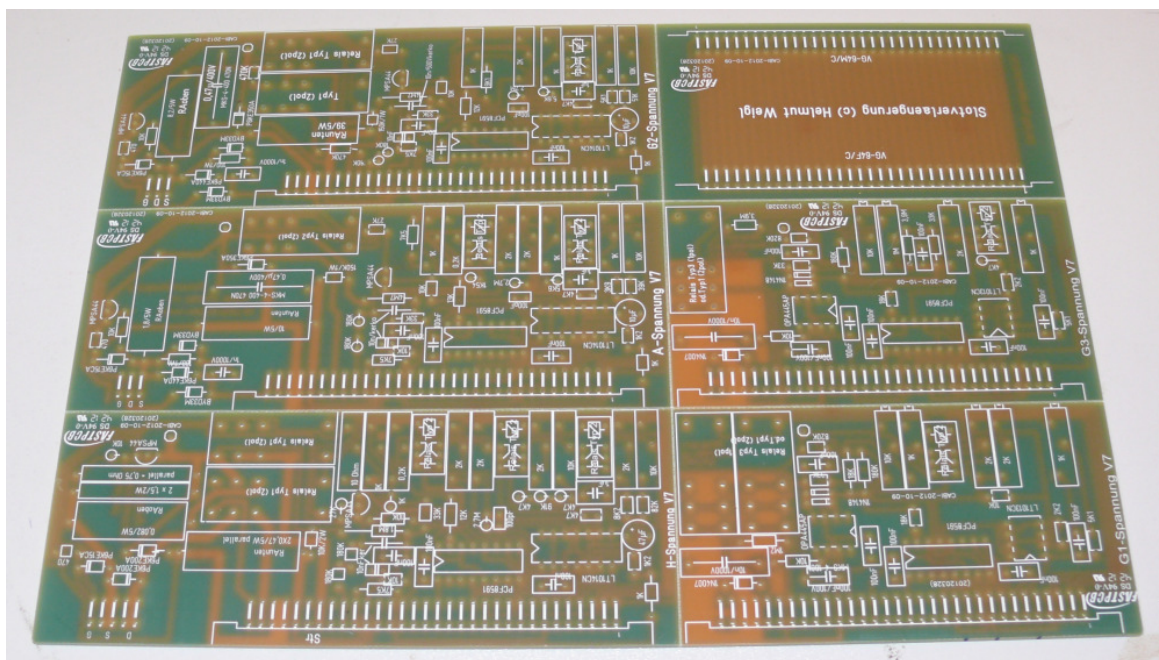
Application	Number
Main PCB	1
Relay matrix PCBs	10
Microcontroller	1
Voltages (A, G2, G3, H, G1, 600V)	6 (1 of each)
Temperature sensor (optional)	1

And this is what professionally made PCBs look like. Silk screening, solder mask and pre-tinning speed up component placing significantly.

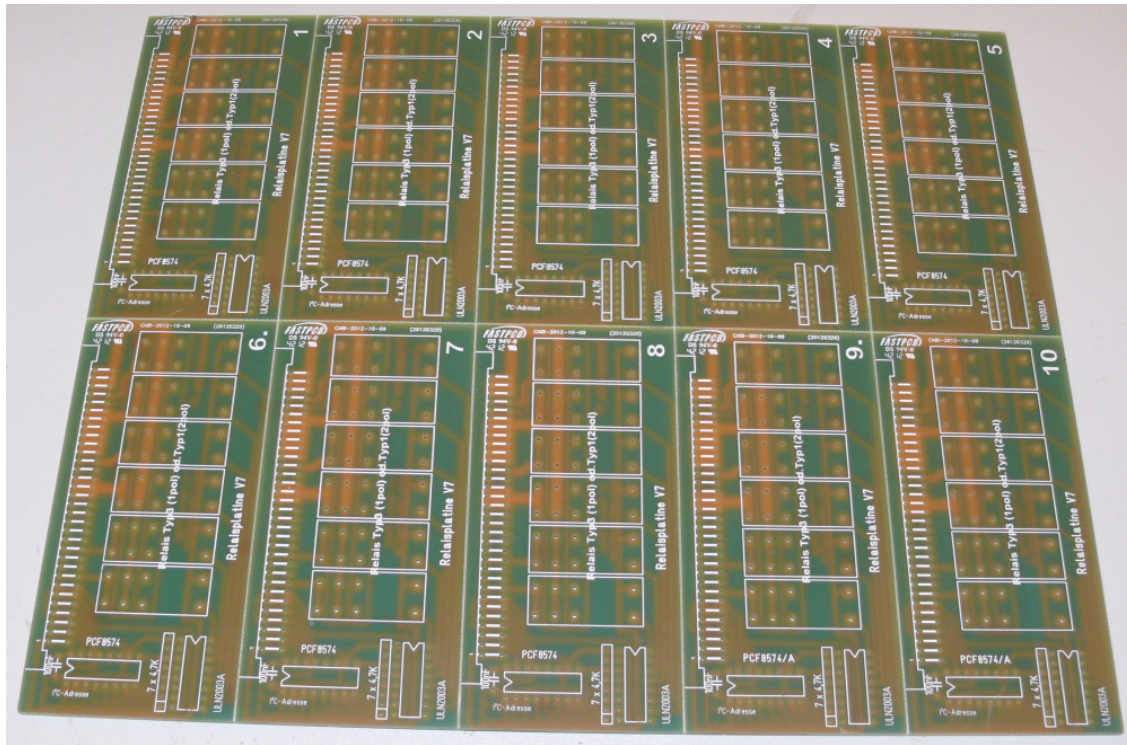
No wire bridges are required thanks to double sided layouts with vias to connect layers. This is true for the main PCB, microcontroller PCB, the 600V PCB and the temperature sensor PCB. The main PCB, microcontroller PCB and 600V PCB are manufactured as a whole, with perforations to separate the PCBs. Use a small saw to separate the PCBs and use a rasp to smoothen the sides. The PCB size is 430 x 230 mm!



Voltage PCBs: despite single sided layouts no wire bridges are needed. You can separate the PCBs for instance by bending them on a table edge right at the PCB edges.



And finally the relay matrix PCBs. Also here despite single sided layouts no wire bridges are needed. You can separate the PCBs for instance by bending them on a table edge right at the PCB edges.

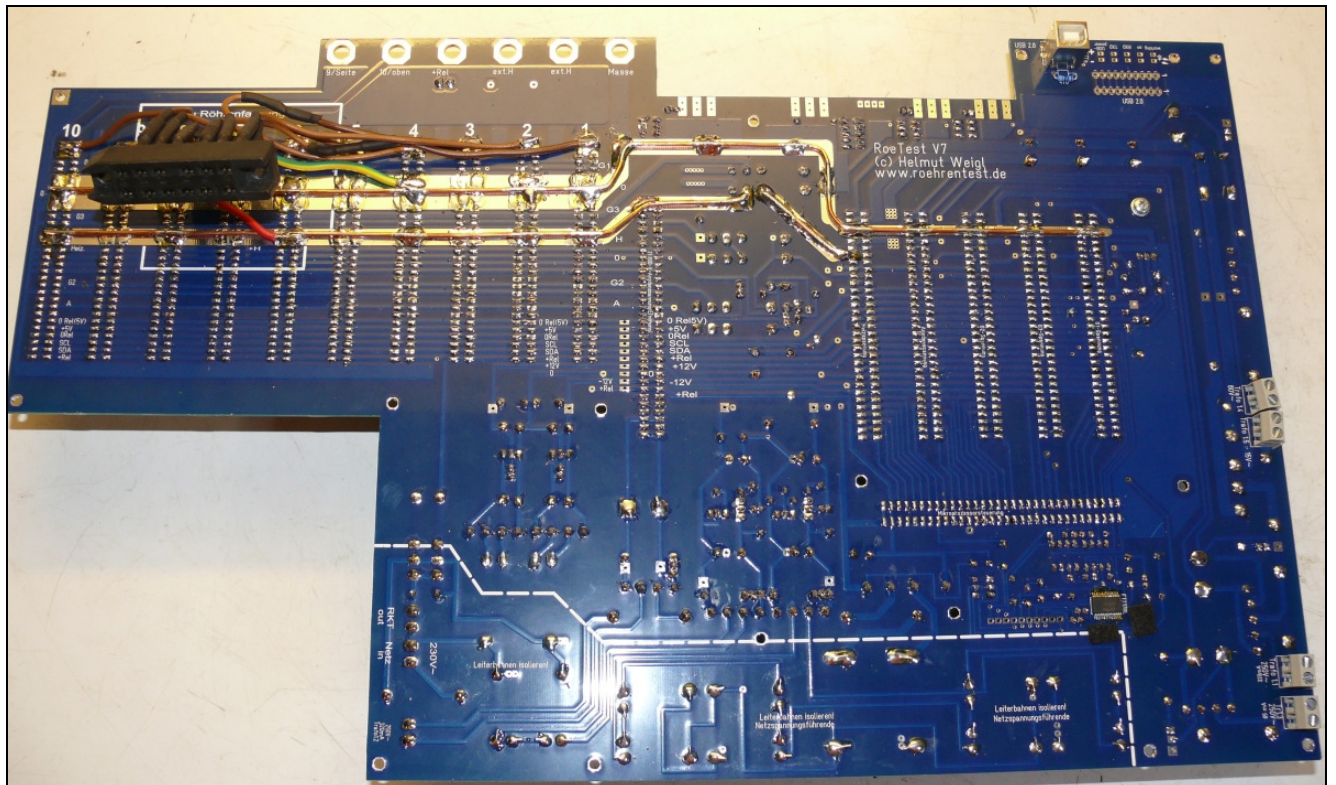


PCB construction:

Main PCB

Please isolate the sections of the main PCBs that have traces directly connected to the mains, so you are protected from accidentally touching them.

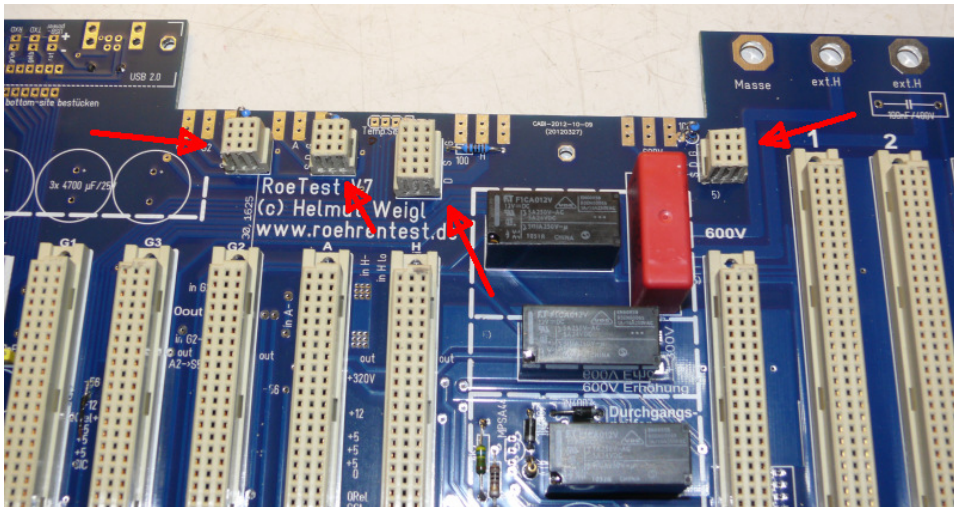
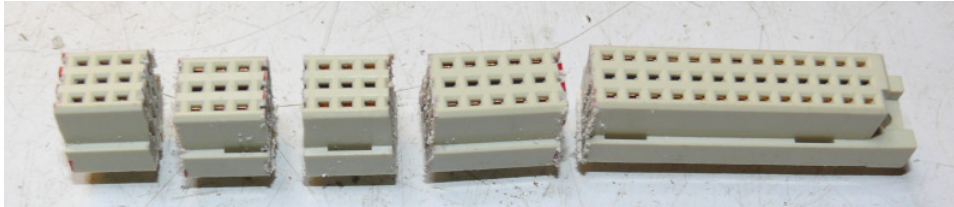
The ground and heater current traces should be reinforced by soldering a 2,5 mm² wire onto them, to minimize voltage drop. For this purpose, no solder mask is put on these traces. See picture below.



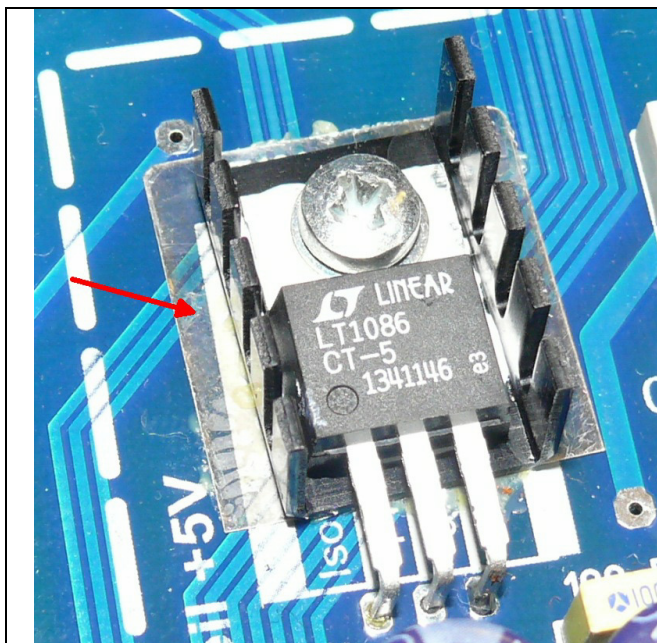
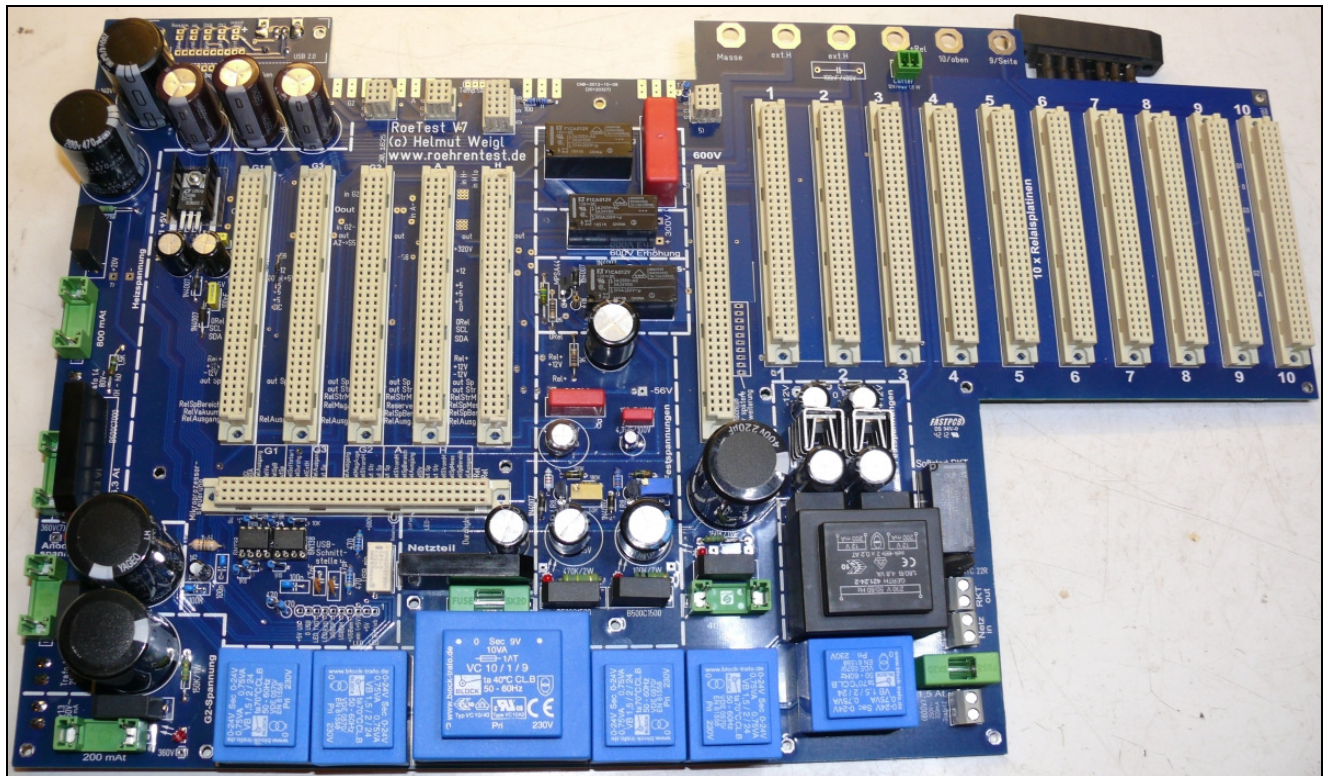
At the right, you can see the 4 transformer connectors, on the bottom side of the main PCB.

Make sure you correctly orientate the 64 pin 4 mm PCB DIN connectors on the main PCB!
(fat lines indicate where the PCBs have to be plugged in).

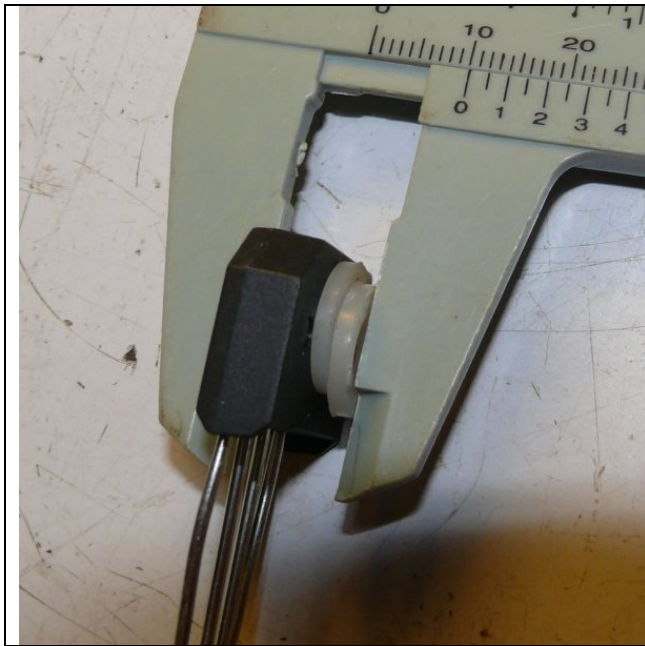
Additional connectors are needed for the A, G2, H and 600V PCBs and you can make these by using a saw to cut up a 64 pin 4mm PCB connector as shown below. You may have to glue the bottom piece back on on the separated pieces:



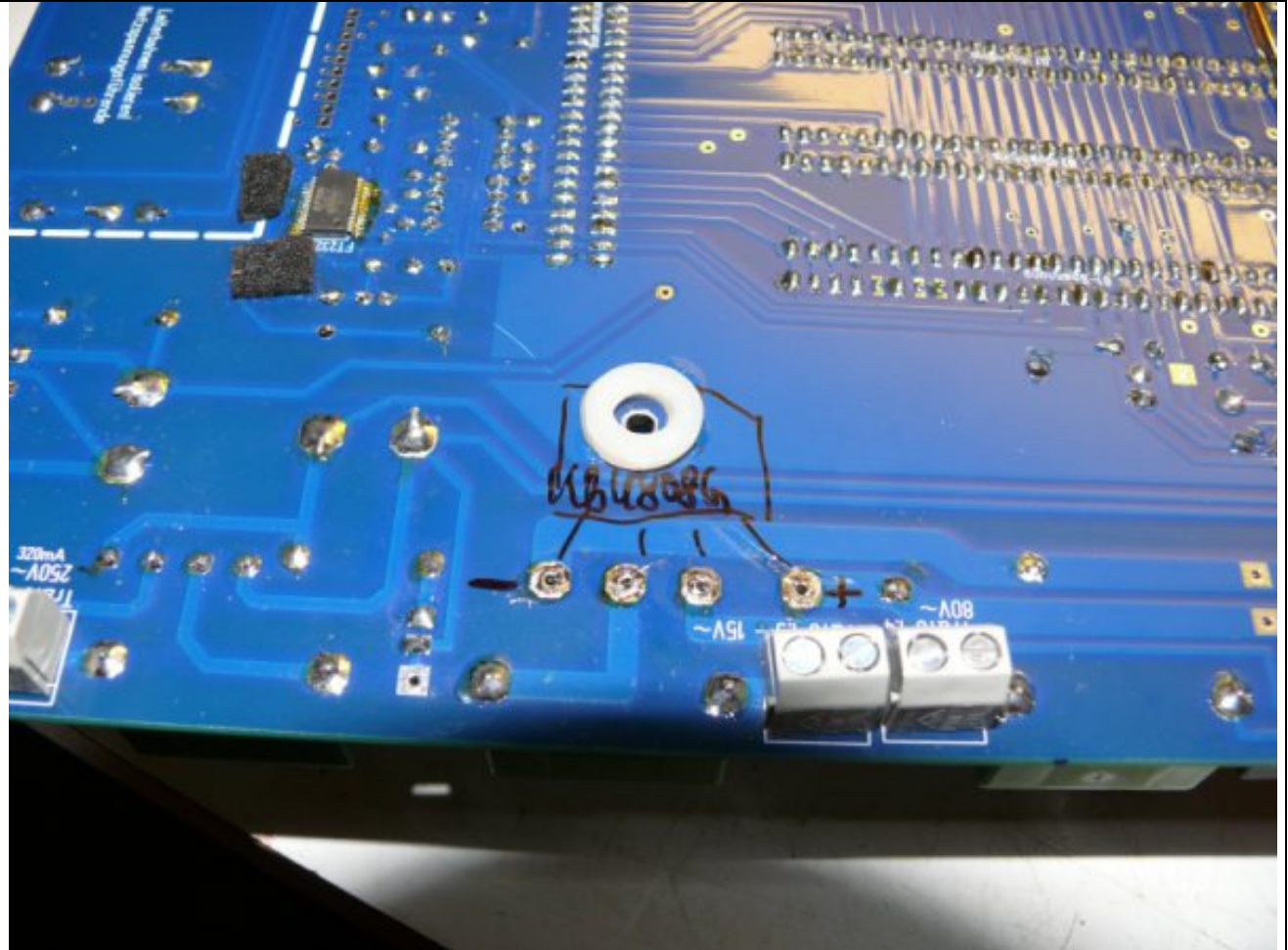
Picture of the V7 main PCB: an industrially made PCB has many advantages over home made PCBs: silk screening, solder mask, double sided with vias, exact fit...all this just saves many hours of work, making it a good investment.

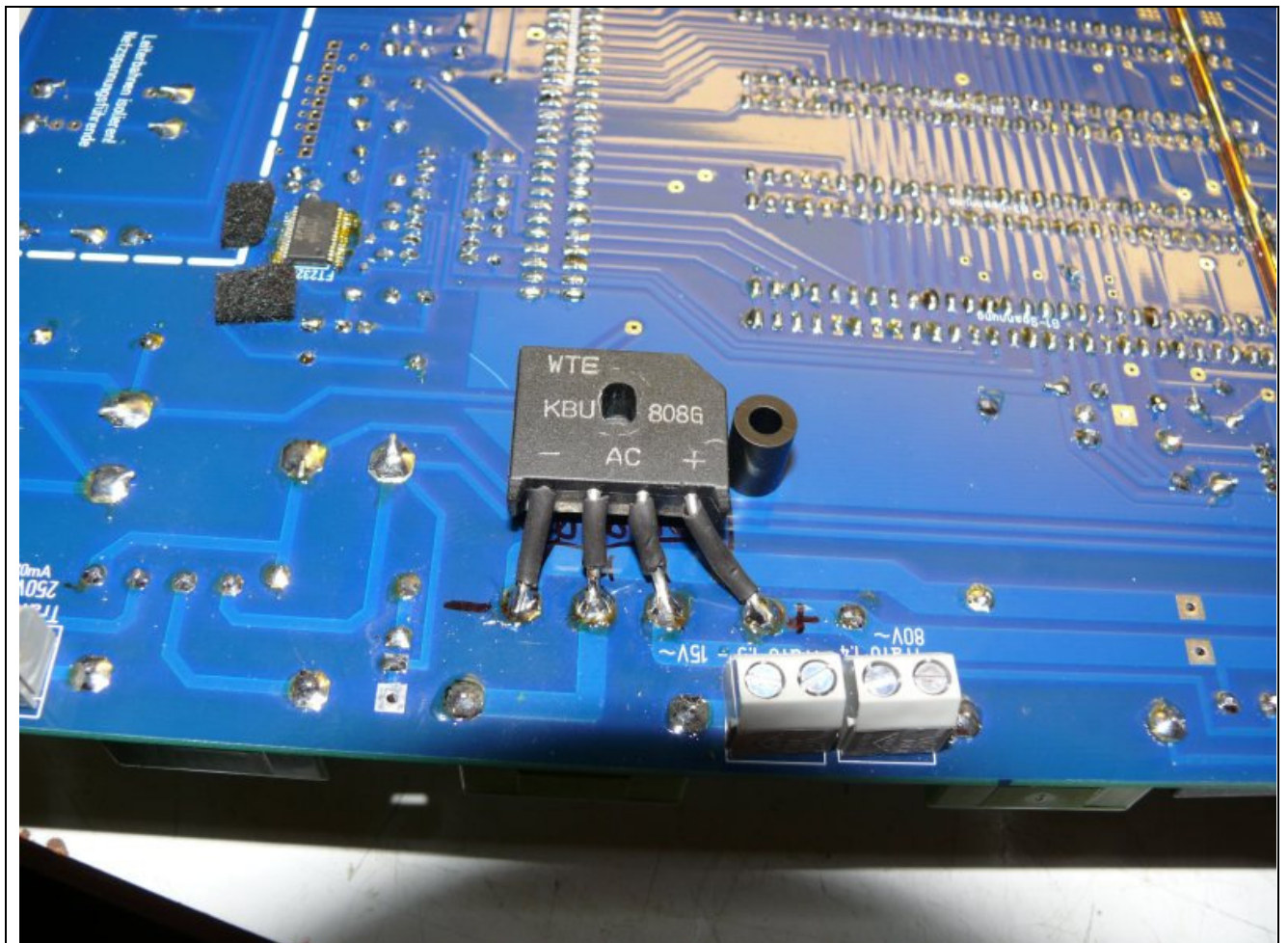


The heat sink for the 5V voltage regulator must be insulated from the main PCB to avoid a short circuit between the heat sink and the traces underneath it. I have used a transparent plastic sheet and glued it between the traces and the heat sink as shown here.



The bridge rectifier for the low heater range (5A) is not a ~~B500C700~~ as indicated by the silk screening / diagram and is replaced by **KBU808G**. This rectifier can deliver up to 8A if a heat sink is used. The front panel is used as the heat sink for this rectifier. For this purpose the component is soldered onto the bottom side of the main PCB. The distance between the main PCB and the front panel is 10 mm. The bridge rectifier is not that thick so plastic rings are put between the rectifier and the PCB as shown to the left. Please use a little heat sink paste (aka thermal compound) between the bridge rectifier and the front panel. A bolt is used to press the rectifier on to the front panel.





The 10 mm spacer is shown as a comparison. At this location you don't need to use a spacer since the plastic rings and the bridge rectifier are effectively used as a 10 mm spacer here.

USB interface:

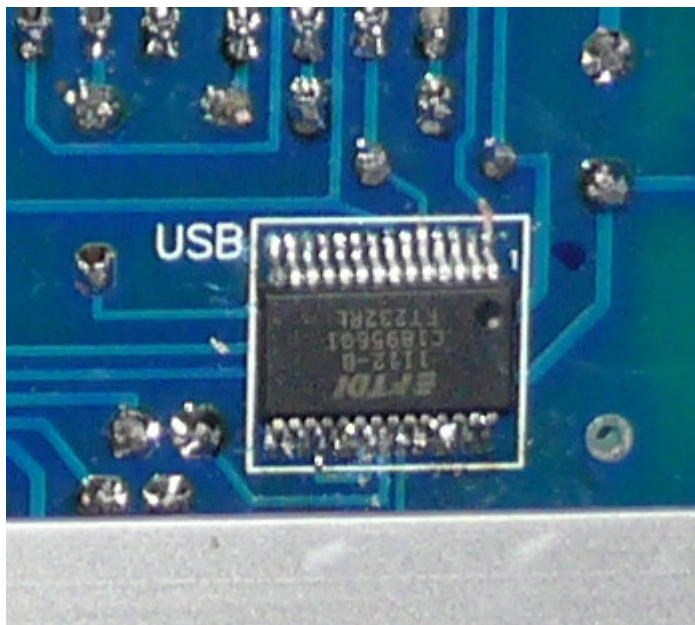
The RoeTest is connected to a PC using a USB interface. The USB interface is on the main PCB. The USB connector and the LEDs are at the top of the main PCB. If you want these at a different spot, you can separate that section of the PCB from the main PCB and connect them using connectors and wires. Please keep the connections as short as possible.

Important note: **the USB interface components such as the USB socket, the LEDs and a resistor are soldered onto the bottom side of the main PCB**, such that they stick out a little. The distance between the main PCB and the front panel is 10 mm (10 mm spacers are used).

These days USB is the standard for serial interfaces. USB 2.0 is used, the old USB 1.1 is too slow. Both your PC and operating system must support USB 2.0 (Windows XP service pack 2 or higher).

Soldering the tiny USB to RS232 converter IC FT232RL is a tricky job. This part is only available as an SMD part. It must be soldered on the **bottom** side of the main PCB. You will need a very fine solder tip and a steady hand. Lots of light and a magnifier are also very helpful.

Put a tiny amount of solder tin on the PCB connections. Then place the IC into position and hold it there with one finger and solder one corner pin. Then solder the opposite corner pin and then the remaining pins. You'll want to keep a desoldering pump handy to be able to remove excess solder.



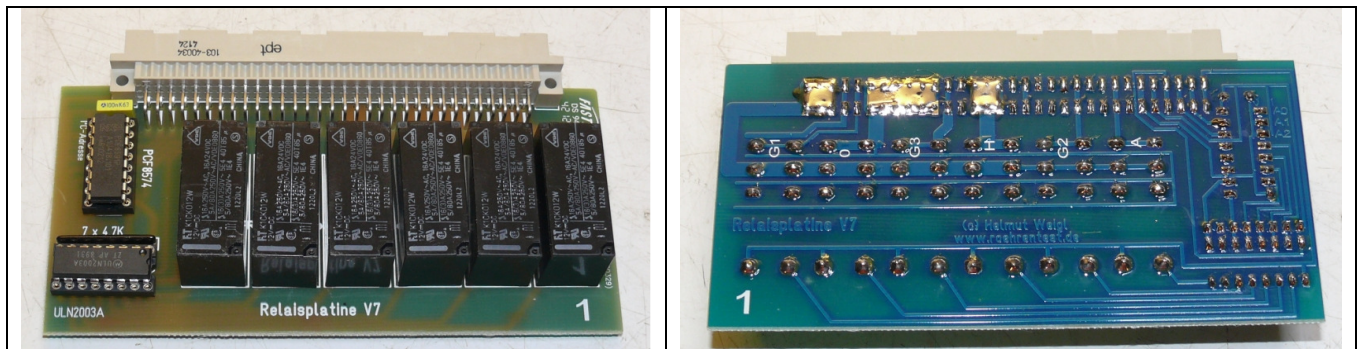
Windows needs a driver for the USB interface. The most recent drivers can be found on the FTDI web site. You can also find a driver on the CD-ROM. Before you connect the RoeTest and switch it on you must have the driver unzipped. After connecting the USB interface Windows will report that it found a new USB interface and wants to install a driver for it. You can then indicate where (which directory) the driver is to be found. After successful driver installation the interface will report itself as a serial interface.

Relay PCBs

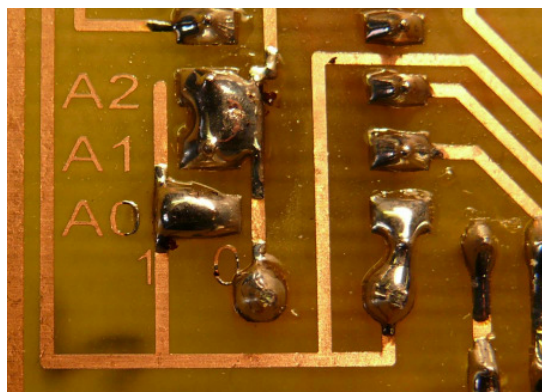
- 10 of these need to be built (one PCB for each tube pin)

The I²C addresses hardware programmed – you do that by connecting the address pins of the correct **IC types** (PCF8574 or PCF8574A) to either +5V (= 1) or 0V (= 0) as per the table below:

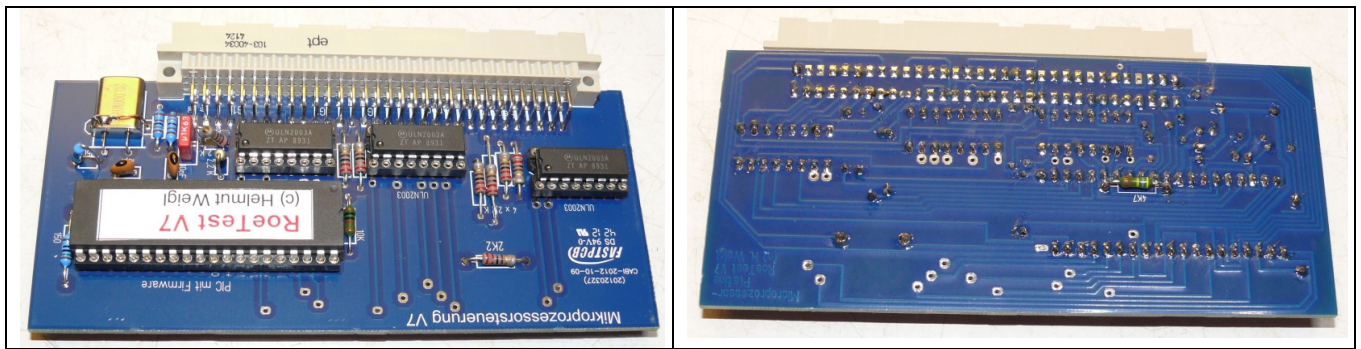
Tube pin/ PCB nr.	IC-Type	I ² C-address	A0 Pin1(IC)	A1 Pin2(IC)	A2 Pin3(IC)
1	PCF8574	64	0	0	0
2	PCF8574	66	1	0	0
3	PCF8574	68	0	1	0
4	PCF8574	70	1	1	0
5	PCF8574	72	0	0	1
6	PCF8574	74	1	0	1
7	PCF8574	76	0	1	1
8	PCF8574	78	1	1	1
9	PCF8574 A	112	0	0	0
10	PCF8574 A	114	1	0	0



Below you can see home made relay PCBs and the bridged connections (using solder) to make the addresses for the PCF8574(A) chips as per the table above. **If you are using the ready made PCBs ordered from me you will not need to do this – the connections are already made as part of the manufacturing process.**



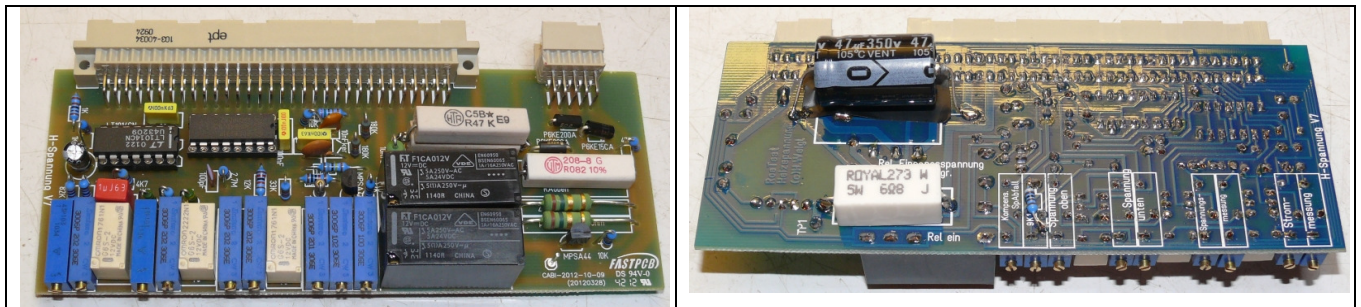
Microcontroller PCB



Microcontroller PCB RoeTest

This PCB is double sided with vias to connect the layers. On the bottom side you will need to solder a 4.7K resistor.

Heater voltage PCB

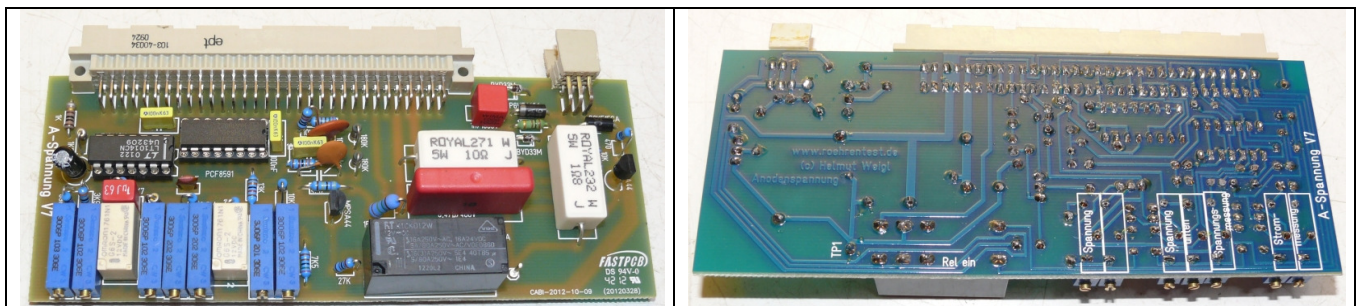


The measurement resistor RV low is constructed by soldering **two parallel 0.47 Ohm/5W resistors** in place on the PCB bottom side that together make a 0.24 Ohm/10 Watt resistor.

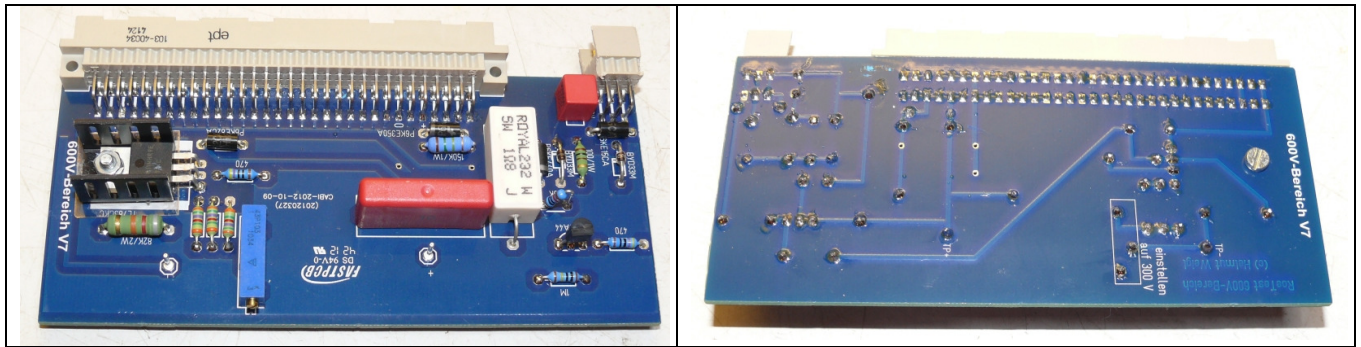
Also the following components need to be soldered on the PCB bottom side:

- 6,8-Ohm/5W resistor
- 9,1K metal film resistor
- 47µF/350V electrolytic capacitor (as indicated on the bottom side silk screening)

Anode (aka plate) voltage PCB



11) 600V range PCB



If you do not want to use the 600V range for anode voltages you do not have to insert/build this PCB. It generates a fixed 300V, that is switched in series with the anode voltage PCB using a relay. The heatsink must be isolated from the PCB to avoid short circuits e.g. using a plastic sheet.

Note: the spread in the voltage regulators is such that sometimes the trimmer resistor is all the way at the end and you're still not getting 300V. In that case replace the 3x36K resistors with a lower value to decrease the voltage or increase the value to increase the voltage.

12) Temperature sensor PCB



This PCB must be mounted to the heat sink in such a way that the LM75 IC touches it. I recommend the use of thermal compound between the IC and the heat sink.

Very important: please always be sure to insert each PCB into the correct connector slot on the main PCB and don't put them in the wrong positions as this will destroy the circuits. Don't insert or remove PCBs when power is switched on!

Socket box receiver – mechanical construction:

Tube sockets are not built in permanently, instead a socket box receiver is used so that individual socket boxes can be connected.

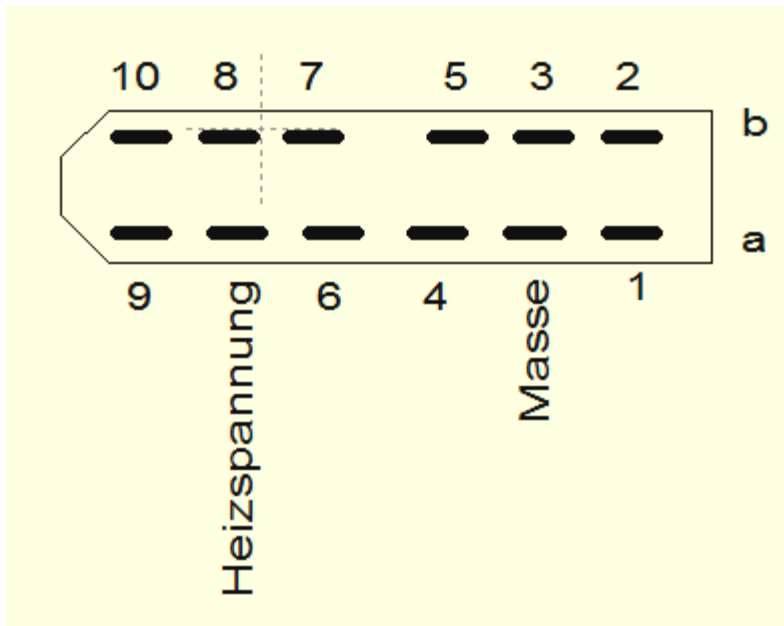
Advantages:

- Less risk of oscillations
- flexible, it's easy to add socket boxes for new tube socket types

Recommendation:

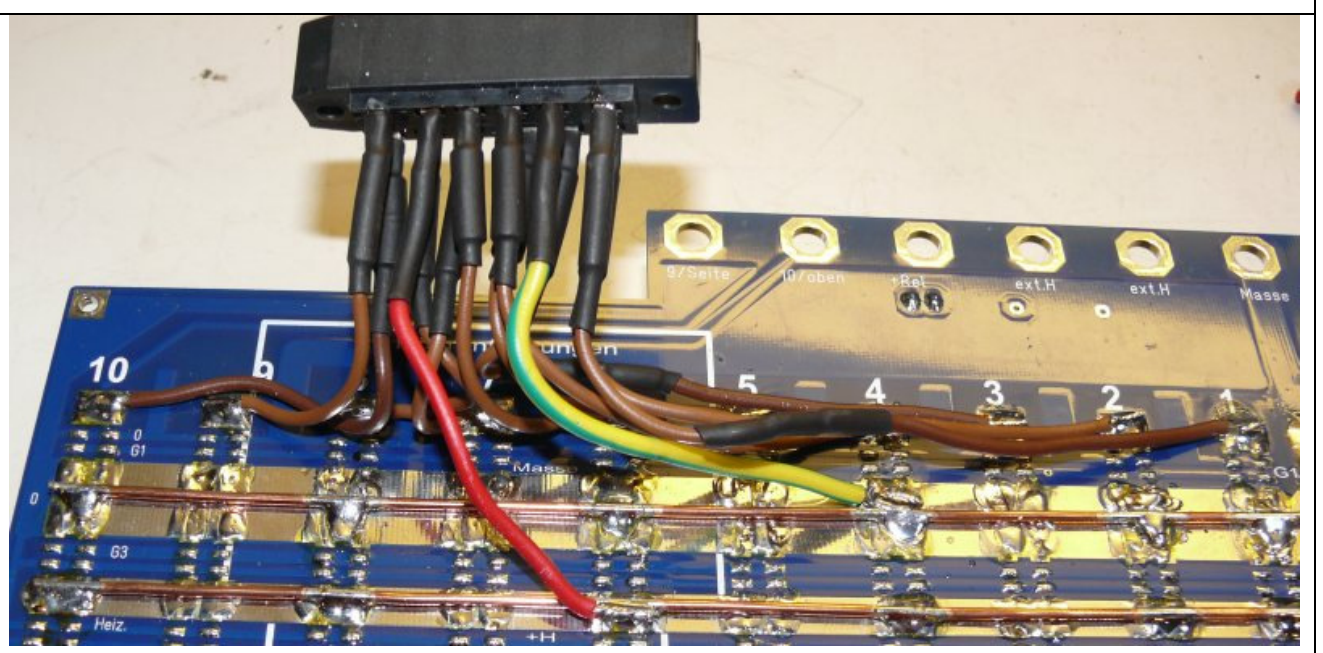
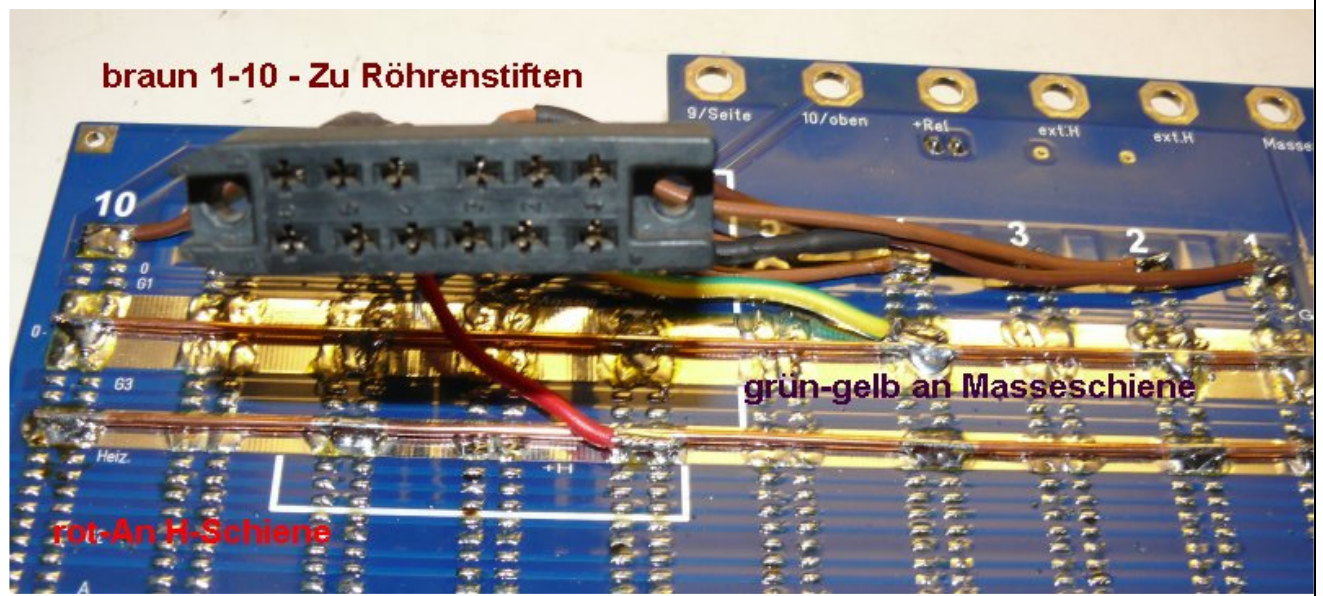
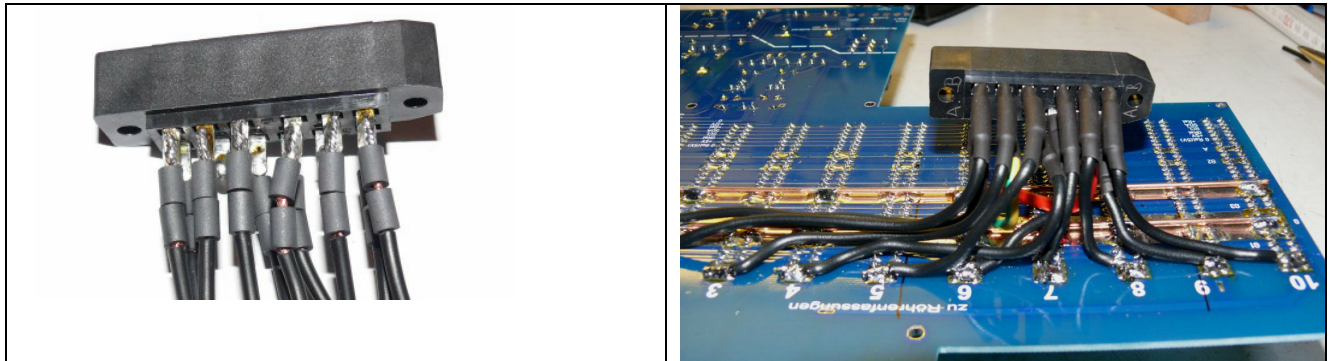
- use good quality connectors
- make sure to select an appropriate wire size for the current
- use wire with good isolation
- **keep wire connections as short as possible**
- use RF ferrite beads or RF chokes when soldering the wires to the tube sockets and connectors especially for anode connections
- make sure all metal parts like screws or nuts are connected to ground (for safety reasons)
- the best is to have socket boxes with only one tube socket

I recommend 12 pin DIN41622 female connectors (*Reichelt FL B12, DIN41622*). 10 connections are used for the tube pins, one pin is used for ground and the remaining pin is connected to the heater voltage.



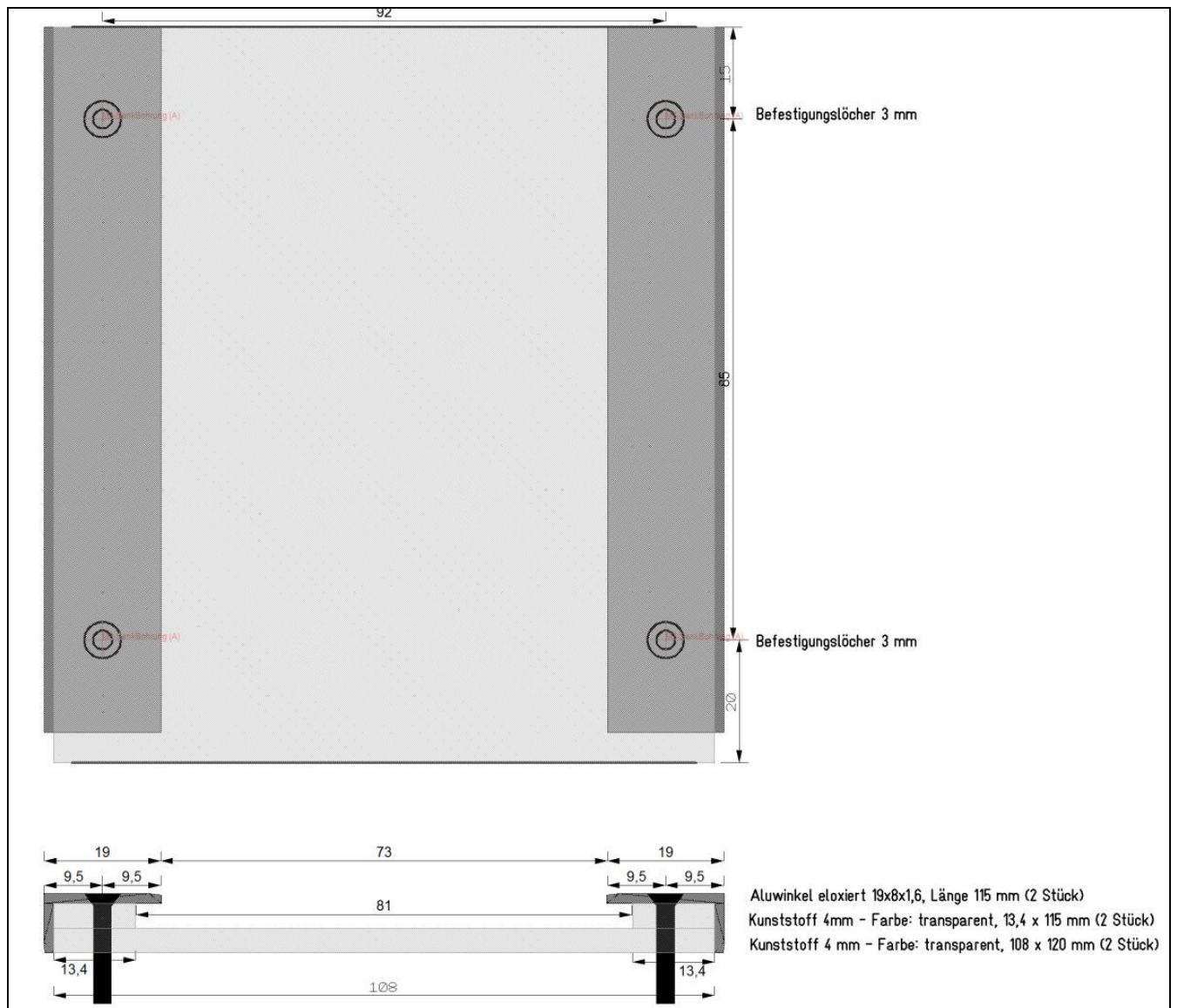
Connection diagram for the DIN 41622 connector (seen from front on socket box receiver)

Connector for socket box receiver: use 2 ferrite beads for each of the 10 tube pins and use heat shrink tubing to cover/isolate them. Don't use ferrite beads for the ground and heater connections. The connector itself is bolted to a metal (aluminum) angle and a plastic box is used to cover it all. This has to be screwed to the front panel from above since you can't get to the bottom side anymore once the PCB is mounted to the front panel. The front panel must have M3 threaded holes for that.





The socket box receiver is made from plastic (clear acrylic or similar) and an aluminum angle and has two rails to guide the socket boxes. Make sure that the bolts are not too long and touch the PCB or the main transformer. Make sure to bolt the main transformer to the frontpanel before bolting on the socket box receiver.




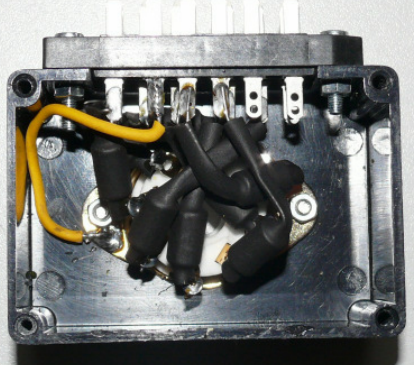


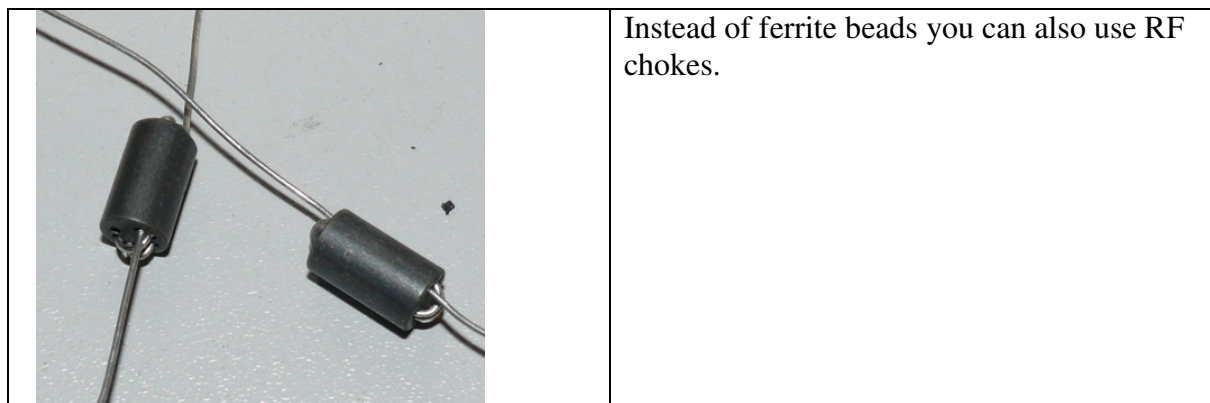
socket box receiver



socket box receiver components

Here are some examples of the tube socket boxes I'm using from RoeTest V7 onwards: the low cost plastic boxes get new bottom plates that is wider as the box itself (in my case it's 80 mm wide) so they fit within the rails of the socket box receiver.

	
	<p>It's best to have only one tube socket per socket box. That way wiring can be kept simple and there is less chance for oscillations.</p>
	<p>All pins that could be used as an anode connection must have ferrite beads. Make sure they don't touch each other (heat shrink tubing). As you can see here, also here keep wire connections as short as possible.</p> <p>Make sure to connect any metal parts that can be touched to ground.</p>

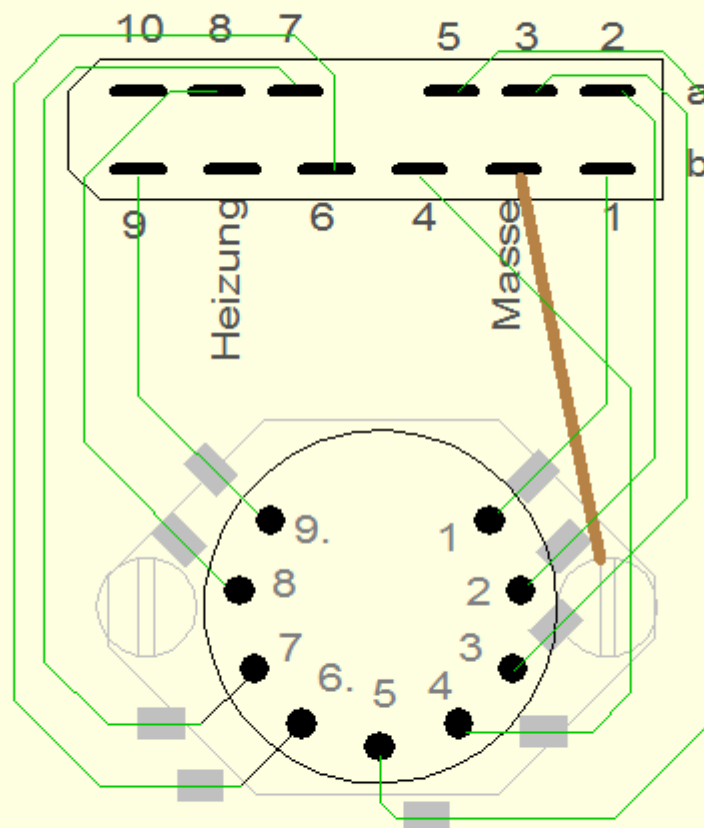


Instead of ferrite beads you can also use RF chokes.

Schaltplan Fassungsbox

Beispiel Noval-Fassung

(von vorne auf Fassungsboxhalter gesehen) Reichelt ML A12 (DIN41622)



Masse an Metallteile

Ferrite vor Fassung, falls Anoden

Heizung wird nur für Spezialzwecke verwendet (z.B. Compactron-Fassung)

Ferrite dürfen sich nicht berühren (Schrumpfschlauch)

Large socket boxes: Reichelt GEH KS 50

Small socket boxes: Reichelt GEH KS 35

Male connector (DIN41622): Reichelt ML A12

Very important:

Make sure to use ferrite beads (or even better, RF chokes) when soldering the wires to the tube sockets, as a minimum for each pin that can be connected to a tube's anode. You don't need ferrite beads for the other tube connections.

Make sure to use appropriately sized wire for the heater connections since heater currents can be quite high. To have an overview of which pins could be connected to anode or heater I created a table for the most common tube socket types – see the "**Socketübersicht.xls**" file. Make sure the ferrite beads don't touch each other causing shorts – use heat shrink tubing.

Banana sockets are mounted to the front panel for:

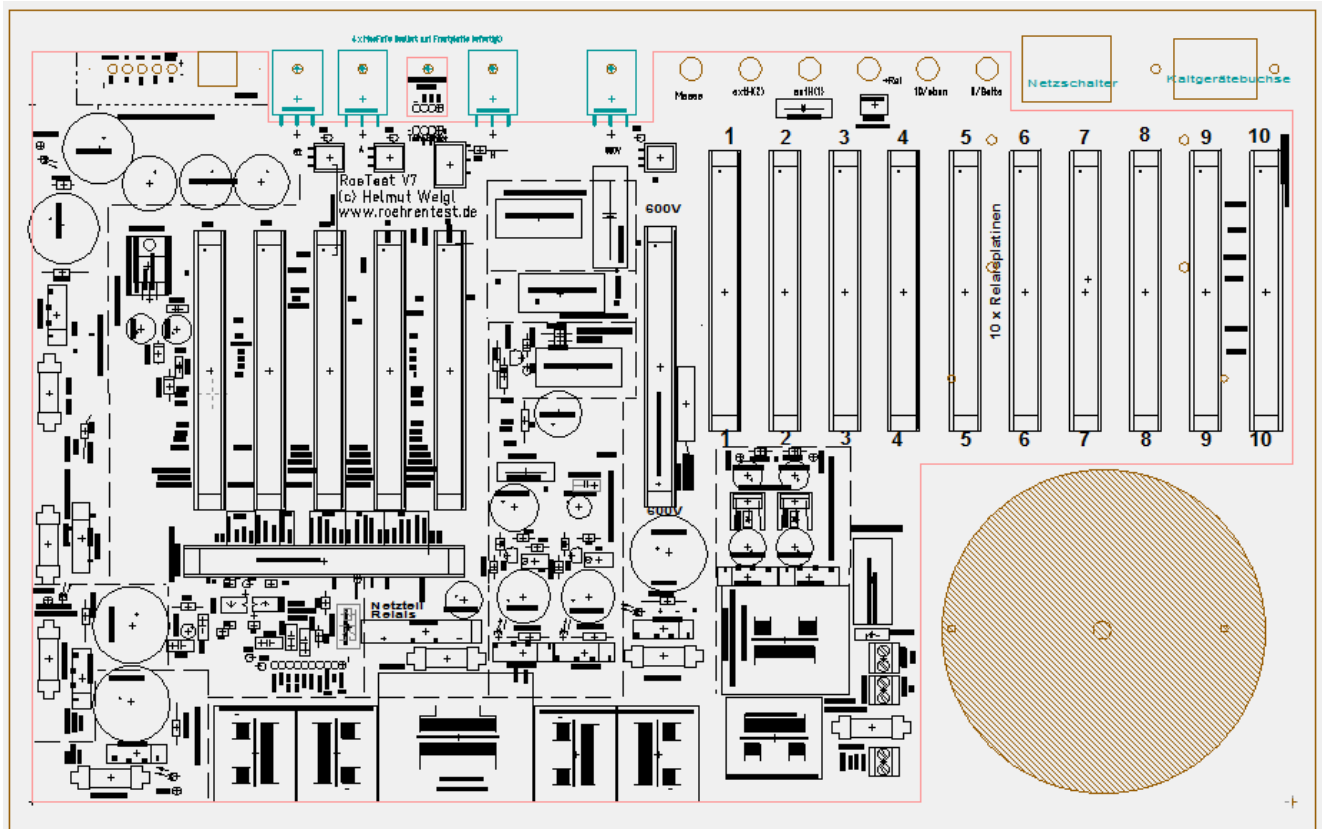
- Pin 9 and 10 – these are used for making connections to tubes with top connectors and for voltage regulator tubes
- ground
- unstabilized relay voltage (+12V)
- 2 connections to be able to connect an external heater voltage source.

Housing/cabinet:

A chassis is no longer required because all parts can be bolted on to the front panel and that front panel can also be used as the heat sink for the MOSFETs (that need to be electrically isolated from the front panel). Therefore 5 mm or thicker aluminum panel should be used for the front panel. On the CD you can find a front panel design made with the Front Panel Designer software that can be downloaded for free from Front Panel Express Inc (<http://www.frontpanelexpress.com/>) in the US or from the Schaeffer AG company in Europe. With the Front Panel Designer you can modify my front panel design and order it online from Front Panel Express or Schaeffer. Of course if you have the tools you can make the front panel yourself.

.

The front panel with all electronics bolted to it can be put in for example an aluminum suitcase or some other suitable cabinet. Make sure you know the dimensions of the suitcase or cabinet you want to use before ordering the front panel. You can make the front panel larger but not smaller.



Front panel seen from behind with main PCB and the spot for the toroid transformer. The sockets for the required connections (USB, banana sockets, mains switch etc.) are on the front side. You can also see the locations of the MOSFETs and the temperature sensor.

The above is just a recommendation, obviously you can use a chassis or put the whole thing together in another way.

Whatever you use, it should be some metal housing that is connected to ground for safety reasons and also to avoid RF radiations. Please pay attention to:

- Ventilation (heat must be able to escape)
- All metal parts that can be touched must be connected to ground!

To cool the 4 MOSFETs a heat sink must be used. You can use the (properly dimensioned) front panel as the heat sink. Make sure the MOSFETs are electrically insulated from the front panel. The MOSFETs are soldered directly onto the main PCB. Use a ferrite bead for the gate connection of the MOSFET.

Make sure that the front panel and the cabinet are connected to ground and also that the central ground connection of the main PCB is connected to ground.

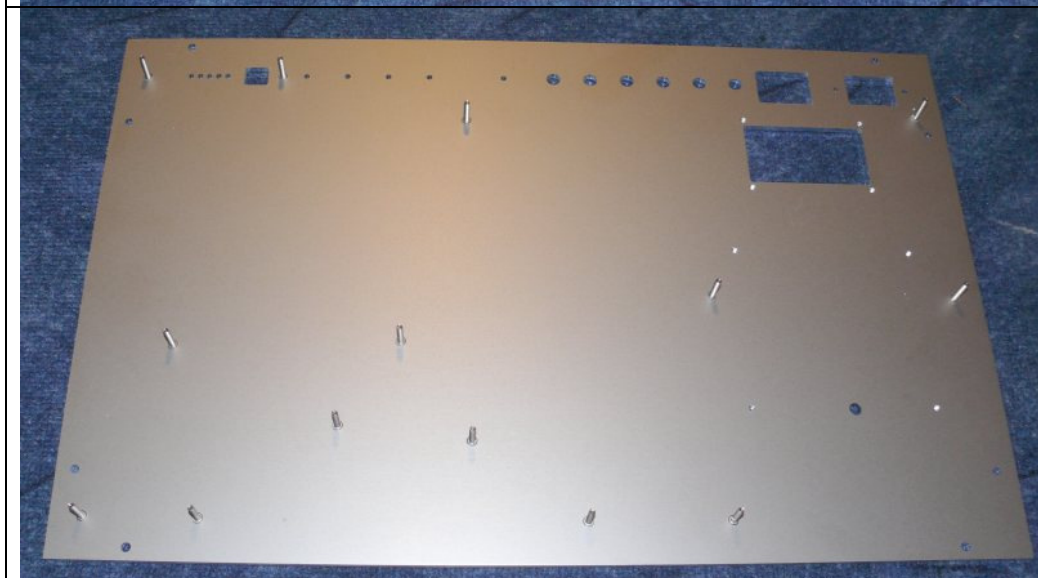
Make sure safety is ensured for example make sure you can't touch anything that carries a high voltage, and be sure to comply with all the safety regulations and requirements of your country.

My RoeTest V7 prototype:

Below are some pictures of my RoeTest V7 prototype:



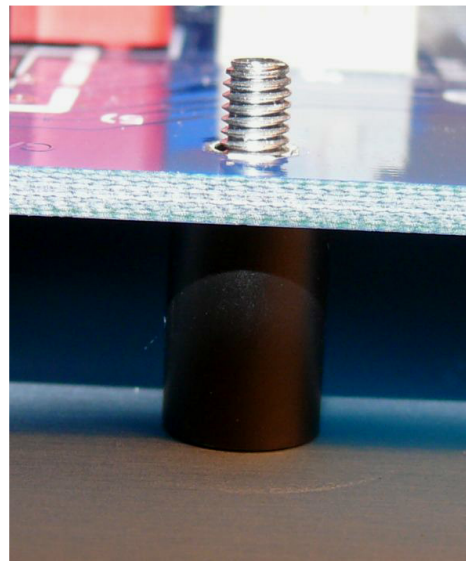
Front panel made by Fa. Schaeffer AG (made from 5 mm thick or more aluminum panel)



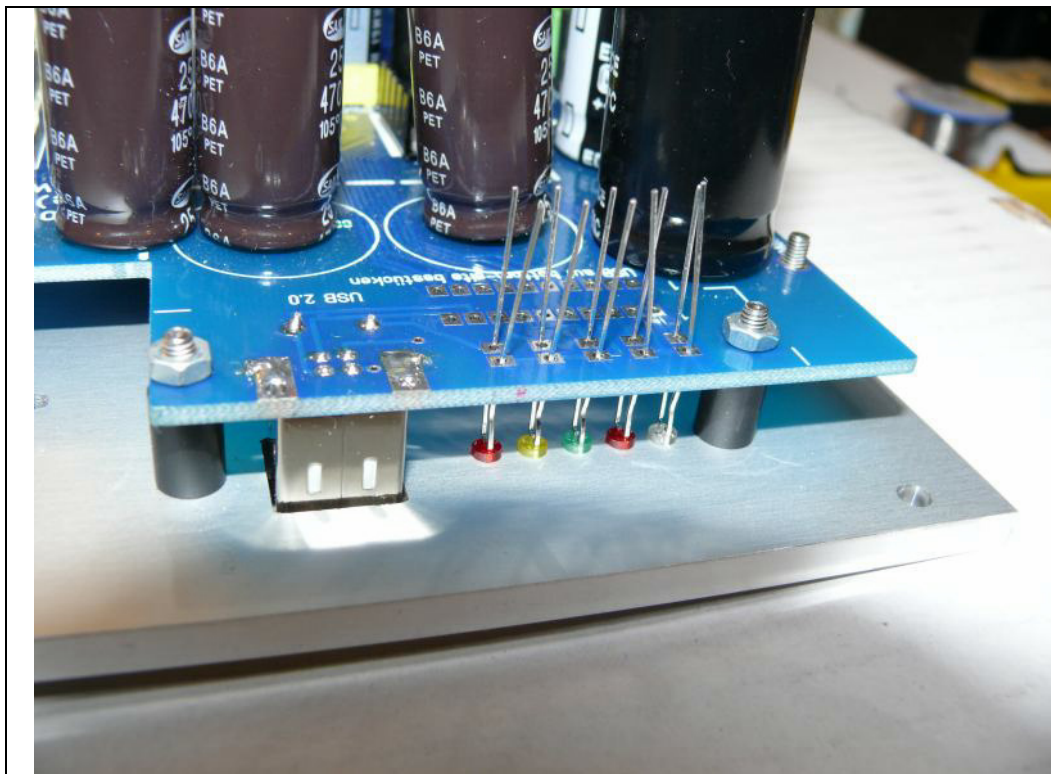
Back side with pressed in threaded M3 bolts



The stainless steel handles are available at home improvement stores. Use countersunk screws at the panel back side to attach these.



Threaded bolts: use 10 mm plastic spacers (except for the ground connection where a metal spacer should be used) when the main PCB is attached to the back side of the front panel.



The LEDs are soldered in after the PCB is attached to the front panel so that it's easy to solder them in at the correct height.

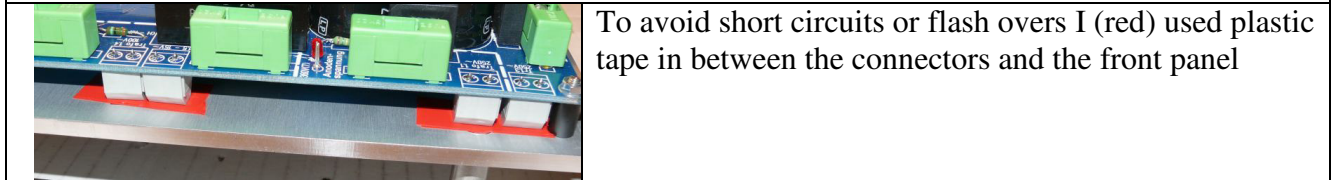
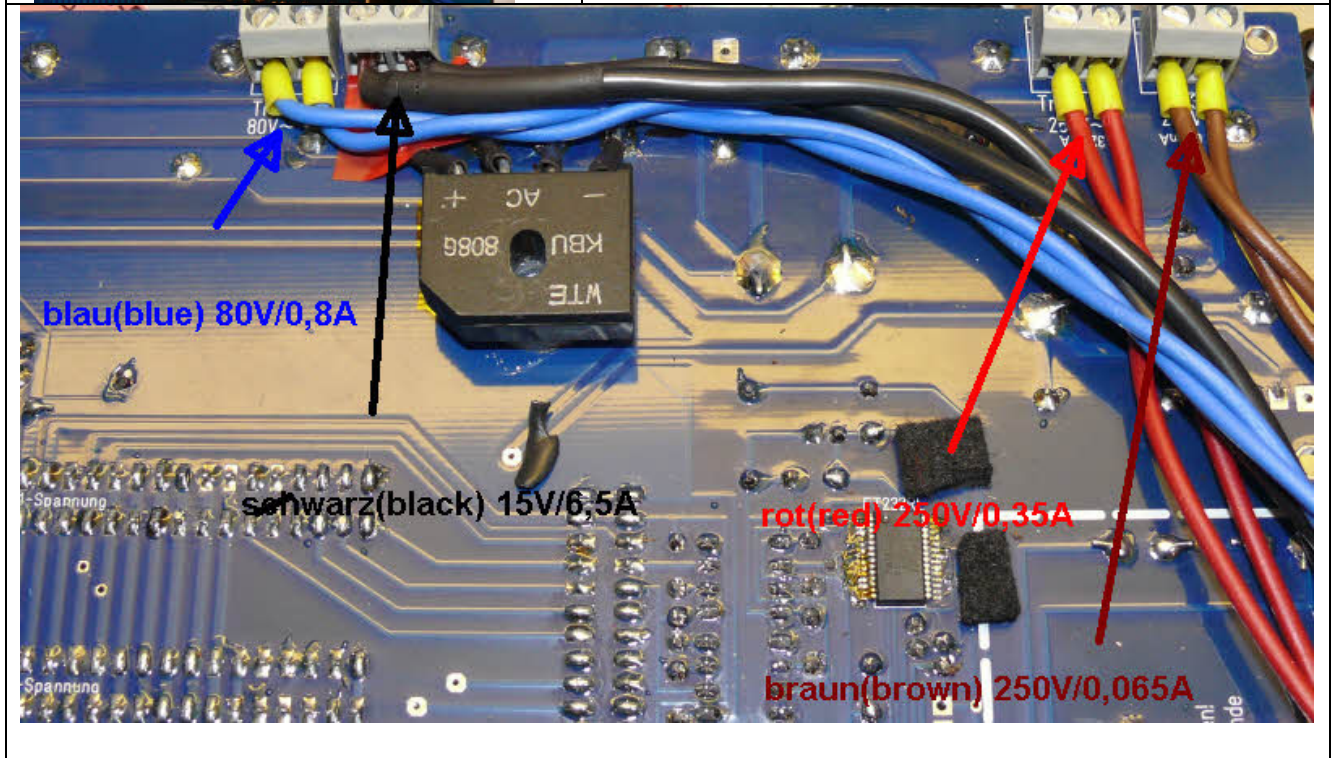
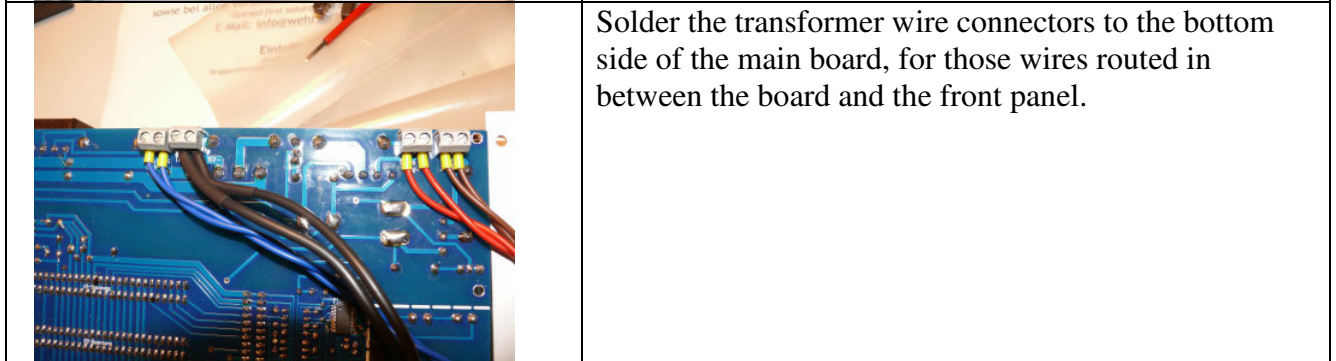
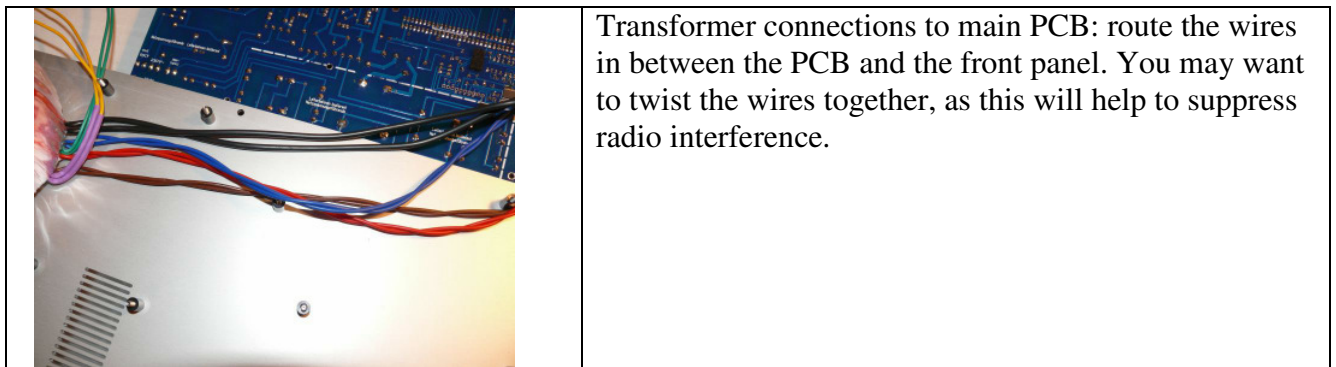


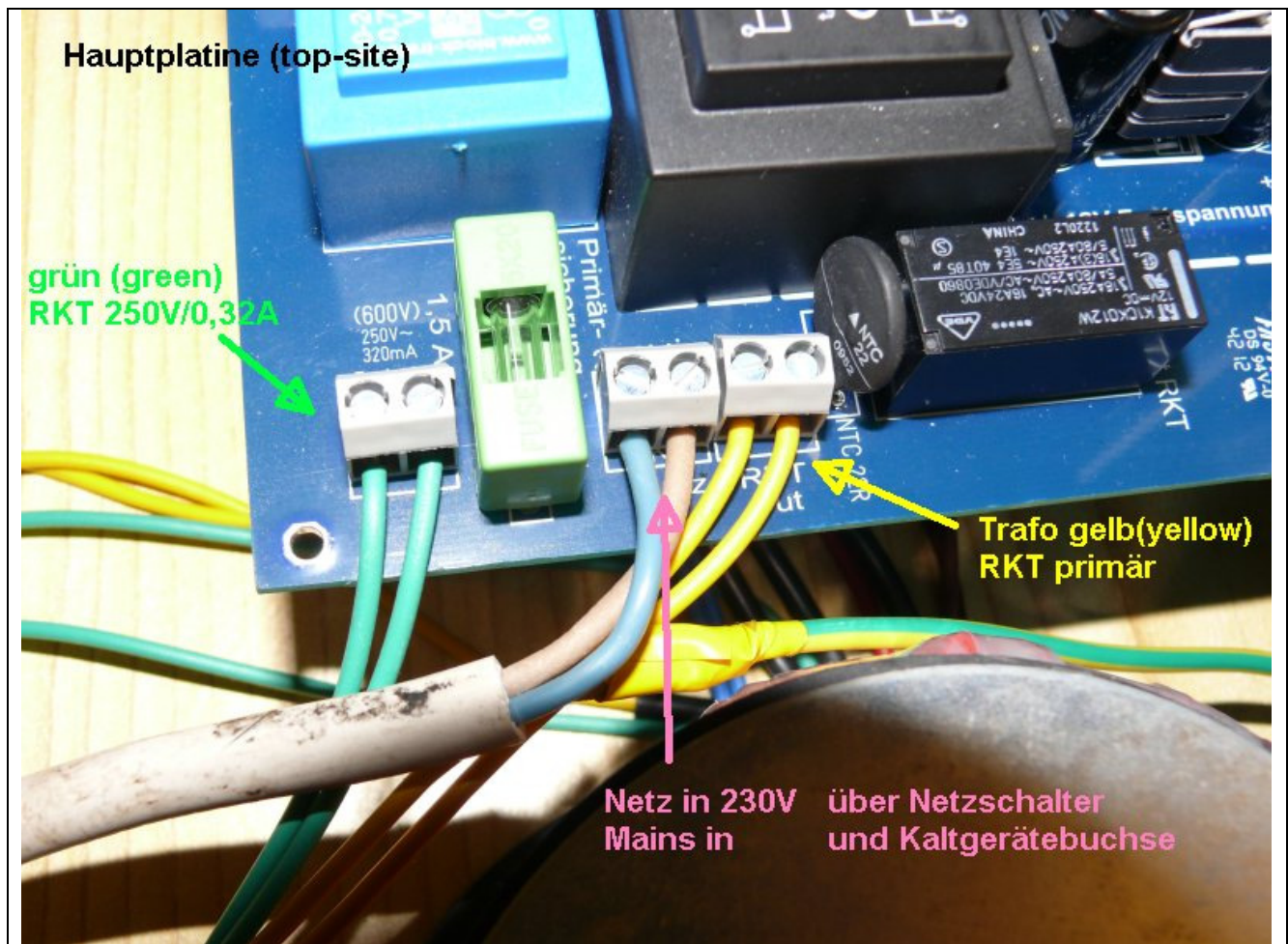
Attaching the banana sockets:

Besides the banana socket you will need heat shrink tubing and 8 mm plastic spacers (available at home improvement stores). The 8 mm spacer together with the washer and the nut are 10 mm, exactly the distance between the board and the front panel. The banana sockets also function to attach the board to the front panel.



Bolt the toroid transformer to the front panel.

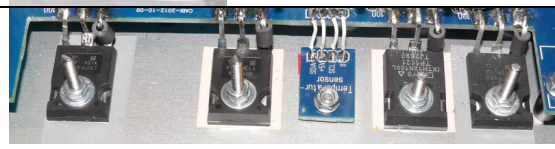




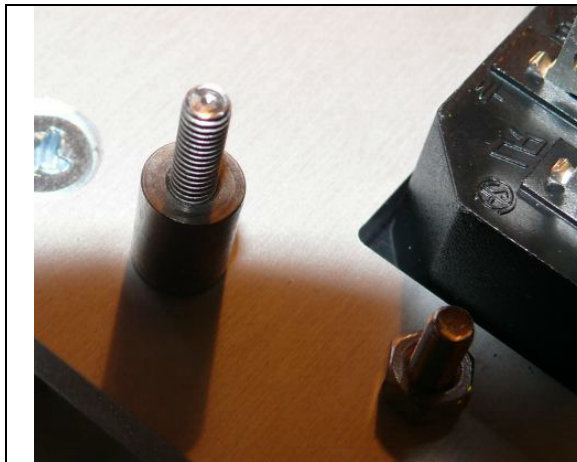
The main board connectors close to the toroid tranformer are soldered on the top side of the PCB. The cable for the mains connection is also routed in between the front panel and PCB. I used regular 2 wire mains cable.



The MOSFETs are soldered directly on the the main board. With this layout wire length is kept to a minimum. Use a ferrite bead for the gate connections of the 4 MOSFETs as shown.



The MOSFETs must be electrically insulated from the front panel. I use high end "Kapton" insulators – their thermal resistance is only 0.15K/W



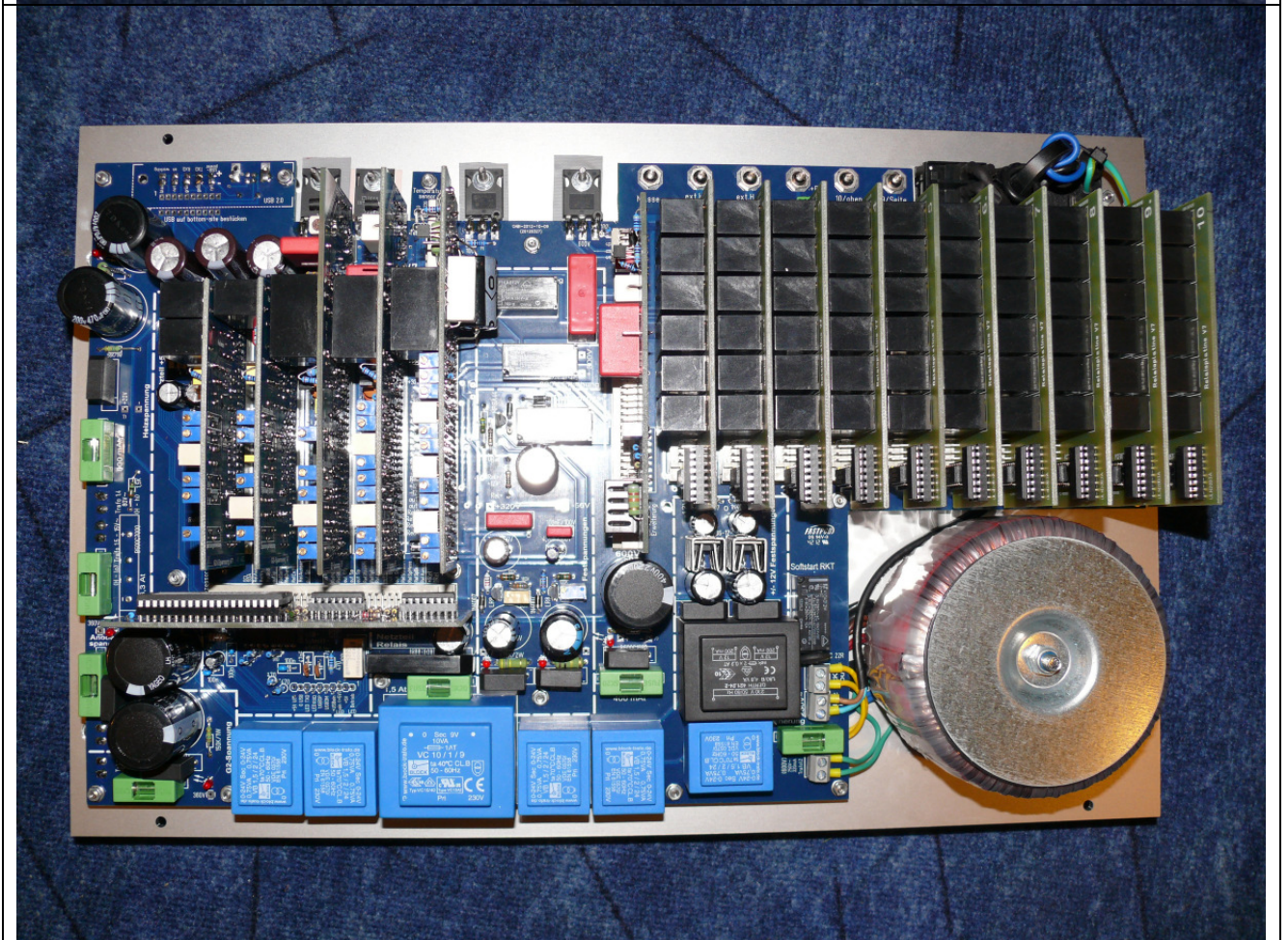
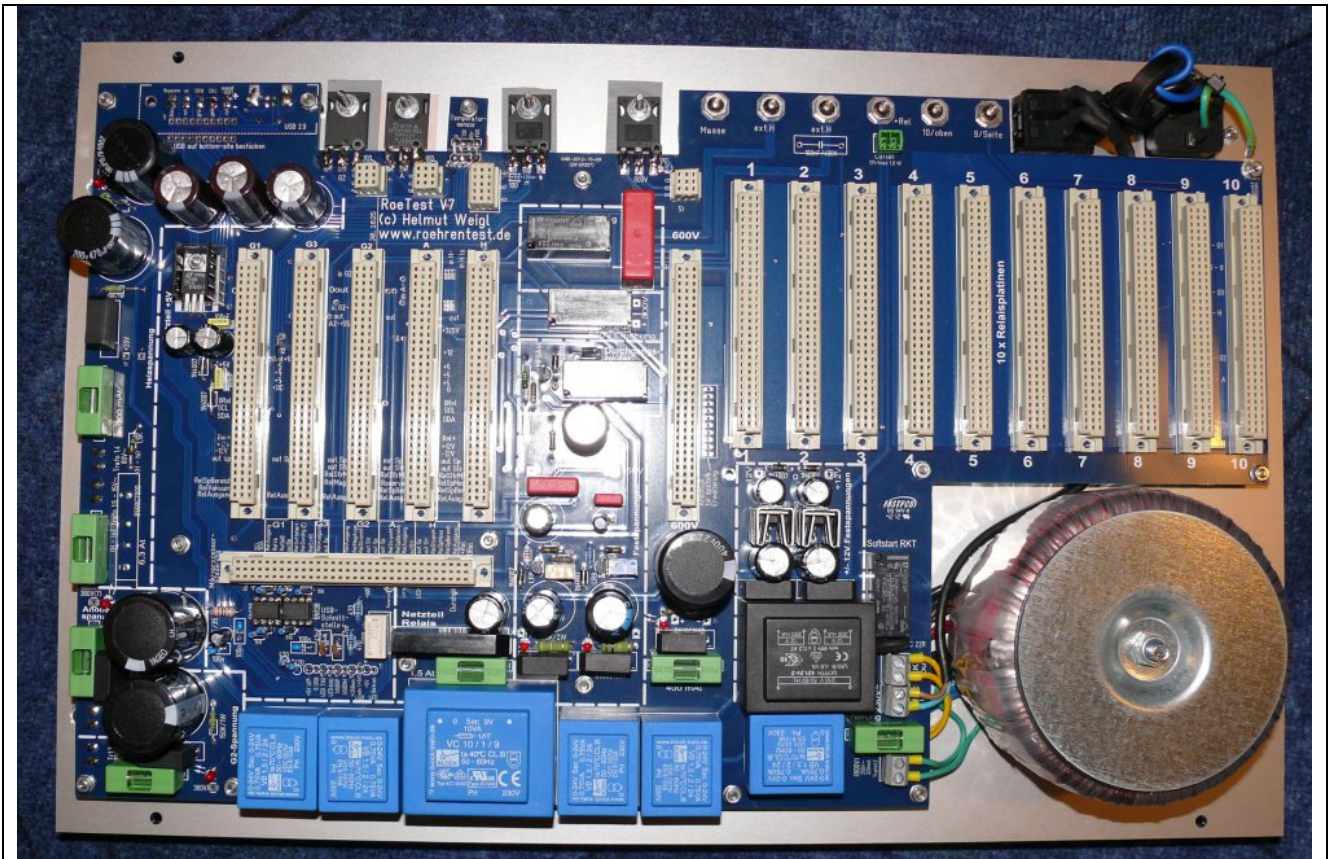
Use a metal (brass or copper) spacer to connect the main board ground connection with the front panel and connect the ground wire coming from the mains to the front panel. This way all metal parts that can be touched from the outside are grounded.

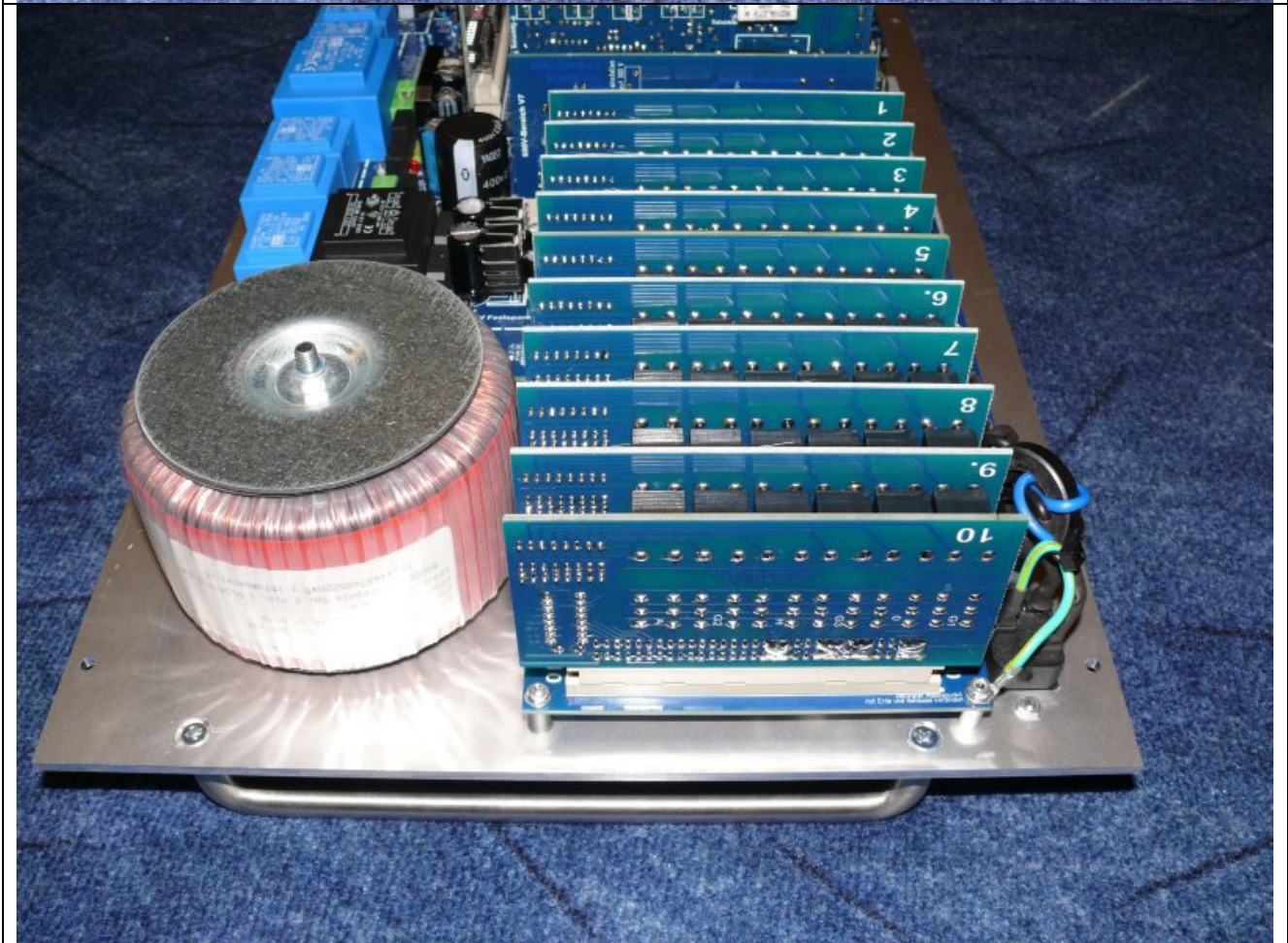
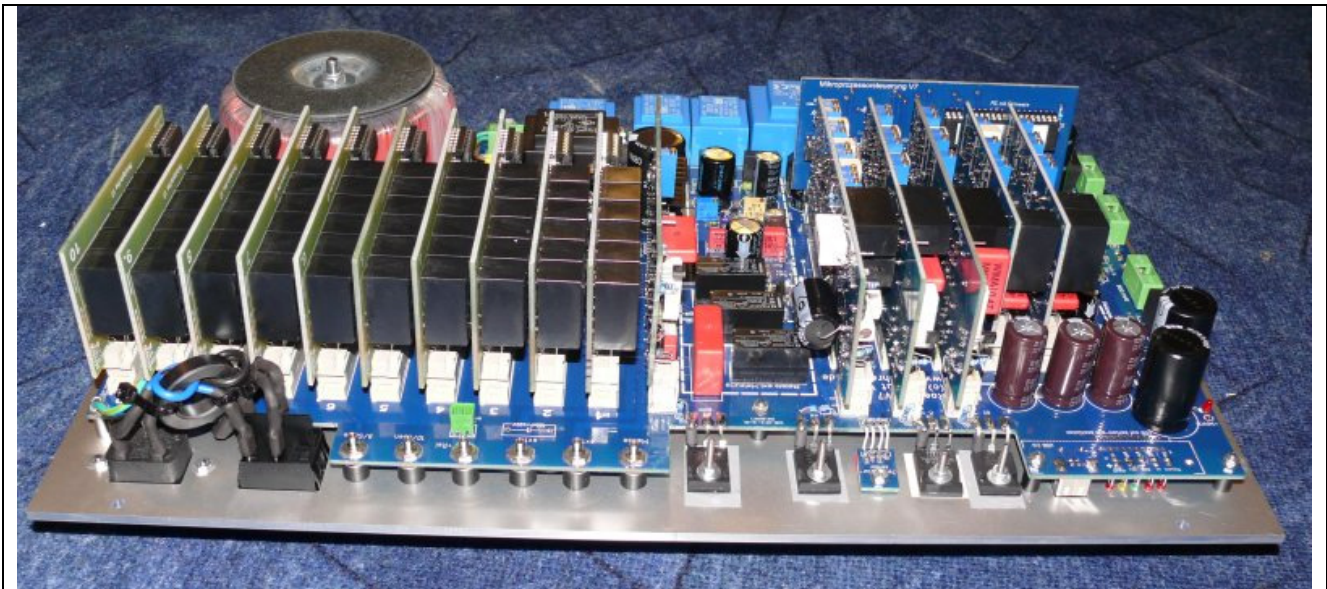


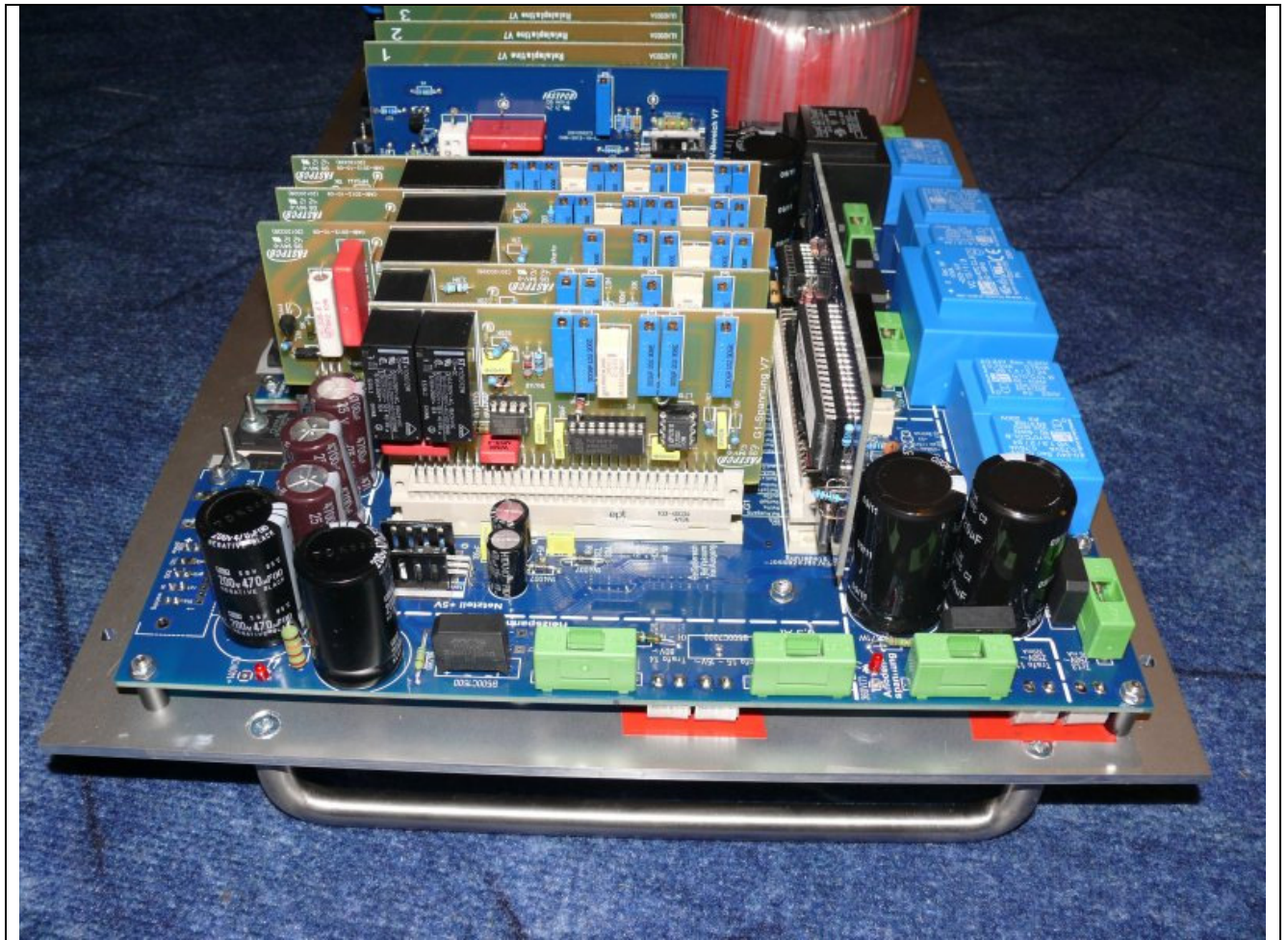
All other spacers are plastic!



All done:
front panel







The complete unit can be mounted in an appropriate enclosure for instance a tabletop cabinet (pictures RoeTest V6):





oder auch in einen Alukoffer: Um ausreichende Belüftung zu gewährleisten, habe ich hier einen Lüfter eingebaut. Dieser bläst von Hinten auf die MosFet's. Der Luftstrom tritt links und rechts der Frontplatte wieder aus. Zu diesem Zweck habe ich die Frontplatte etwas schmaler als den Koffer geplant.



Wiring:

The remainder of the wiring can be done quickly. Make sure you use quality wiring with good insulation and a sufficient wire gauge:

1. From the power cord connector to power on switch to main PCB (you may want to use an RFI filter)
2. Ground connection from power cord connector to front panel
3. Connect main transformer to main board (twist the wires)
4. Wires from main board to tube socket receiver (1.5 mm² wire gauge/thickness). Solder the wires to the main board bottom side. At the socket box receiver side I used 2 ferrite beads for each of the wires, with an internal 2 mm hole and insulated those using heat shrink tubing).

Additionally use ferrite beads or RF chokes in the socket boxes especially for those pins that can be connected to an anode.

Important: keep all wires to the socket box receiver as short as possible. The longer the wires, the more risk there is for oscillations with certain tube types.

Fuse table:

In my machine (RoeTest V6) I used the following fuses (all slow blow):

Primary:	1,6 A
Secondary:	
Heater low range:	6,3 A
Heater high range:	0,8 A
Anode supply:	0,4 A
600 V board supply:	0,4 A
G2 supply:	0,2 A
Relay power supply:	1,6 A

All fuses are on the main board.

First time power on / test procedure:

Please check all the wiring (again) before you switch power on for the first time.

The recommendation is to test parts of circuits separately, as much as possible. You can remove and insert various fuses and/or PCBs to do that, as indicated below.

Important note: when you need to make changes or fix something be sure to turn power off and wait until the filter capacitors in the power supply circuits have been discharged. Also don't insert or remove PCBs in/from the main PCB with power switched on or with charged filter capacitors!

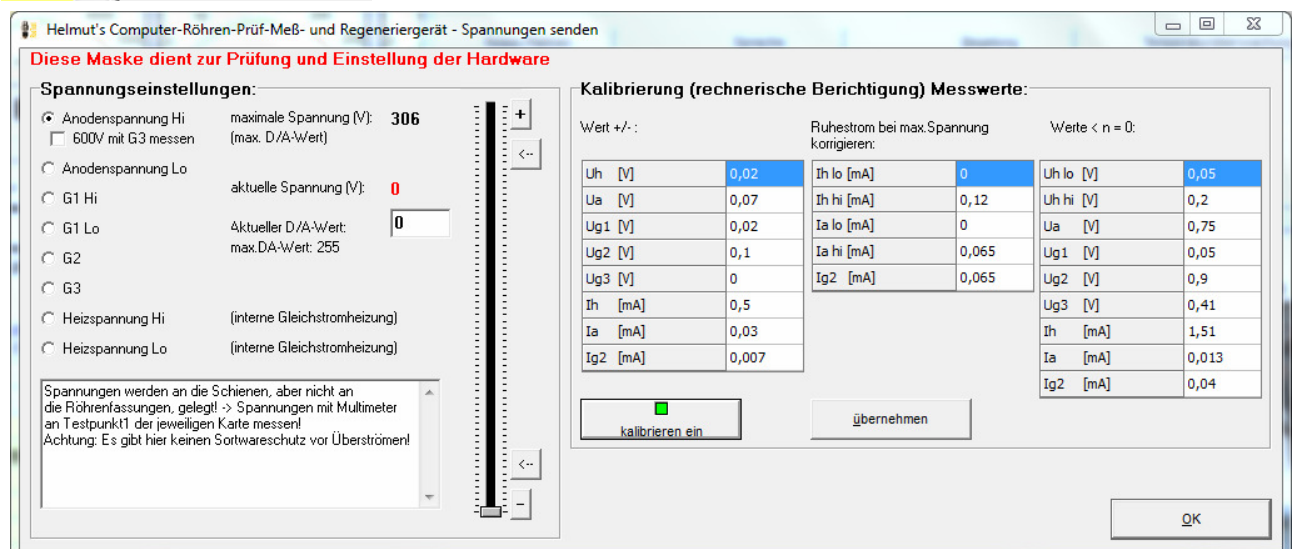
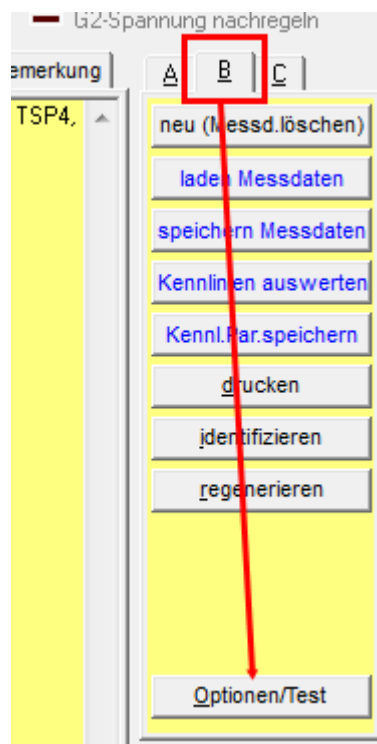
The following sequence of steps is recommended:

- First test all the power supplies connected to the mains and make sure the no-load (idle) voltages are not too high (they should not exceed the voltage limits of the filter capacitors), don't insert any of the PCBs in the main PCB sockets yet.
- First check +12 and -12-Volt, +5V, relay voltage, set the -56V using the trimmer pot (the idle voltage of the circuit before the regulator is ca 90V) and set the +320V using the trimmer pot (the idle voltage of the circuit before the regulator is ca 380V, the maximum allowed value is 400V)
- Then insert the fuses in the secondary circuits of the heater, anode and G2 power supplies and measure the output voltages. Test pins can be soldered onto the main PCB at the indicated locations and can be used to discharge the capacitors and/or measure the voltages. The idle voltages for the A, G2 and 600V supplies should be approx. 360V.
- For the next steps switch off the anode, G2 and heater supplies by removing the fuses in the secondary circuits.
- Insert the microcontroller PCB, first make sure the +5V supply and unregulated relay power supply are working. After switching on power the working/on indicator LEDs should blink a few times, indicating the PIC microcontroller is starting up. Now it should be possible for the PC software to communicate with the PIC microcontroller when connecting the two using a USB cable. Make sure all necessary drivers are properly installed, see the user manual for instructions. The PIC should now respond when it gets commands for instance when you start the „testing for shorts“ test the working indicator LED should blink.
- With +5V and relay power on, initiate the relay test: PC-Software->Options/Test->Relays, and test the check for continuity relay on the main board
- Now insert the relay PCBs (make sure the PCF8574(A)) chips are inserted) and test the relays: PC-Software->Options/Test->Relay-PCBs, one card after the other, and test each of the pin relays.
- Now insert the remaining PCBs and test the relays (using the PC-Software)
- Test the PCF8991 in the H, A, G1, G2 and G3 boards: select PC-Software->Options/Test->sending voltages. With the slide control you can set the PCF8591 pin 15 output voltage for each of the boards. Measure the output voltage on pin 15, you should be able to set it from 0 to +5V using the slide control.
- Now test if you can set/control the G1 and G3 output voltages using the PC software. Connect your meter to test point 1 on these boards. Note that the boards are not yet calibrated.
- Now switch the A, G2 and H power back on by reinserting the fuses in the secondary circuits and test if you can set and control the output voltages, one card after the other.
- And finally insert the 600V PCB which has a 300V fixed output voltage that is switched in series with the anode board voltage.

Calibration:

In order to calibrate the machine it should be put together completely and all circuits must be operational. The first step is a rough calibration/setting of all trimmer potentiometers (henceforth called trimmer pots). For a final calibration the machine must be fully warmed up (which takes about 30 minutes at room temperature) and the calibration steps are repeated after warm up for that purpose. I recommend recalibration after the machine has been used for a few days and then at regular intervals..

To calibrate the hardware, select menu "B" ->Options/Test->setting voltages) and you should see the following screen:



Here you can set the output voltages of each individual card.

Note:

- the output voltages are connected to the voltage rails but not to the tube socket connections, Connect your test instruments and load resistors to test point 1 on each of the cards.

- **There is no software over current or short circuit protection** – don't overload the MOSFETs (for instance, connect the maximum anode board output voltage of 300V to ground to simulate a short circuit for a longer period of time. The hardware limited maximum output current is about 350 mA so that means that with a source-drain voltage of about 350V the MOSFET has to dissipate some 115W, and it can do that for a very limited amount of time but will get very hot in any case. And if it gets too hot it dies...).

Constant voltages

The +320V and -56V constant voltages can be calibrated using the trim pots marked with the green arrows in the picture below. Use the test points indicated with the red arrows to measure the voltages, measure the voltage between the test point and ground.



Variable/microcontroller controlled output voltages:

The next section describes the calibration procedure of the anode board. The heater and G1, G2 and G3 boards are calibrated in a similar manner.

Select the 400V range on your multimeter and measure the voltage between test point 1 on the anode board (or the other boards) and ground.

On the pc software, select Options/Test->sending voltages. Make sure software offset compensation is switched off.

The output voltage is calibrated at low and high end points of the output range. Because the D/A converters get inaccurate and the end of their ranges (digital 0 and 255) we don't calibrate at the extreme end of their range but use the D/A values of 20 and 230 to calibrate.

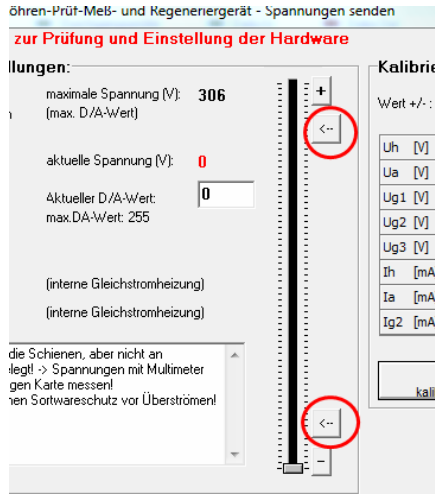
Select the anode/plate voltage low range (0-51V) and select a current DAC value of 20 either by moving the slider to the appropriate position or by entering the value in the field where it displays the DAC value. Then turn the trim pot marked unten, kleiner Bereich on the anode PCB and adjust it until your multimeter shows the same voltage as the voltage (red) on the screen. Then enter a DAC value of 230 and adjust the trim pot marked oben, kleiner Bereich until the measured voltage equals the displayed voltage on your computer screen. Since one trim pot adjustments may affect the other repeat this calibration sequence a few times.

Then select the anode/plate voltage high range and calibrate using the same procedure.

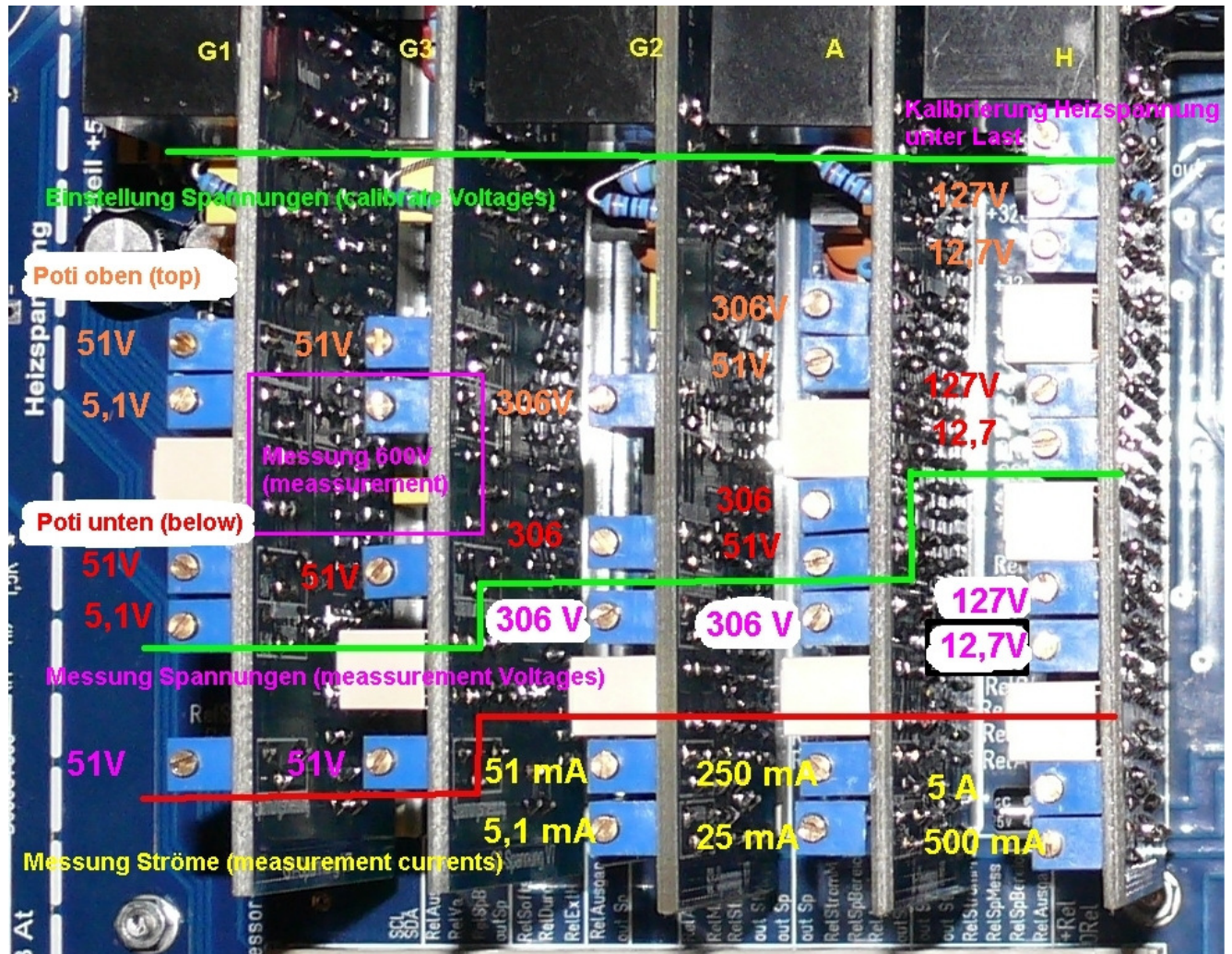
All calibrations are done without a load resistor.

Then calibrate the G1, G2, G3 and heater boards in a similar manner.

To select the 20 or 230 DAC values for calibration you can also use the buttons circled in red as indicated in the picture below:



Position of the trimmers on the PCB's:



Calibration of the 600V board:

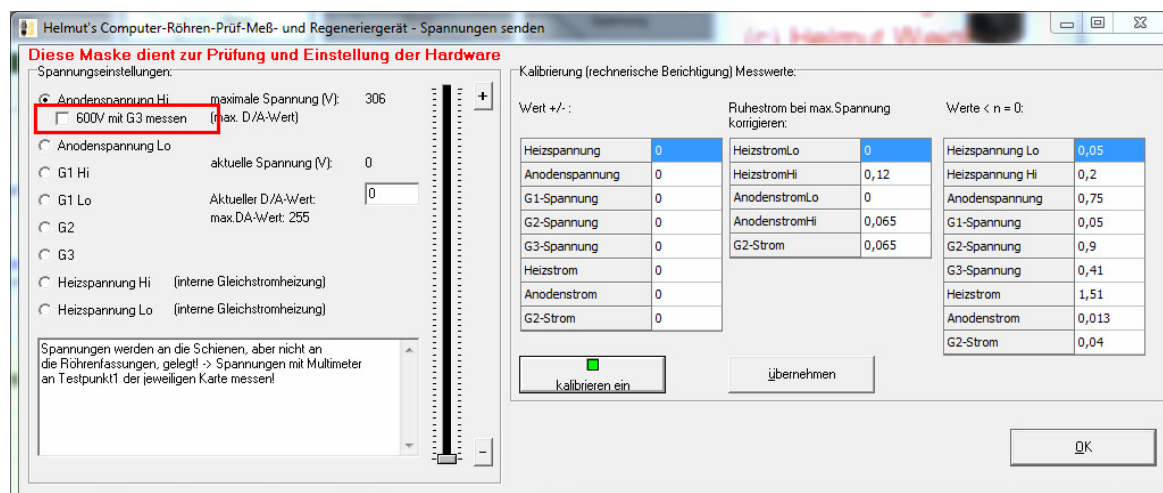
Connect your multimeter to the test points marked +/- on the PCB and adjust the trim pot until you measure 300V exactly. Connect a load resistor for instance a 15W bulb (make sure the bulb can handle the voltage) and verify that the output voltage is kept constant within approximately a 1-2V range.

Calibration of the voltage measurement ranges:

Purpose of calibrating the voltage measurement ranges is to ensure that the virtual volt meters displayed by the software indicate the same values as the multimeter. Adjust the trim pots until they do. Do this calibration at the high end of the output range for example at 280V anode voltage. Offset calibration is not possible. Only when there is no other option (and you are sure the hardware is OK) you can set an offset voltage in the software. For example enter if the offset is +0.1V enter a value of -0.1V (Options/Test->sending voltages->software calibration of measured data->rating +/-). The calibration compensation is only made effective after pressing the „use new values“ button. You can also set the limit at which values are ignored (For $< n = 0$). This is useful when your instruments don't exactly indicate 0 when the machine is idle.

Calibration of 600V measurement range on G3 card (from version 6 onwards):

The new G3 card has a 600V measurement range allowing you to measure voltages up to 600V – only when the G3 voltage output is not needed. You can use this for instance when testing voltage regulator tubes. For this purpose, the G3 card is connected to the anode board. You can do this **in the software** by checking the checkbox as indicated in the picture below:



Set the anode board output to 280V and then calibrate the G3 board by adjusting the trim pot for the 600V range on the G3 board until it indicates 280V. **Note: you can only do this if you have set the correct version of the G3 card in the software (Options/test->Options) and your G3 card must be version 6 or higher.** Otherwise the checkbox will not be displayed on the screen. **Never connect the cards using a wire!**

Calibration of the heater voltage measurement:

Problem:

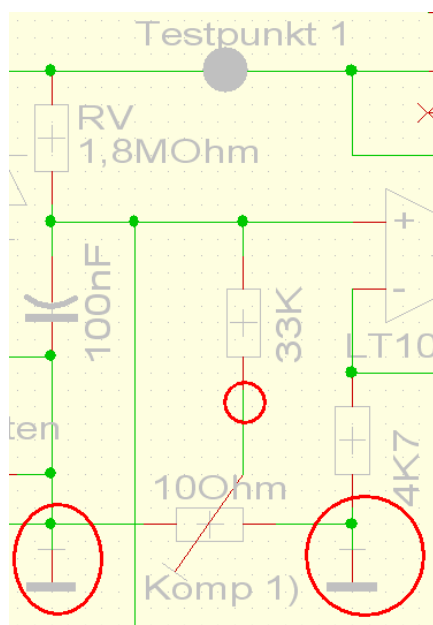
When you connect a multimeter to the tube socket and measure the heater voltage (for instance when you have selected manual mode for the heater voltage) the measured voltage will be the same as what the software indicates. However if a load is connected (for instance by inserting a tube that requires 1A heater current) it is possible that the value measured doesn't exactly match what the software indicates anymore. For example the RoeTest indicates a slightly higher voltage as the multimeter – depending on how much current is drawn.

Cause:

Also copper traces, connector pins, relay contacts, wires etc. have a resistance. If there is a current there will be a voltage drop. This can impact the measured values, also in the RoeTest, even if this voltage drop is very small. This only impacts the heater voltage low range (0 - 12,75V) for the following reasons:

- High currents can flow
- The measurement amplifier amplification factor is high
- So a small deviation in the indicated value compared to the (low) actual voltage can be seen (in case of a 300V anode voltage a measurement error of 0.1V can be ignored!)

The problem with the voltage drop is mostly a ground connection problem. Eventhough the ground traces are wide (and fortified with 2,5 mm³ wire) there is still a voltage drop. This effectively shifts the 0 point for the measurement amplifier with respect to the point where the voltage divider is connected:



Therefore it's important where the 0 points on the main PCB are connected. Slight voltage variations at the different ground connections impact the amplifier output and result in measurement errors.

Current measurement calibration:

Current measurement ranges must also be calibrated.

For example, for the anode current, there are two measurement ranges (a 0-25 mA low range and a 0-250 mA high range).

Select the 0-51 anode voltage range.

Connect a resistor that can handle the wattage e.g. a 1200 Ohm / 75W resistor in series with a milliamp meter to the anode board (test point 1) and ground.

Tip:

If you don't have the required high watt resistors you can use a 230/240V light bulb e.g. a 15W light bulb for the G2 board.

Increase the output voltage until the multimeter indicates about 20 mA. Now adjust the trim pot for the low range until the virtual instrument displayed by the software also indicates the same value.

Then select the 300V output range and set the output voltage such that your multimeter indicates 150 mA, and adjust the trim pot for the high current measurement.

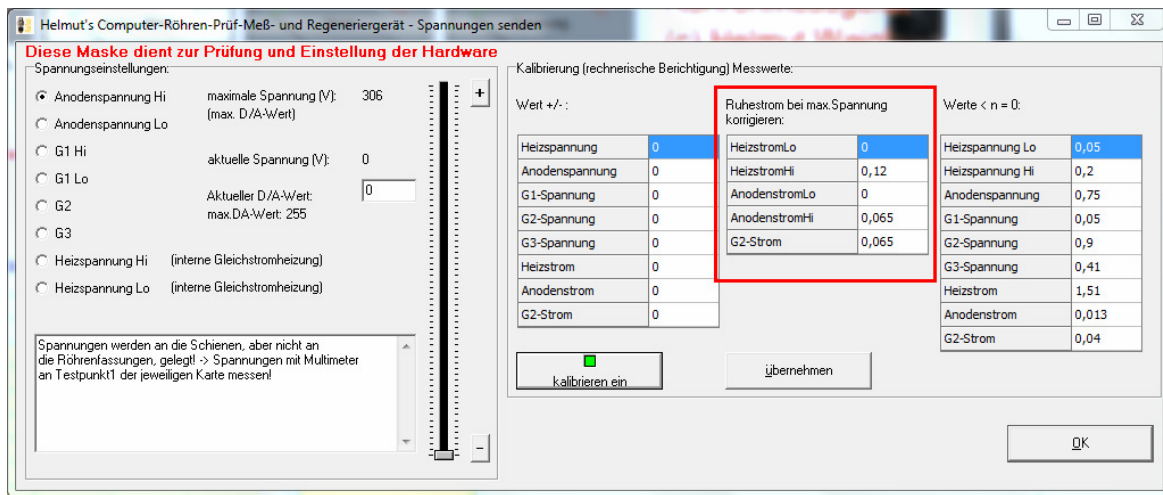
Now set the slider to 0V and slide it slowly up. At about 25 mA you should hear the relay click that switches the current measurement from the low to the high range. Now lower it again and it should switch back to the low range – with a little hysteresis. Only when the ranges are properly calibrated the switch from low to high range and back happens at the right moment!

Offset calibration is not possible. If need be you can set an offset in the software e.g. if the offset is +0.02 mA enter a -0.02 mA value in the software calibration screen (Options/Test->sending voltages->software calibration of measured data->rating +/-) and press the „use new values“ button.

If there are large offset values you should find the cause of it and remedy the problem e.g. replace the opamp if it causes the problem.

Remove the load resistors. If now the output voltage is set to the maximum value the current display should still indicate 0. If not and the value shown is high you likely have a ground connection problem and you should try and find the cause. My prototype shows a leakage current of 0.075 mA for the A and G2 boards. There will always be a small current flowing at maximum output voltage, if only because the voltage divider network at the output used to measure the output voltage puts a small load on the output ($300V : 4.733.000 \text{ Ohm} = 0,065 \text{ mA}$) so the remaining 0.01 mA is offset. For this case there is the option to have the software „remove“ the idle current at maximum output voltage. To do that, enter the indicated current value in the idle current table and the software will adjust it's indication accordingly.

Enter the leakage current here and press the uebernehmen (use new values) button!



Current limiter:

The RoeTest uses hardware current limiters to limit the current output as per the table below:

	max. current (design)	Current limiter kicks in at
Heater low range	ca. 5000 mA (maximal)	ca. 6000mA
Heater high range	500 mA	ca. 670 mA
Anode/plate	255 mA	ca. 350 mA
G2	51 mA	ca. 68 mA

Note that the actual values at which the current limiter kicks in depend on the tolerances of the semiconductors and the resistors. The transformers used should be able to continuously deliver about 1.25 times the maximum current.

To test the hardware current limiters:

Test the heater, anode and G2 voltage boards, one after the other:

-> connect a suitable resistor (that can handle the load, or if you don't have one use for instance a light bulb) to test point 1 and ground and increase the output voltage until the current limiter kicks in. Only do this for a short period of time! Output voltage should decrease when the current limiter kicks in and the output current should not further increase.

Note: make sure to use **5W wire wound resistors** for the current measurement and current limiter circuits – other resistor types repeatedly failed on me.

To test voltage regulation of the H, A and G2 boards:

From Roetest 4 onwards electronic voltage regulation is used. Output voltages must remain stable as long as the output current limiter does not kick in. Connect a resistor and verify the output voltage is constant.

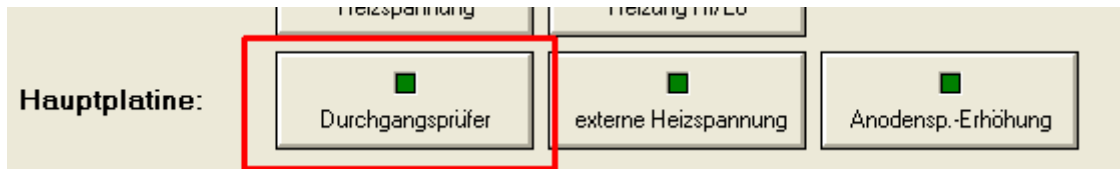
Test check for continuity circuits:

The check for continuity circuit is used in various tests – for instance for the filament test or when testing for shorts.

Test the check for continuity circuit without a tube inserted!

The S2 (A) and S4(G2) rails are used when checking for continuity. When the „check for continuity“ relay is switched on 5V fed through a resistor and diode (for protection) is connected to the S4 rail and should be measurable on S4. You can switch the relay on using the software as shown (PC software->Options/Test->Relays->check for continuity). Now when the S2 and S4 rails are connected the MPSA44 is switched on and the signal B7 at the PIC goes from hi to lo. The software should indicate that as shown below.

Note: if the check for continuity circuit doesn't work reliably, you may have voltages somewhere that are not properly connected to ground. Make sure the A, G2 and H boards are inserted when testing this and that the fuses are inserted in the secondaries of the power supplies for these cards. If you have the 600V board inserted than this must be connected to ground through a 0,47µF/630V capacitor.



Now if everything works correctly and is correctly calibrated you can insert your first tube and test the tube!

Closing remarks:

When you have successfully built the RoeTest you'll have a tube tester you can't find elsewhere. If you don't count the hours to build it, it'll cost you less than a well maintained vintage tube tester. You can do a lot more with the RoeTest, and it is much simpler to operate.

Good luck with building the RoeTest and have fun testing tubes.

Helmut Weigl

Additional documentation:

The following documents are on a CD-ROM that can be ordered from me:

- complete circuit diagrams
- PCB layout diagrams
- parts database
- more pictures
- PC-Software (measurement software, drivers, database)
- many documents and manuals for instance this manual.

The programmed PIC microcontroller can only be obtained from me.

As long as supplies last, you can also order the PCB set and the main transformer from me.

You can find more information and software updates (published irregularly) on my website

www.roehrentest.de