

Table of contents:

• Safety note	3
• General instructions	4
• Tips and Tricks	7
• Finding your way around the board	12
• Step-by-Step assembly	13
▪ Part 1 - Com port installation	15
▪ Part 2 - The graphical user interface installation ...	16
▪ Part 3 - The RS232 connector cable	20
▪ Part 4 - The voltage regulators	22
▪ Part 5 - The RS232 interface	24
▪ Part 6 - Micro-controller	26
▪ Part 7 - Heater supply	29
▪ Part 8 - Negative power supplies	31
▪ Part 9 - Grid bias circuit	33
▪ Part 10 - Boost converters	37
▪ Part 11 - HV switches	42
▪ Part 12 - Calibration	46
○ Heater supply	47
○ Grid bias circuit	49
○ Boost converters	51
○ Current amplifiers	53
○ Backup calibration values	55
• Configuration	56
• Wiring of the uTracer	57
• Stopping Oscillations	61
• Schematics	62
• Floorplan	64



Important!



Please read this page carefully!

Although your uTracer is not directly connected to the mains, it generates dangerous and potentially lethal voltages. Please observe the following guidelines:

- Never touch the PCB when the high voltage LED is on!
- If you need to work on the circuit:
 1. First wait until the high voltage LED is off;
 2. Then switch off the power;
 3. Check with a DMM that the 100 μ F / 500 V caps are fully discharged
- When you test the circuit on the bench use the standoffs to distance the PCB from the table.
- Make sure that there are no loose wires or metal clipping lingering around or underneath the PCB.

Please note that even when the high voltage LED is off, the -100V grid supply is still working !



The design of the circuit, PCB, the selection of the components as well as the preparation of this document have all been done with the greatest care to guarantee a successful construction and to prevent disappointments,

Nevertheless, small errors and/or omissions may have occurred and I will be grateful if they are reported to me.

The following pages contain important notes and tips
Please read them carefully!

In case of any problems occurring during the construction of the kit please first consult the FAQ page on my site www.uTracer.nl. Additionally, I will be happy to offer assistance by email as good as possible. It is unfortunately not within my means to offer any kind of guarantee or refund.

Finally, I would like to stress again that the voltages that are generated by the circuit can give you a nasty shock and are potentially lethal. I can never be held responsible for any accident or injury resulting from the (improper) use of the uTracer6 circuit.

Purchase of this kit implies that you agree with these terms and conditions.

Before you begin

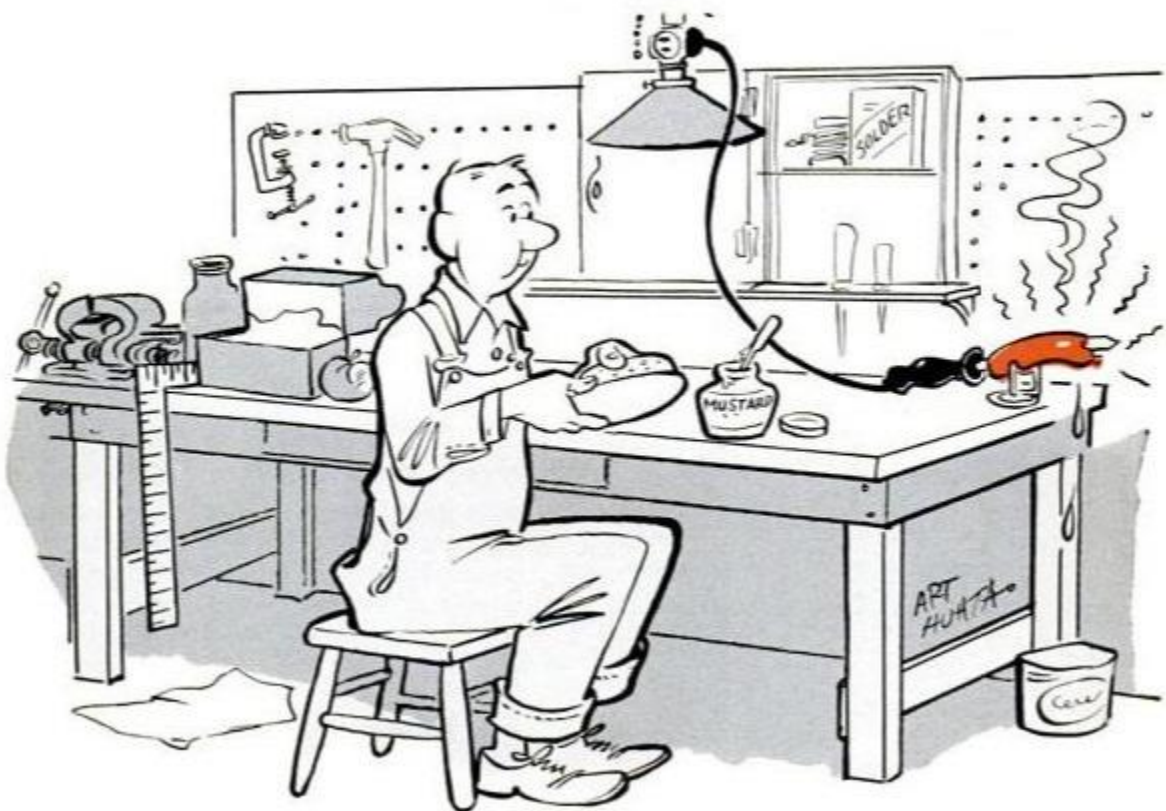
1. In order to build the μ Tracer6 kit successfully , you need a certain level of experience in building modern electronic circuits. Although no complicated soldering of SMD components is involved, you need to have some skills in fine pitch soldering. A look at the PCB will give you an idea of the kind of skills that you will need.
2. The most important requirement to successfully build the μ Tracer is patience. There are a lot of components on a relatively small PCB. One error and the circuit will not function properly. Fortunately it is possible to build and test the circuit in steps. I strongly recommend to follow the construction manual exactly, and to take the time to do the tests as described in this manual. The best way is to go through this manual page-by-page and line-by-line. Every action line is provided with a so that it can be ticked-off after it is executed.
3. This kit only contains the components to construct the electronics, the heart, of the tube tester / tracer. It is up to you, the user, to add tube sockets and some kind of electrode selector matrix according to your specifications. At the end of this manual an example of how this can be done will be shown.
4. Although this circuit is supplied by a low voltage supply, and although the boost converters can only generate a relatively small amount of current, the charge stored in the high voltage capacitors can be lethal nevertheless! Here are a few simple measures to avoid accidents:
 1. The “High Voltage ON LED” is an indicator that high voltages are present in the circuit. Don’t touch the circuit when the LED is on unless you know exactly what you are doing.
 2. The “High Voltage ON LED” might not function properly! So before you start working on the circuit, always check if the high voltage capacitors are discharged to a safe level
 3. If you need to touch the circuit while it is operating, always keep one hand in your pocket!
5. The μ Tracer6 is deigned to operate from an old Laptop power supply. These supplies can generate a lot of power. During assembly and testing it is better to use a normal variable lab bench power supply adjusted to 19.5 V, 500 mA max, unless otherwise noted. In this way the damage done when an accidental short circuit occurs will be limited.
6. The μ Tracer6 is provided with a protection diode that will short circuit the power supply in case the polarity of the power supply is accidentally reversed (+ and – exchanged). Although it will protect the circuit, your power supply will not like it! To prevent accidentally reversing the power supply wires, I use a simple 90 degree clip to prevent accidentally connecting the power leads the wrong way (see Tips and Trick 13). I strongly recommend you to use a similar method.
7. Although I have tried to use as much as possible exactly the same components in the kit as the components shown in the photographs, the shape and color of some of the components may be different, depending on availability.
8. I have put a lot of effort and time in making this kit and the manual as complete and accurate as possible. Nevertheless there is always the possibility that some (small) errors or omissions have been made. I will be highly obliged if they are communicated to me (Ronald@dos4ever.com).

Tools you absolutely need:

1. A soldering iron with a fine tip, suitable for fine pitch work, including a wet sponge to clean the tip;
2. Solder 0.5 mm diameter but finer is even better. I still prefer to use lead containing 60/40 solder;
3. A multi-meter, a digital one is preferred;
4. A pair of test cables with miniature clips;
5. A micro flush cutting nipper, or similar to cut the remaining end of components soldered to the PCB;
6. A small screw driver.

Things that come in handy:

1. A lab bench power supply at least 0-20 V, 500 mA;
2. A pair of tweezers;
3. A magnifying glass;
4. A small de-soldering pump;
5. Some solder wick to de-solder components;
6. Soldering flux (bottle or pen).

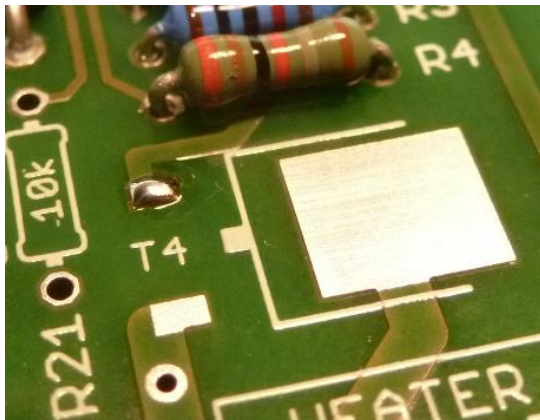


Tips and Tricks:

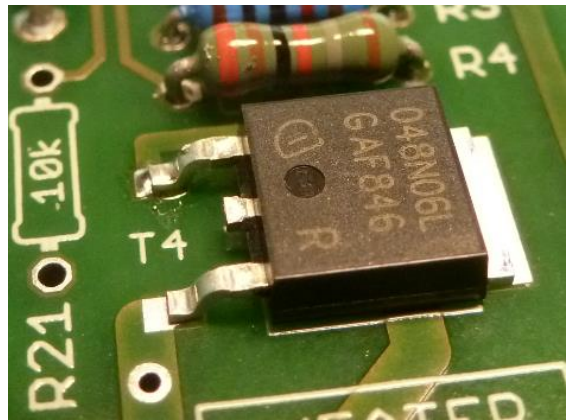
1. Bend the leads of the resistors in a sharp angle close to the body of the resistor.



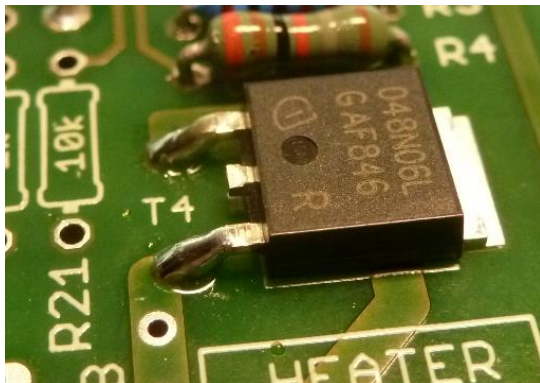
2. This kit contains a number of SMD type transistors. These transistors are easy to mount on the PCB and they have the great advantage that they can also be easily replaced when needed. Although not absolutely necessary, some soldering flux either in a bottle or in pen is recommended. This is my preferred procedure:



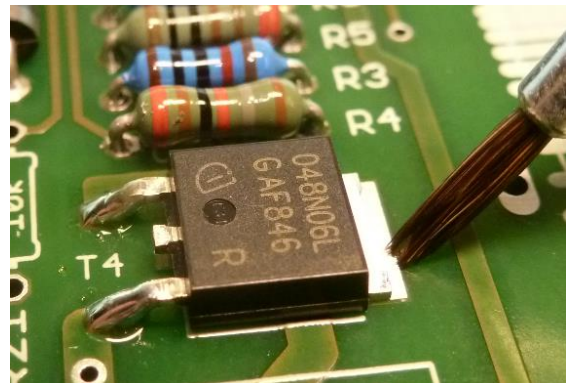
Apply solder to one pad



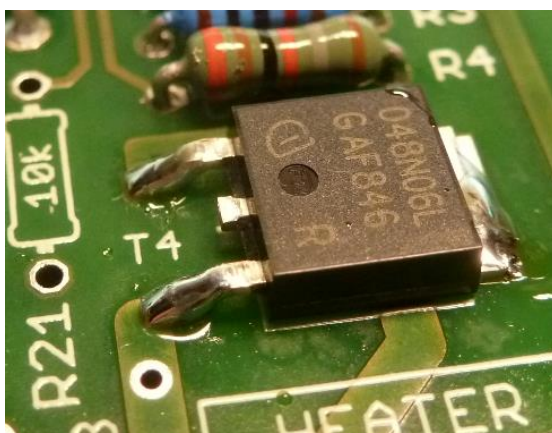
Fix one leg of the transistor and adjust position



Solder the other small terminal



Apply some solder flux to the tab

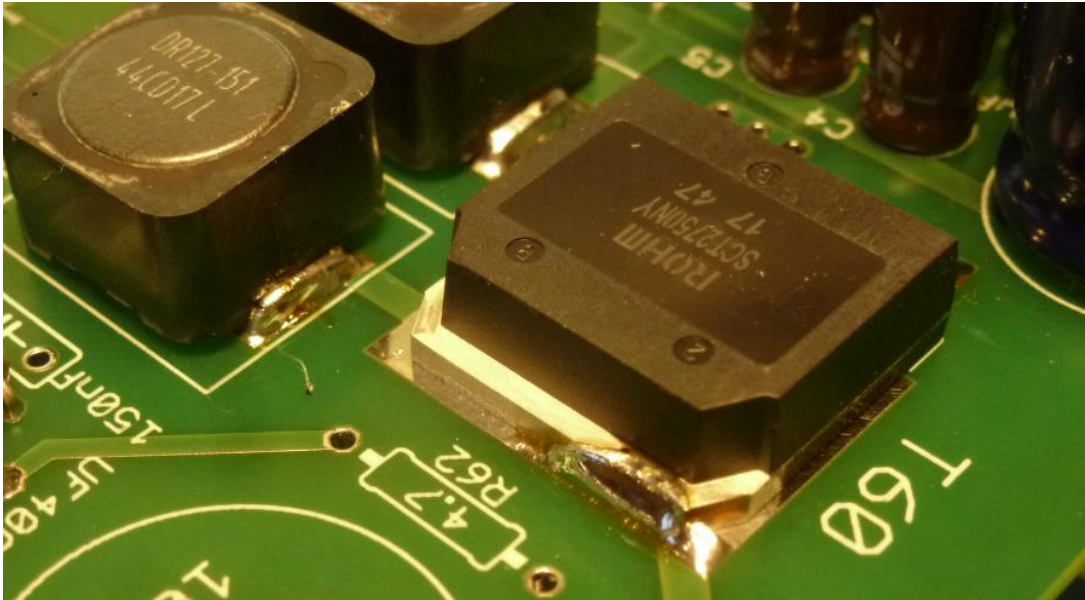


Heat the tab with your soldering iron and add a bit of solder. When hot enough the solder will ready flow between the tab and PCB.

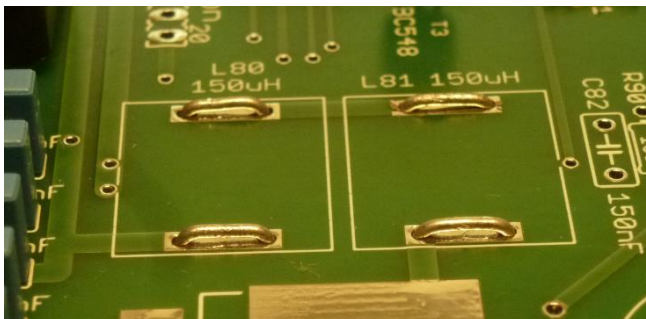
Note!

None of the transistors in the uTracer will get hot during operation, or conduct huge amounts of current. It is therefore sufficient to solder the tab only on one corner.

- The huge SIC transistors (SCT2750NY) will not get warm during operation nor will they conduct high currents. It is therefore sufficient to just solder them on one small corner of the backside tab. This will make it easier to remove the transistor later if needed. To make the solder flow, heat the tab until it is sufficiently hot. Use a solder flux pen or liquid when available (see Tip and Tricks 2).

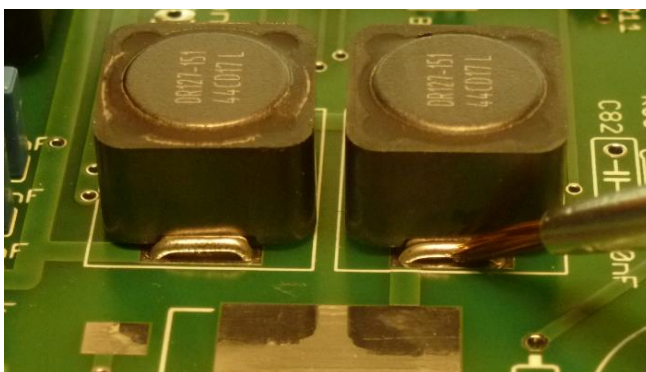


-

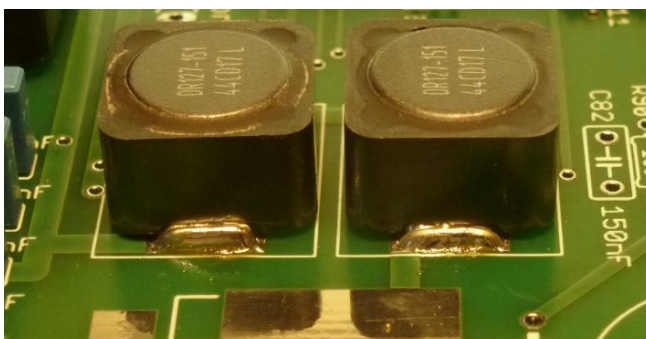


For the mounting of the 150uH inductors L60, L61, L80 ad L81 I recommend the procedure depicted on the left.

Bend four pieces of not too thin wire so that they fit in the holes as shown in the top picture.



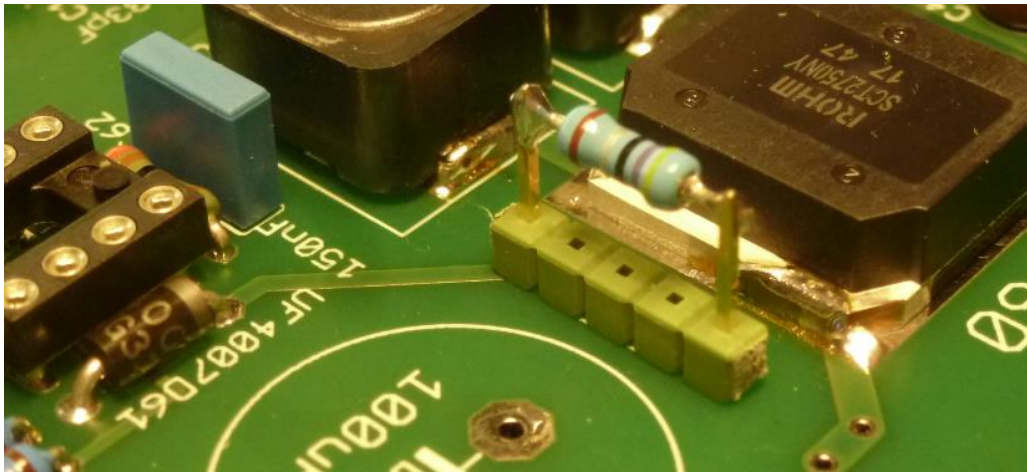
Position the inductors between the wires with the contacts facing the wires. Apply a small amount of soldering flux when available (middle).



Apply some solder and let the solder flow between the wire and the contact of the inductor (bottom).

Turn the board around and make sure the solder has flown all the way through the holes of the contact pads.

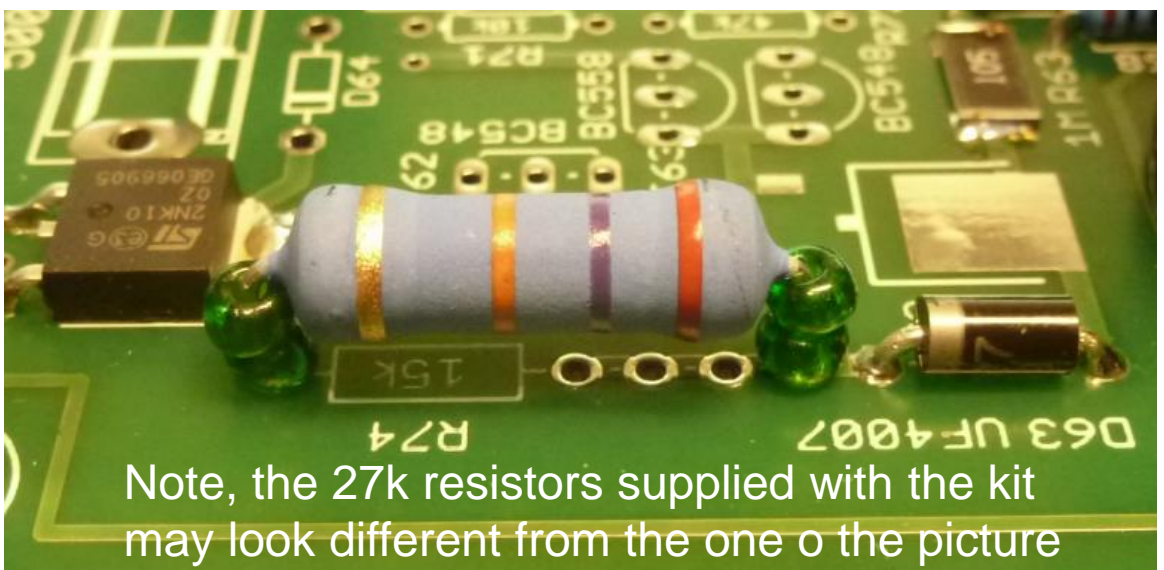
- To facilitate experimenting in the future with different current sense resistors R62 and R82, it can be handy to mount these resistors on a piece of modified pin header (not supplied with the kit).



- Use a ceramic bead on the anode side of diodes D60, D80. This will create space on the cathode side to position a test clip during calibration and testing.

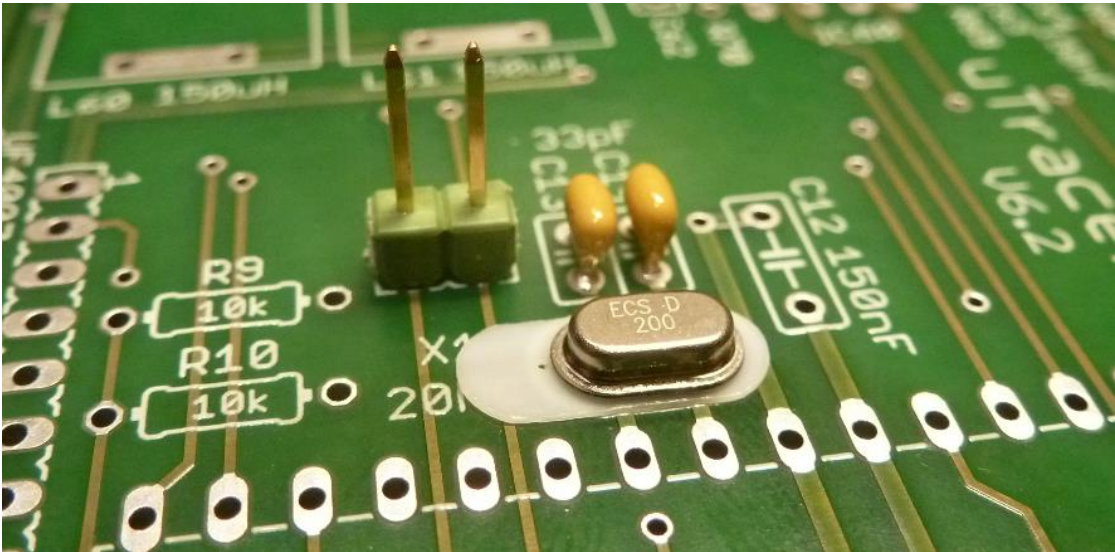


- The discharge resistors R74 and R84 can get quite warm during operation. Separate them from the PCB with two ceramic spacers per side.

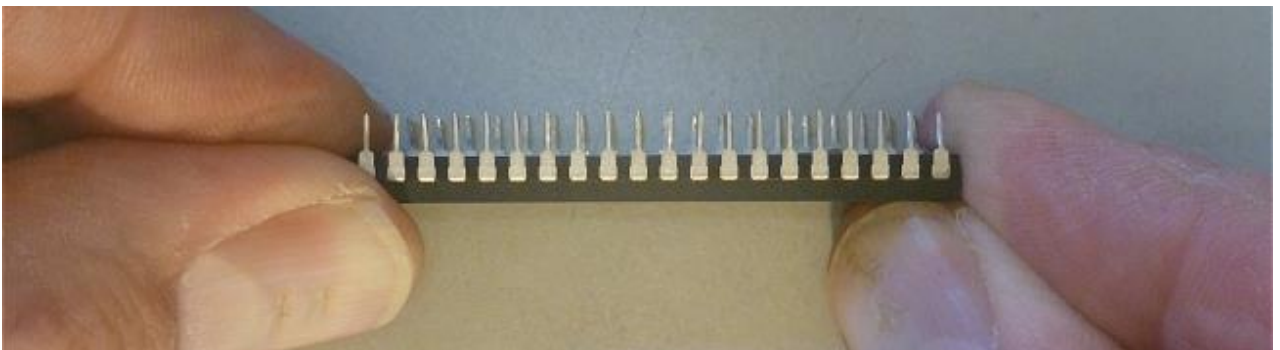


Note, the 27k resistors supplied with the kit may look different from the one on the picture

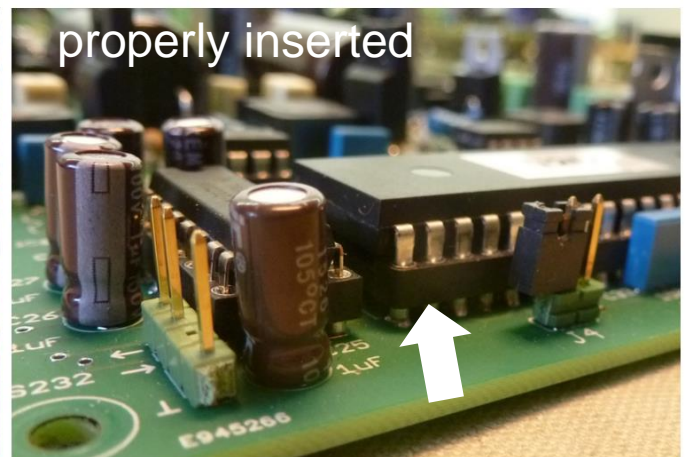
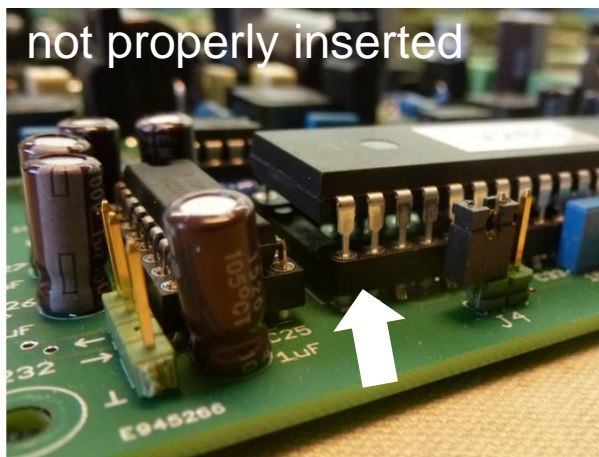
8. Use the small Teflon spacer provided to isolate the Xtal from the PCB.



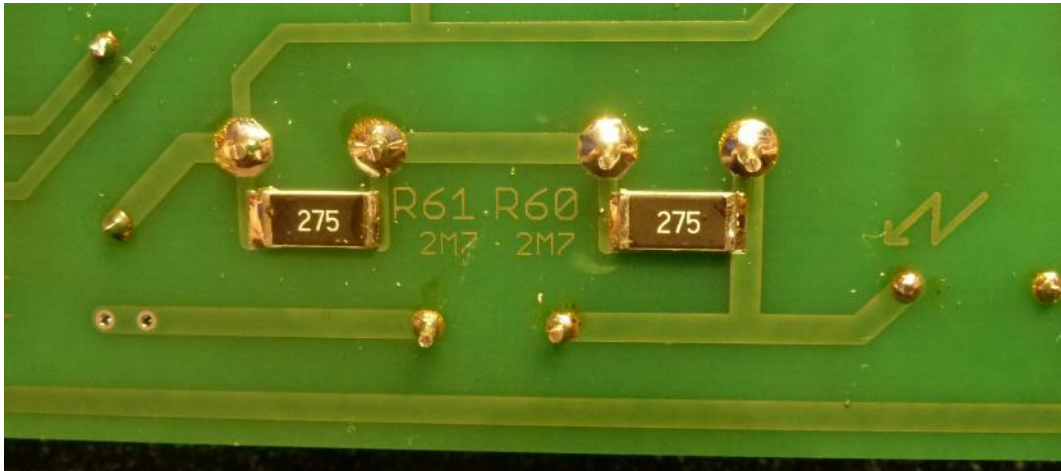
9 As usual, the pins of the PIC processor are at an angle so that the PIC doesn't fit directly in the 40 pin socket. The best way to adjust the angle of the pins is to hold the processor at the sides and correct the angle of 20 pins at once by bending them against a flat surface e.g. the table.



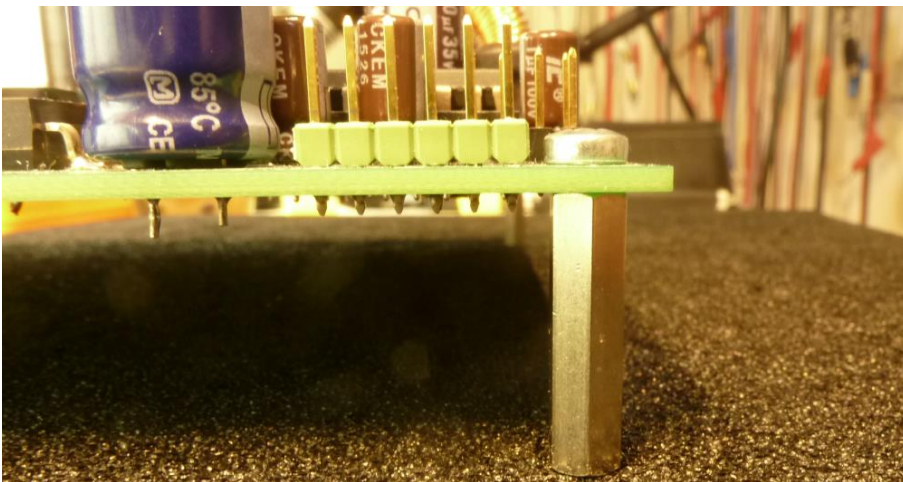
10. Make sure that you fully insert the PIC into the socket !! A not correctly (half) inserted PIC has given problems in a lot of instances causing faulty or erratic behavior. It requires the force of two thumbs to deeply and properly insert the PIC into the socket! (Picture taken from uTracer3+)



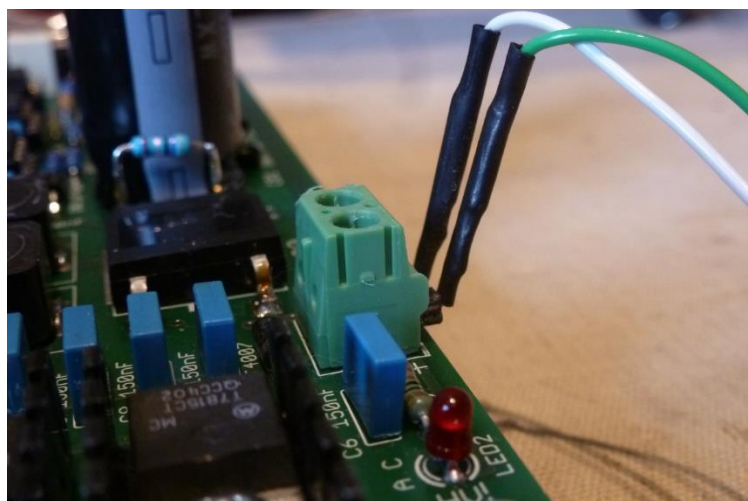
11. The four SMD type 2.7 Mohm resistors R60, R61, R80 and R81 are mounted on the backside of the PCB.



12. There are 6 standoffs supplied with the kit. Mount them underneath the PCB. This will make it easier to insert components and will make it safe to test the PCB with high voltages present on the bottom side.



- 13.. By using a pair of 90 degree bent clips it becomes impossible to accidentally exchange the + and - of your power supply.



Step-by-step construction of the uTracer

In the following pages we will go through the step-by-step construction and testing / calibration of the uTracer.

The uTracer construction manual consists of the following parts:

1. COM port installation;
2. The Graphical User Interface (GUI);
3. The RS232 cable;
4. The voltage regulators;
5. RS232 interface;
6. Microcontroller;
7. Heater supply;
8. Negative power supply;
9. Grid bias circuit;
10. High voltage Boost converters and current amplifiers;
11. High voltage switches.

Each part consists of a “construction” and a “testing” part.

In contrast to the uTracer3 the calibration is done after the construction has been completed. Note: normally very little calibration is needed.

Each part consists of a list of instructions preceded by a so that the line can be ticked off when it is executed.

Do not skip parts of the manual, because the testing sequence may be different in that case.

Normally it is recommended to first place “the lowest” components and then to go to “the higher” components. In this case the order may be different in some cases because it may be difficult to solder some components when others are already in place, or because of testing reasons.

The following SMD components have already been assembled on the PCB:

Grid section:

IC40 DAC8562
IC41 OPA2227
IC42 LTC6090

Anode section:

D62 BAT74
IC62 PGA113

Screen section:

D82 BAT74
IC82 PGA113

During every test/calibration step an indication for the current consumption is given, which may be used to detect problems.

For most of the resistors 1% types are supplied despite the fact that 5% types are specified.

For most resistors values some spares are supplied so don't worry if after construction of the uTracer you have some left.

Always check the resistance value of each resistor with a multimeter before you assemble it onto the board.

When starting the construction of a new part, first read through the whole section and familiarize yourself with the photo's. It can be that they clarify things that are otherwise perhaps difficult to understand from the text.

Some components may have been replaced by alternative types depending on availability from my supplier. Below you find a list of replacements used.

Component in circuit diagram	Replacement type(s)

Part 1. COM port installation (if needed)

The GUI communicates to the uTracer6 hardware via an “old fashioned” RS232 serial link.

If your PC or laptop does not have a native COM port you will need a USB to RS232 adaptor. If you do not have one already please buy a good quality one, preferably one that uses an FDTI chipset. Avoid very cheap Chinese devices!

Make sure that your converter supports RS232 level signals **and not simply TTL (5V) or 3V level signals.**

- 1. Buy a good quality USB-to-serial adapter.
- 2. Install the USB-to-serial driver if needed.
- 3. Make sure the communication protocol is set to 9600 baud, 1 stop bit, no parity check.
- 4. Be sure you know the COM port assigned to the adapter.

Part 2. The graphical user interface (GUI) installation

In order to install the GUI for the uTracer6, it is necessary to have the GUI for the uTracer3 installed first !!

If you do not have the GUI for the uTracer3 installed, continue with part 2a of this manual.

If you have already installed the uTracer3 GUI (any version), proceed to part 2b of this manual.

In the future I will make an installer to directly install the version 6 GUI, but for the moment this is the most reliable procedure.

If you encounter problems during installation, or if you want to install the GUI on a Mac or on a LINUX system, please first consult the frequently asked question (FAQ) page on the website.



“This software will help you manage stress as long as you don’t try to install it.”

Part 2a. The uTracer3 GUI – installation (if needed)

In order to be able to use the uTracer6 GUI, it is necessary to first install the GUI for the uTracer3. You will not be using it, but the uTracer6 GUI will need some software components that are installed during the uTracer3 installation. If you have already in the past installed the uTracer3 GUI you can skip this part and proceed to part 2b.

1. Create a folder to download the zipped file with the GUI into.

2. Download the GUI into the folder from:

http://dos4ever.com/uTracer3/uTracer3_pag10.html

3. Unzip the downloaded file into that folder.

4. Double click “setup”.

5. Follow the instructions of the setup program.

6. After successful installation of the uTracer3 GUI continue with

Part 2b of this manual.

Part 2b. The uTracer6 GUI – installation

In order to be able to use the uTracer6 GUI, it is necessary to first install the GUI for the uTracer3. You will not be using it, but the uTracer6 GUI will need some software components that are installed during the uTracer3 installation. If you do not have the uTracer3 GUI installed goto part 2a, otherwise proceed with this part.

Installation of the uTracer6 GUI:

- 1. Create a folder to download the zipped file with the GUI into.
- 2. Download the GUI for the uTracer6 into the folder from:
http://dos4ever.com/uTracer3/uTracer3_pag10.html
- 3. Unzip the downloaded file into that folder.
- 4. Double click the executable.

Part 2. The GUI – testing

For a 100% test of the functionality of the GUI the RS232 cable described in the next section is necessary. Without this cable a quick test is possible that will be described here.

- 1. Install GUI as described.
- 2. Start the GUI.
- 3. Open the “Debug/Communications” window by pressing “Debug” in the “Miscellaneous” section of the GUI.
- 4. Select the proper COM port (the port com number can be saved by pressing “safe to calibration file” in the “Calibration” form).
- 5. Press the “ping” command button.
- 6. The “Send string” and “Echo string” should now display:

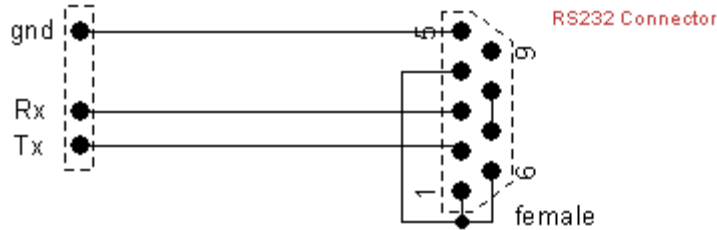


Explanation:
the GUI tries to send a “0” this zero is not echoed. After a few seconds the GUI detects that no character is returned resulting in the timeout message.

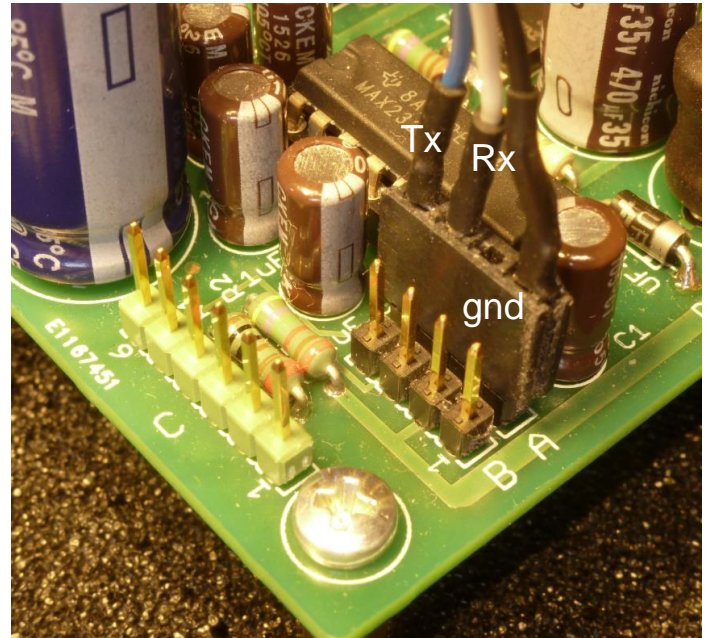
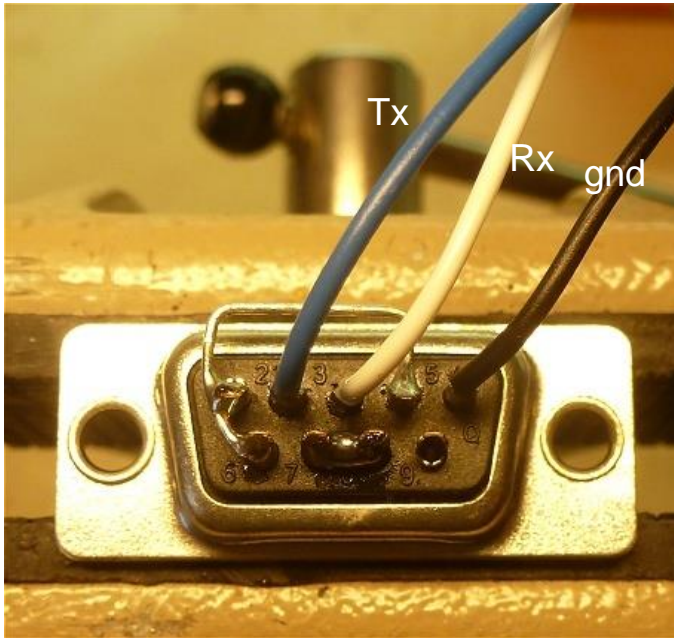
- 7. At no circumstance the message “Run time error 8020, Error reading comm device” should occur. When it does, there is a problem with the USB-to-serial adapter.
- 8. For a full check, connect the RS232 cable (next section) and perform the test described in that section.

Part 3. The RS232 connector cable - construction

I intentionally didn't integrate the RS232 connector on the PCB because most people will want to position that connector somewhere at the rear of their case. A cable connects the 9 pin D-SUB female connector located at the desired location to the RS232 connector on the PCB.

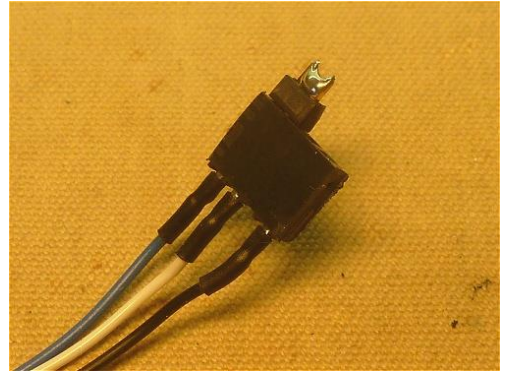


On the D-SUB connector pins 1,4,6 and pins 7,8 have been connected to "fool" the handshake signals. Strictly speaking this should not be necessary because the GUI does not check for these signals, but I am never sure what USB to serial adapters do?

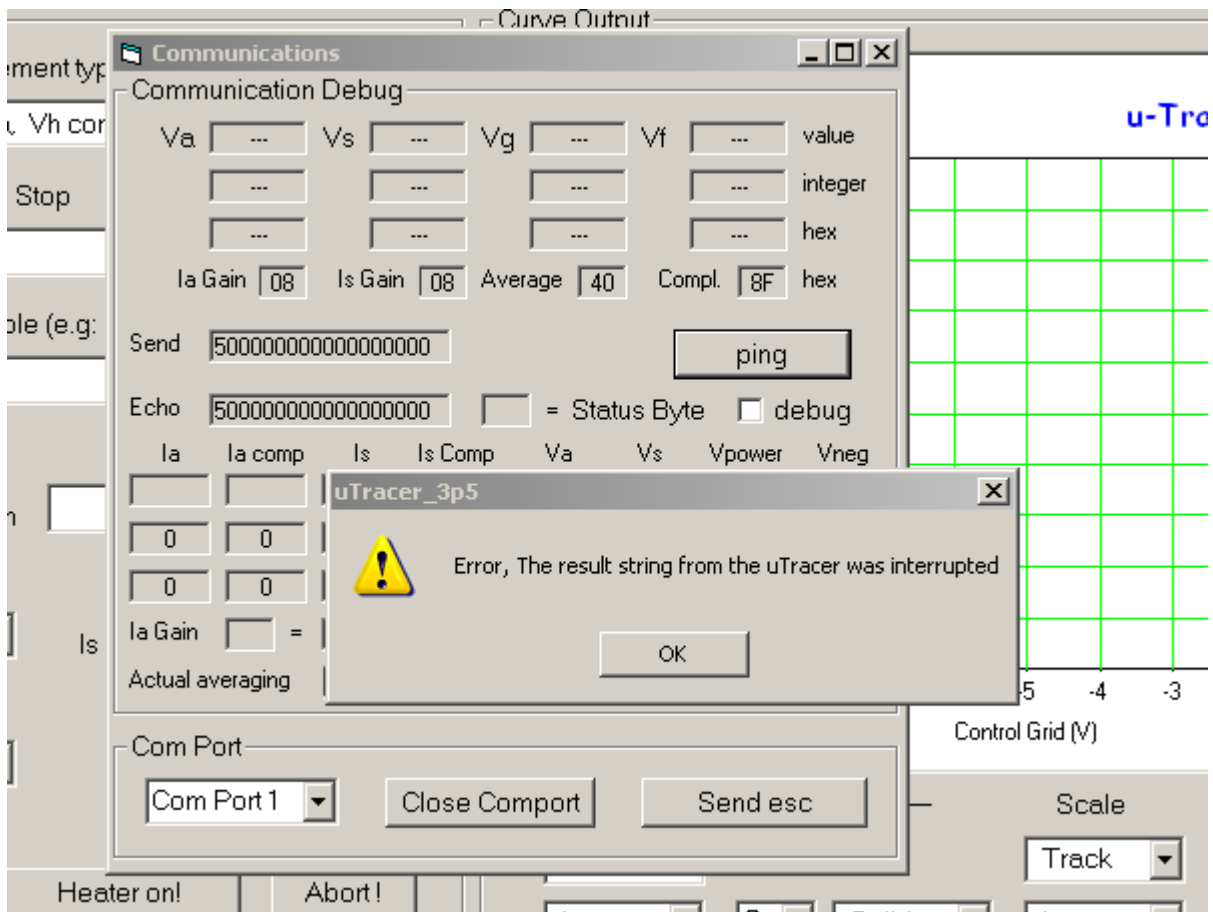


Part 3. The RS232 connector cable – testing

- 1. Make sure that the GUI is properly installed and tested.
- 2. Connect the RS232 cable to the PC.
- 3. Connect the Tx pin to the Rx pins.



- 4. Start the GUI.
- 5. Open the “Debug/Communications” window by pressing “Debug” in the “Miscellaneous” section of the GUI.
- 6. Press the “ping” command button.
- 7. The “Send string” and “Echo string” should now display:



- 8. After approximately 10sec the error message: “The result string from the uTracer was interrupted” should appear . This is the proper response.

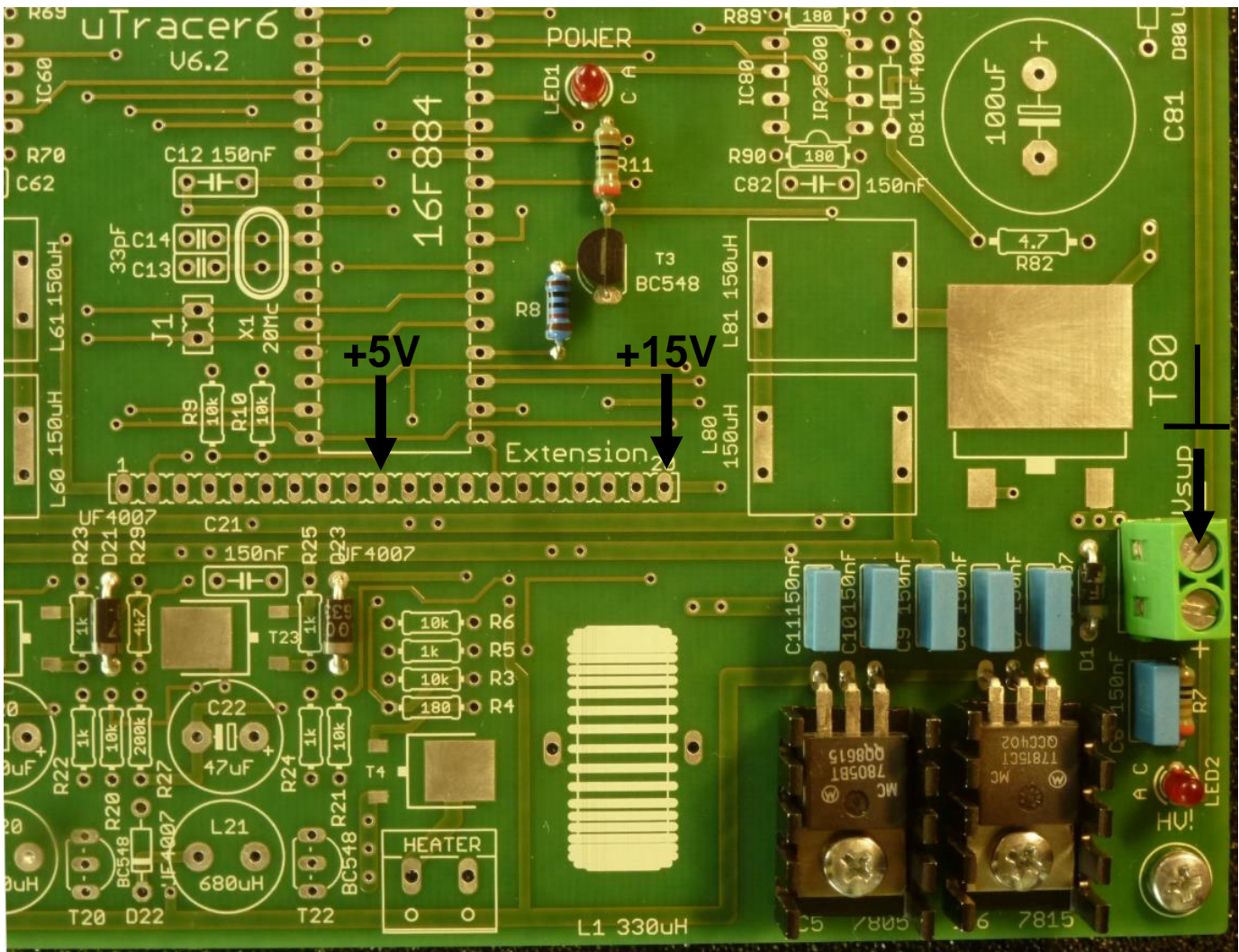
Part 4. The voltage regulators - construction

- 1. Position the 4 standoffs, see **“Tips and Tricks 12”**
 - 2. D1, diode UF4007 note polarity !
 - 3. D21, diode UF4007 note polarity !
 - 4. D23, diode UF4007 note polarity !
 - 5. R7, 1k
 - 6. R11, 1k
 - 7. LED1, note polarity ! → A = anode = long pin
 - 8. LED2, note polarity ! → A = anode = long pin
 - 9. C6, 150 nF
 - 10. C7, 150 nF
 - 11. C8, 150 nF
 - 12. C9, 150 nF
 - 13. C10, 150 nF
 - 14. C11, 150 nF
 - 15. R8, 10k
 - 16. T3, BC548 note orientation !
 - 17. IC6, 15V regulator, LM7815, use heatsink
 - 18. IC5, 5V regulator, LM7805, use heatsink
 - 19. 2 screw power terminal
- } Failing to install these at this stage will cause damage to the SMDs during the first tests !

Part 4. The voltage regulators - testing

Both LEDs are off during this test.

- 1. Connect the uTracer to a power supply ca. 19.5V, if possible limit the current to ca. 100 mA.
- 2. Switch the power supply on (supply current $\approx 25 \text{ mA @ } 19 \text{ V}$).
- 3. Both LEDs are off.
- 4. Touching the leads of R8 should light up the HV LED (LED1) weakly.
- 5. Check +15 V on pin 20 of the extension connector.
- 6. Check + 5 V on pin 10 of the extension connector.

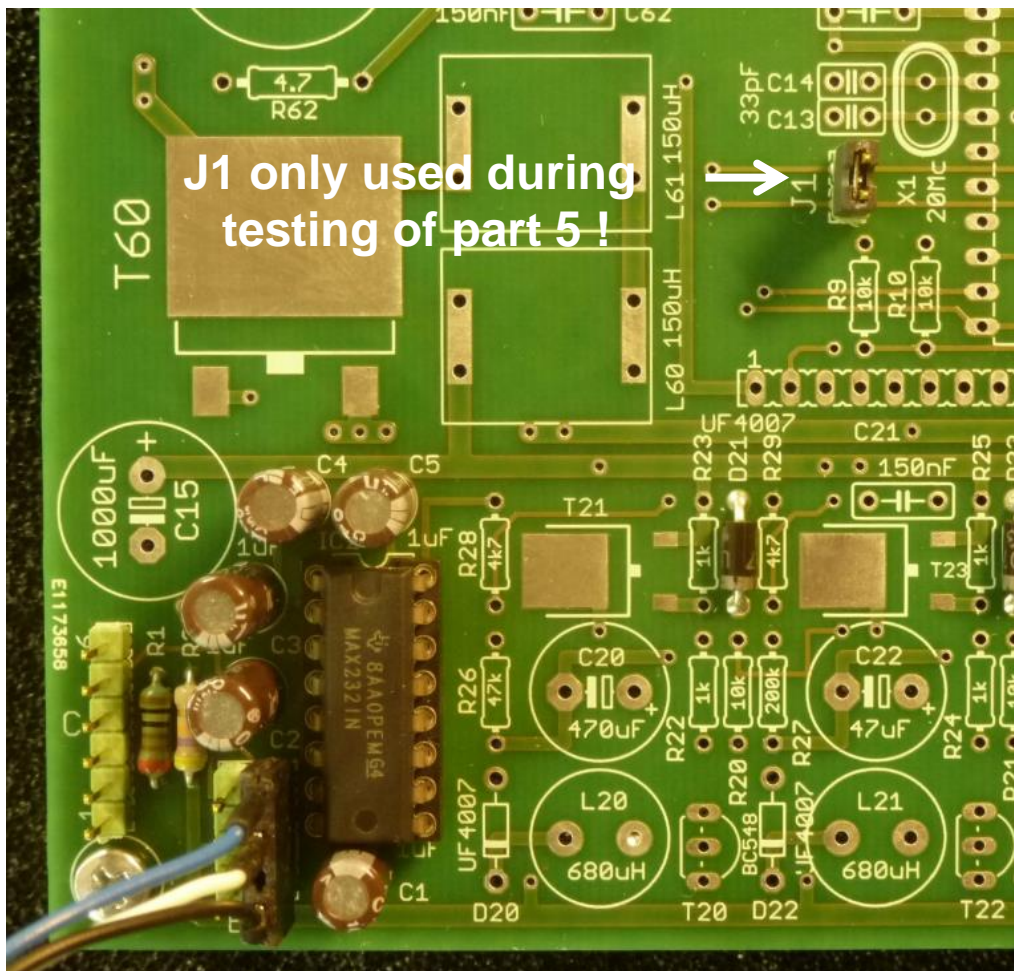


Overview of the PCB after completion of part 4 of the construction, and the location of the test points. Make sure that during testing your measurement probes do not slip!

Part 5. RS232 Interface - construction

- 1. R1, 1k
- 2. R2, 47k
- 3. 16 pin IC socket for IC2, note orientation !
- 4. 3 pin serial interface male pin header A
- 5. 4 pin interface male pin header B
- 6. 6 pin interface male pin header C
- 7. C1, 1uF / 50V, note polarity !
- 8. C2, 1uF / 50V, note polarity !
- 9. C3, 1uF / 50V, note polarity !
- 10. C4, 1uF / 50V, note polarity !
- 11. C5, 1uF / 50V, note polarity !
- 12. 2 pin male jumper pin header J1
- 13. Insert IC2, MAX232, note polarity !

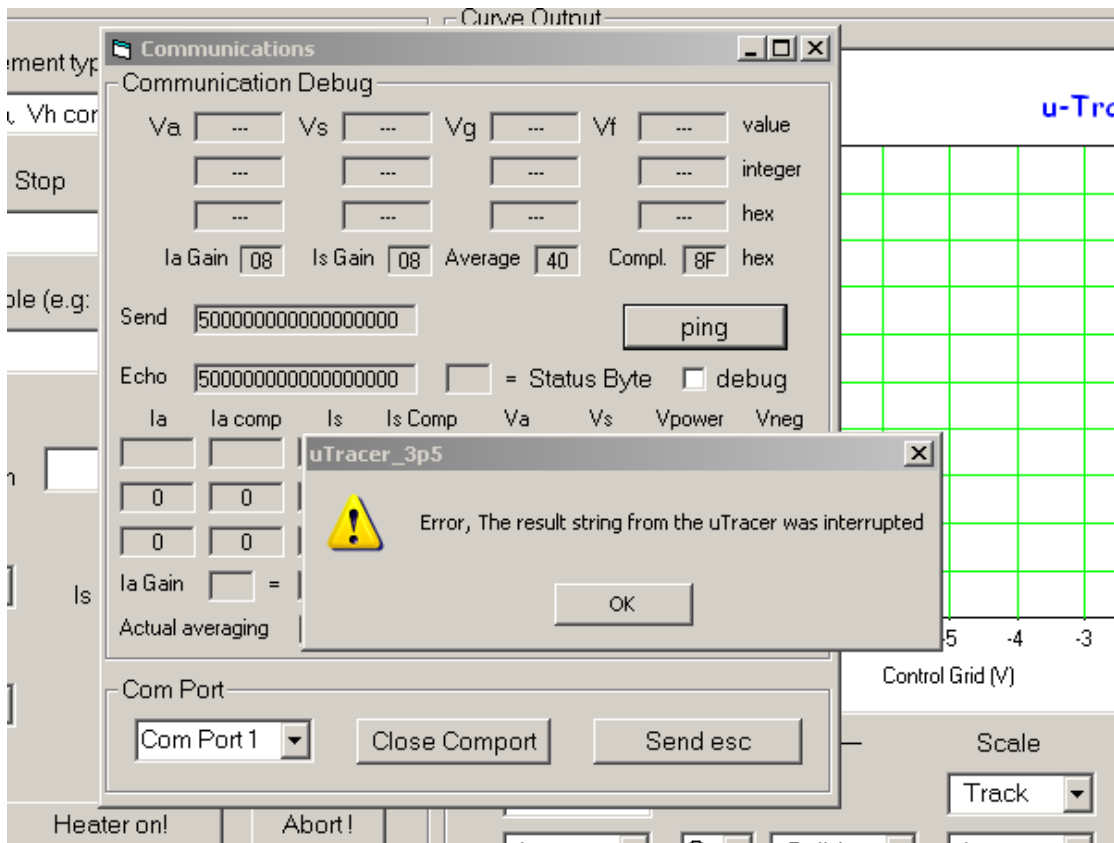
The RS232 interface after completion of part 5.



Part 5. RS232 Interface - testing

Both LEDs are off during this test.

- 1. Make sure that the GUI is properly installed and tested.
- 2. Connect the uTracer to the PC.
- 3. Connect the uTracer to the power supply, if possible set current limit to 100 mA. (Note, both LEDs will be off)
- 4. Make sure jumper J1 is in place.
- 5. Start the GUI.
- 6. Switch on the power supply (supply current $\approx 35 \text{ mA @ } 19 \text{ V}$).
- 7. Open the “Debug/Communications” window by pressing “Debug” in the “Miscellaneous” section of the GUI.
- 8. Press the “ping” command button.
- 9. The “Send string” and “Echo string” should now display:

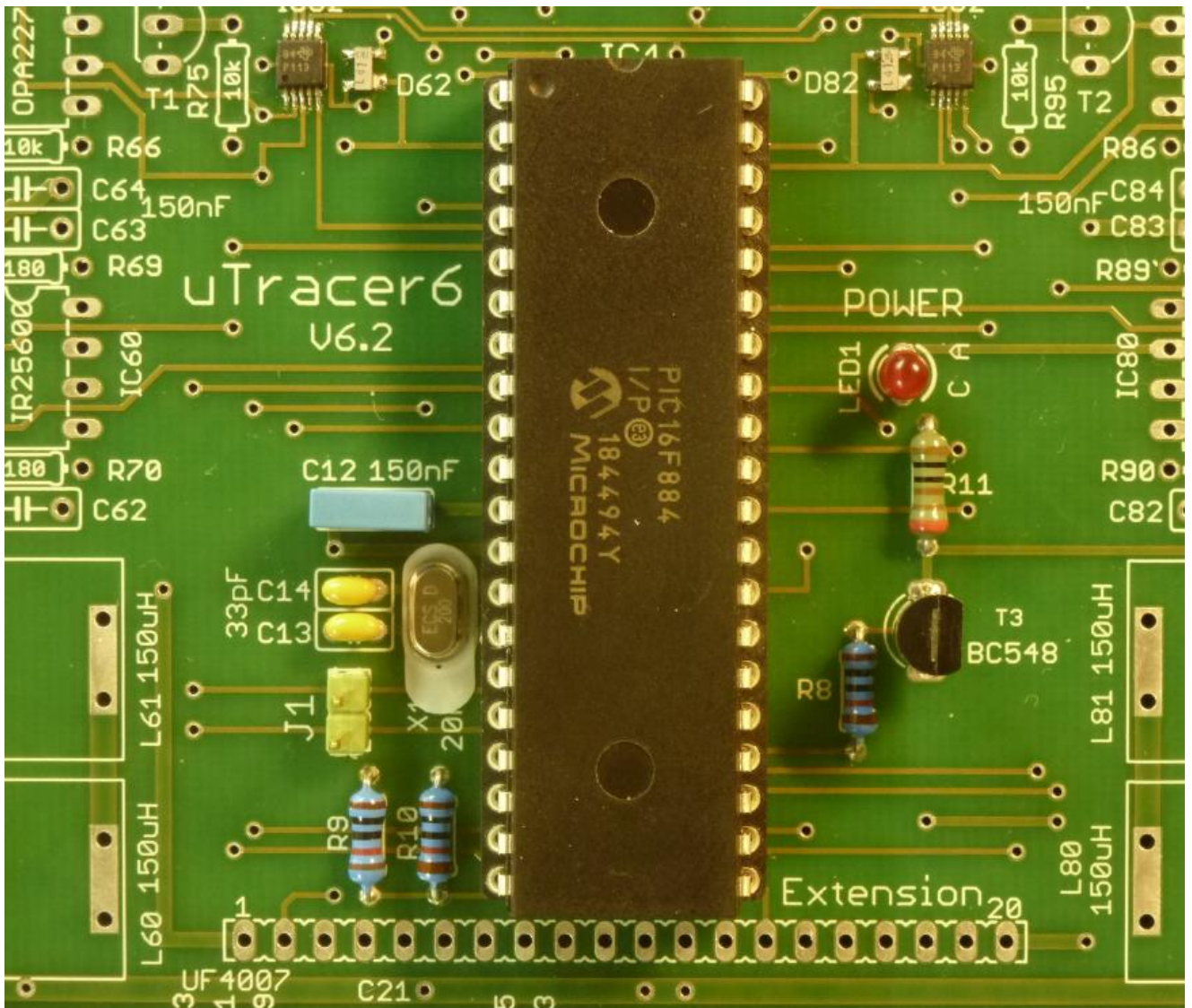


- 10. After approximately 10 sec the error message: “The result string from the uTracer was interrupted” should appear . This is the proper response.
- 11. Be sure to remove jumper J1 !! ← !

Part 6. Micro-controller - construction

- 1. Make sure jumper J1 is removed !
- 2. C13, 33 pF
- 3. C14, 33 pF
- 4. X1, 20 MHz Xtal
- 5. C12, 150 nF
- 6. R9, 10 k
- 7. R10, 10 k
- 8. R6, 10 k
- 9. R5, 1 k
- 10. 40 pin IC socket for IC1, note orientation !
- 11. Insert IC1, PIC16F884 **NOTE ORIENTATION** see “Tips and Tricks” no. 9

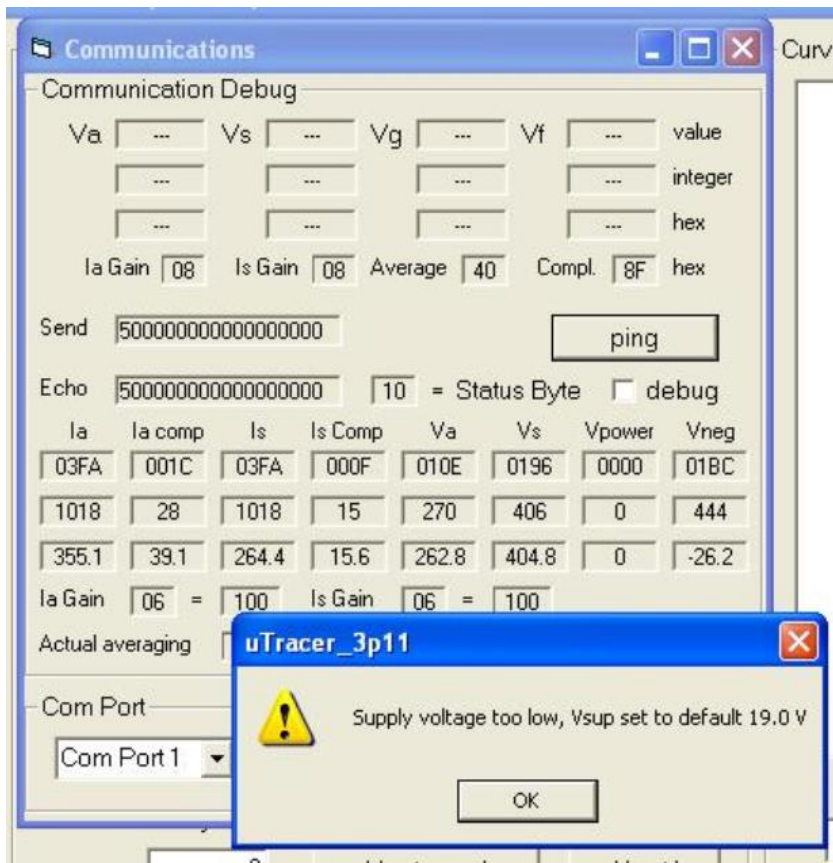
The Xtal is located on the conductive foam.
Insert plastic isolator between Xtal and PCB,
see “Tips and Tricks” no 8 !



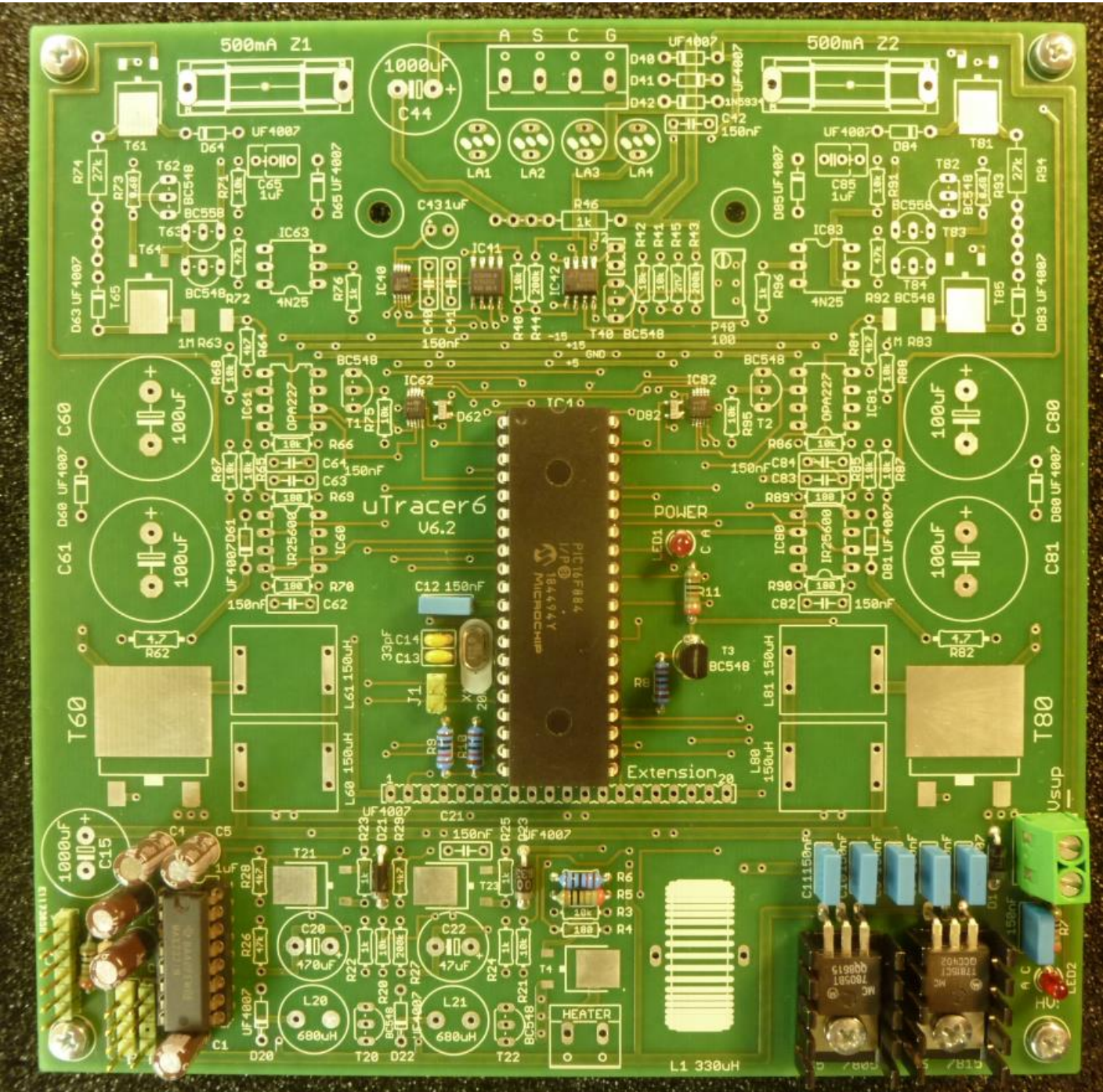
Part 6. Micro-controller - testing

Both LEDs are off during this test.

- 1. Make sure that the GUI is properly installed and tested.
- 2. Connect the uTracer to the PC.
- 3. Connect the uTracer to the power supply, if possible set current limit to 100 mA.
- 5. Start the GUI.
- 6. Switch on the power supply (supply current ≈ 70 mA @ 19,5 V).
- 7. Open the “Debug/Communications” window by pressing “Debug” in the “Miscellaneous” section of the GUI.
- 8. Press the “ping” command button.
- 9. The “Send string” and “Echo string” should now display:



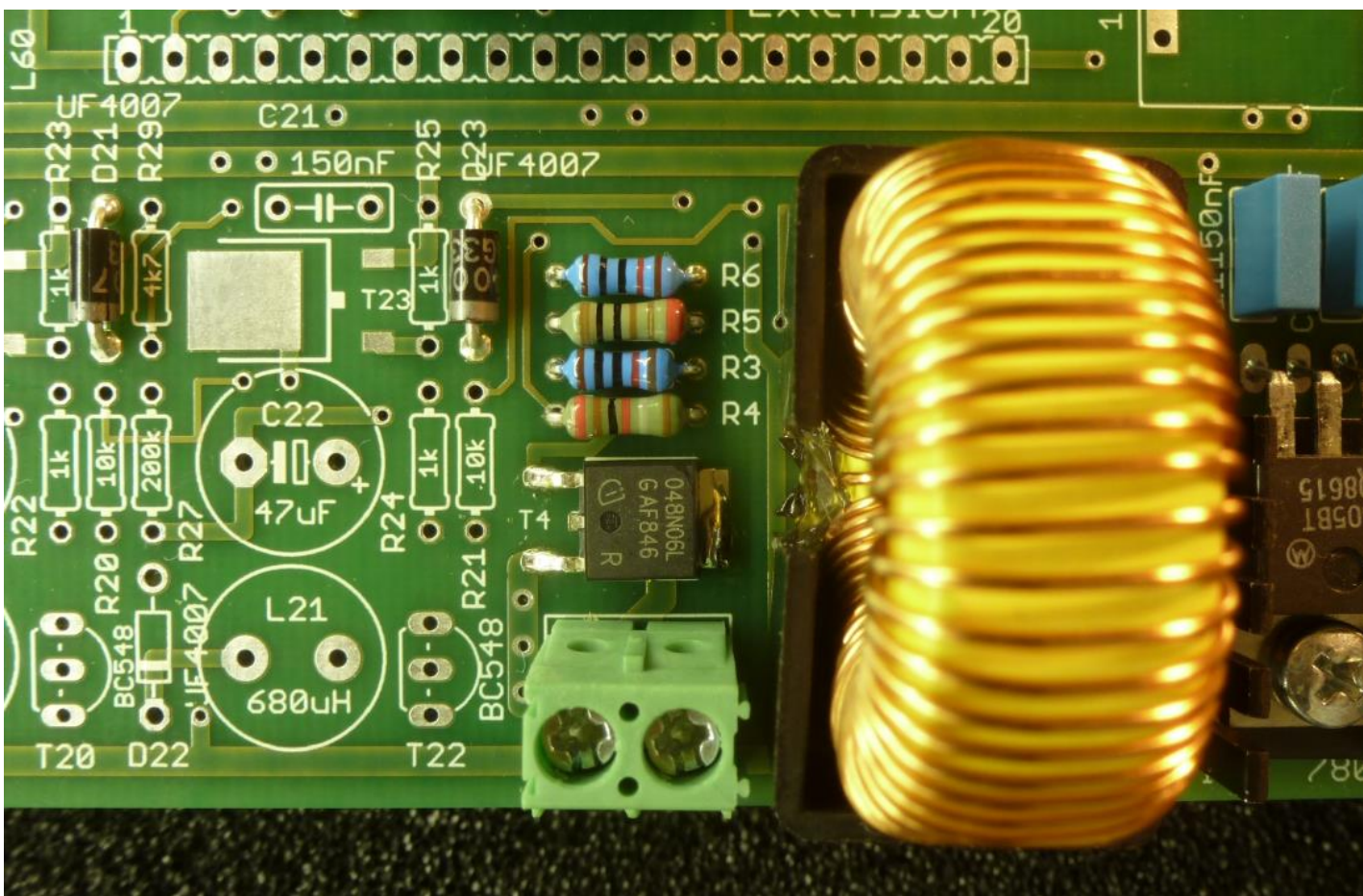
- 10. You will receive the message “Supply voltage too low” ignore this message at this stage.
- 11. It is possible that the high voltage LED will now be on.



Overview of the PCB after part 6.

Part 7. Heater supply - construction

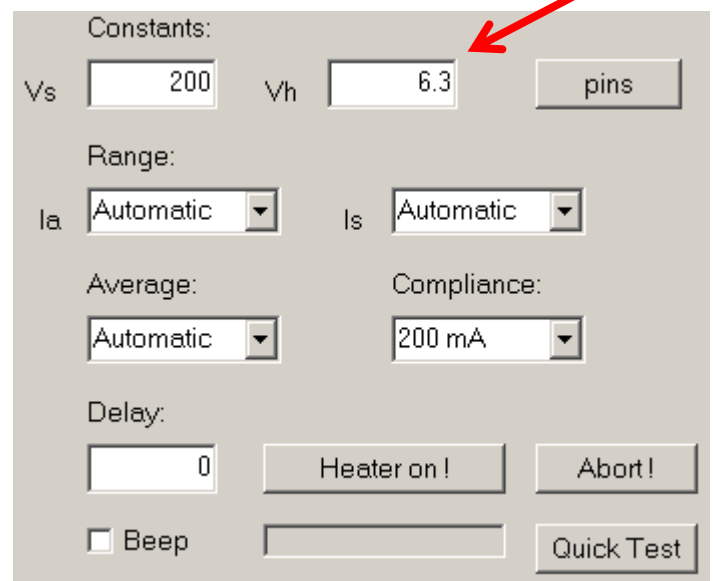
- 1. R3, 10 kohm
- 2. R4, 180 ohm
- 3. T4, MOSFET IPD048N06L3 see “Tip & Tricks” no. 2
- 4. 2 screw terminal for heater connection.
- 5. C15, 1000uF / 50V note polarity !
- 6. L1, 330 uH



Detail of the board after construction of part 7.

Part 7. Heater supply - testing

- 1. Make sure that the GUI is properly installed and tested.
- 2. Connect the uTracer to the PC.
- 3. Connect the uTracer to the power supply, if possible set current limit to 500 mA. The supply current without load is ≈ 70 mA.
- 4. Check that the power LED is now on.
- 5. Connect a lamp 6 or 12 volts / ≈ 100 mA to the heater terminals.
- 6. Start the GUI.
- 7. Switch-on the power supply of the uTracer.
- 8. Enter the voltage of the lamp in the designated field (arrow).
- 9. Press “Heater on!”
- 10. The lamp should now be switched on in 10 sec.
- 11. Press “Abort!” to switch off the lamp.
- 12. Replace the lamp for “a heavier” lamp of 1 to 1.5 A. (Increase the current limit of your supply to 1A or use the final laptop adaptor.)
- 13. Repeat steps 7 to 10.



Constants:
Vs Vh pins

Range:
Ia Is

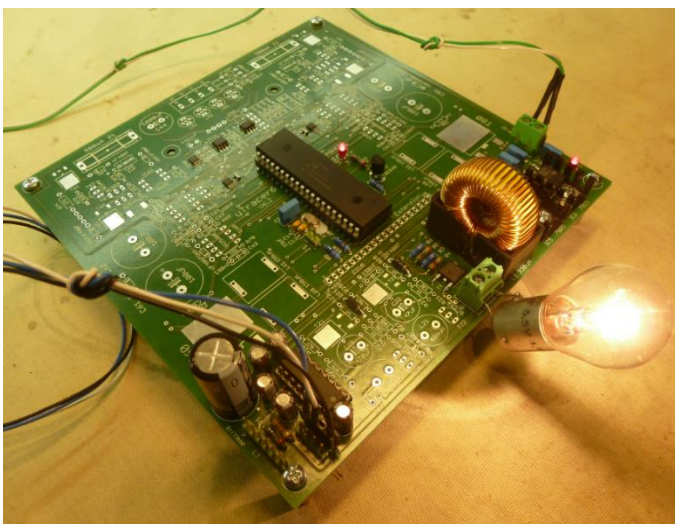
Average:

Compliance:

Delay:
 Heater on! Abort!

Beep Quick Test

The power MOSFET should remain cool even for a 1.5 A lamp.



Note:

Because the heater voltage is a PWM signal it cannot be measured using a normal voltmeter! The reading will not reflect the proper heater voltage!

See weblog for more info:

<http://www.dos4ever.com/uTracerlog/tubetester2.html#heater>

Part 8. Negative power supplies - construction

- 1. R20, 10 k
- 2. R21, 10 k
- 3. R22, 1 k
- 4. R23, 1 k
- 5. R24, 1 k
- 6. R25, 1 k
- 7. R26, 47 k
- 8. R27, 200 k
- 9. R28, 4.7 k
- 10. R29, 4.7 k



please work very carefully, a mistake is easily made and can take ages to find !

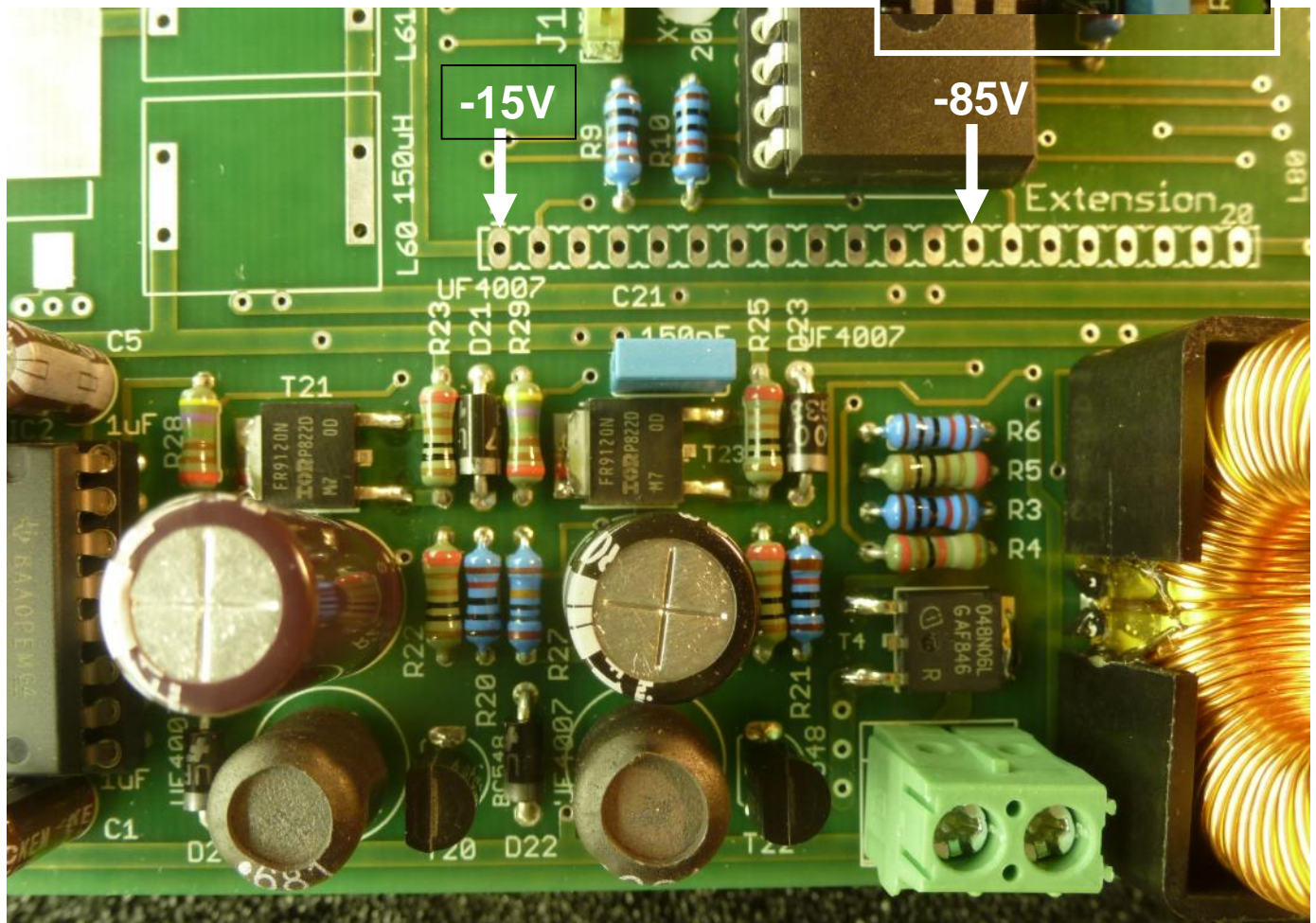
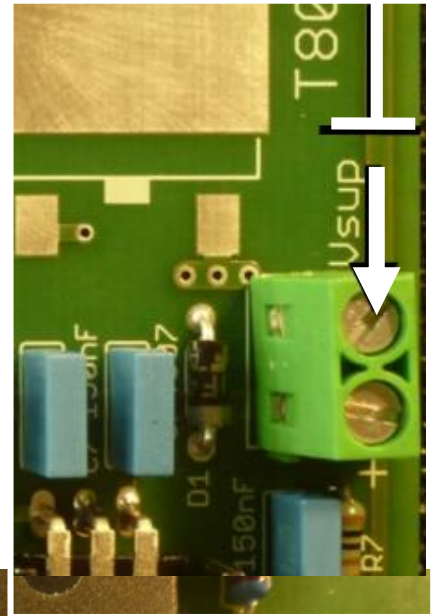
- 11. D20, diode UF4007, note polarity !
- 12. D22, diode UF4007, note polarity !
- 13. T21, MOSFET IRFR9120
- 14. T23, MOSFET IRFR9120 } **see Tip & Tricks no. 2**
- 15. C21, 150 nF
- 16. T20, transistor BC548, note orientation !
- 17. T22, transistor BC548, note orientation !
- 18. L20, inductor 680 uH
- 19. L21, inductor 680 uH
- 20. C20, 470 uF / 35V, note polarity !
- 21. C22, 47 uF / 100V, note polarity !

Part 8. Negative power supplies - testing

- 1. Connect the uTracer to the power supply, if possible set current limit to 500 mA. (No need to connect the uTracer to the PC for this test) (supply current $\approx 80 \text{ mA @ } 19.5 \text{ V}$)
- 2. Measure the $\approx -15 \text{ V}$ on pin 1 of the extension connector.
- 3. Measure the $\approx -85 \text{ V}$ on pin 13 of the extension connector.



Be careful not to slip with the test pins!

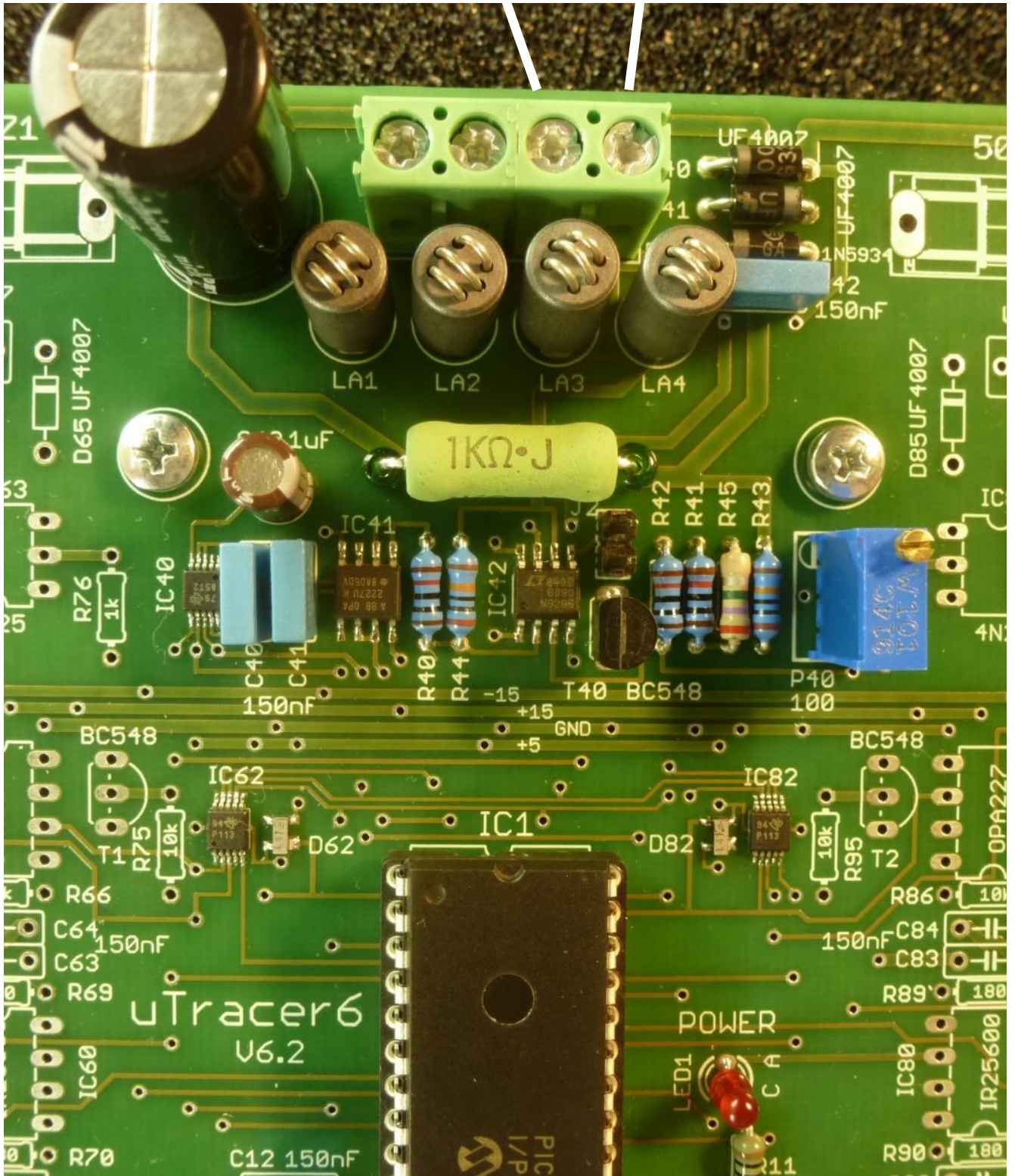


The board after construction of part 8.

Part 9. Grid bias circuit - construction

- 1. D40, UF4007, note polarity !
- 2. D41, UF4007, note polarity !
- 3. D42, 1N5934, note polarity !
- 4. C40, 150 nF
- 5. C41, 150 nF
- 6. C42, 150 nF
- 7. C43, 1 uF / 50V, note polarity !
- 8. LA1 ferrite RFI suppression bead
- 9. LA2 ferrite RFI suppression bead
- 10. LA3 ferrite RFI suppression bead
- 11. LA4 ferrite RFI suppression bead
- 12. 4 terminal screw block
- 13. R40, 10k
- 14. R41, 10k
- 15. R42, 10k
- 16. R43, 200k
- 17. R44, 200k
- 18. R45, 2M7
- 19. Male jumper J2
- 20. T40, transistor BC548, note orientation!
- 21. P40, 100 ohm, trimmer
- 22. R46, resistor 1 k / 3 Watt, use two ceramic beads per side
- 23. C44, 1000 uF / 50V

cathode to - DVM
grid to + DVM



The board after construction of part 9

Part 9. Grid bias circuit – testing

- 1. Make sure that the GUI is properly installed and tested.
- 2. Connect the uTracer to the PC.
- 3. Connect a DVM or multimeter between the cathode and grid terminals as indicated in the figure on the previous page.
- 4. Connect the uTracer to the power supply, if possible set current limit to 200 mA.
- 5. Switch on the power supply (supply current $\approx 110 \text{ mA @ } 19.5 \text{ V}$).
- 6. The grid voltage should be approximately $0 \text{ V} \pm 100 \text{ mV}$. During the calibration in the next section, it will be adjusted to exactly 0 mV .
- 7. In the main form of the GUI select the measurement type indicated below and fill in -80 V as start value for the grid bias.

u-Tracer (V3.12.6)

Measurement Set-Up

Select Measurement type:
I (Vg, Va) with Vs, Vh constant

Start Stop Nintervals
Vg log

Stepping Variable (e.g. 15, 20, 25 ...)
Va

Constants:
Vs Vh pins

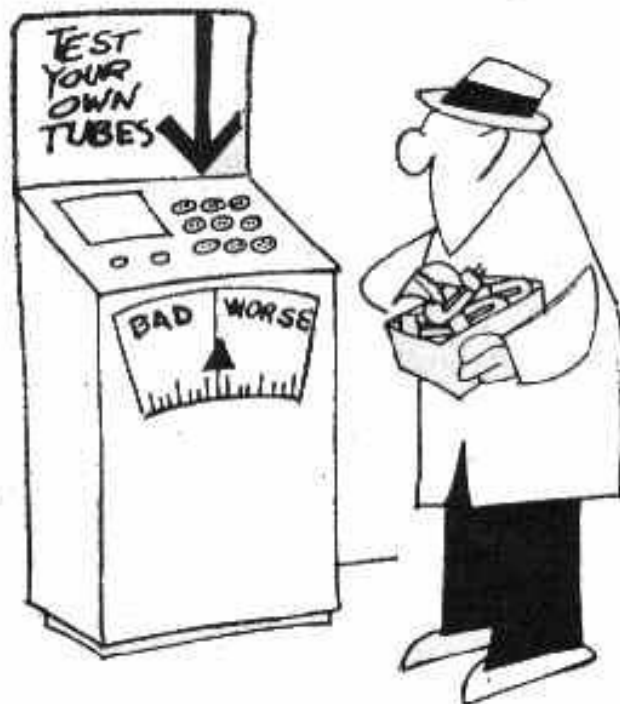
Range:
Ia Is

Average: Compliance:
 Vg low

Delay:
 Heater on! Abort!

Beep Quick Test

- 8. Press “Heater On!” in the main form.
- 9. To skip the “slow heating” feature press again on “Heating ...”
- 10. Start a measurement by pressing “Measure Curve.”
- 11. Read the actual grid voltage on the DVM or multimeter. The grid voltage should be around 80 V +/- a few volts. Again the exact calibration will be performed after the complete construction is finished.



Part 10. Boost converters - construction

- 1. R80, 2M7 SMD resistor
- 2. R81, 2M7 SMD resistor
- 3. R60, 2M7 SMD resistor
- 4. R61, 2M7 SMD resistor
- 5. L60, 150 uH inductor
- 6. L61, 150 uH inductor
- 7. L80, 150 uH inductor
- 8. L81, 150 uH inductor
- 9. R65, 10k
- 10. R66, 10k
- 11. R67, 10k
- 12. R68, 10k
- 13. R85, 10k
- 14. R86, 10k
- 15. R87, 10k
- 16. R88, 10k
- 17. R69, 180 ohm
- 18. R70, 180 ohm
- 19. R89, 180 ohm
- 20. R90, 180 ohm
- 21. D60, UF4007, note polarity ! + use ceramic bead on one side
- 22. D61, UF4007, note polarity ! **see Tip & Tricks 6**
- 23. D80, UF4007, note polarity ! + use ceramic bead on one side
- 24. D81, UF4007, note polarity ! **see Tip & Tricks 6**
- 25. 8 pin socket for IC60, note orientation !
- 26. 8 pin socket for IC61, note orientation !
- 27. 8 pin socket for IC80, note orientation !
- 28. 8 pin socket for IC81, note orientation !

These resistors are placed
on the backside of the PCB!

see Tip & Tricks 11



see Tip & Tricks 4

- 29. C62, 150 nF
- 30. C63, 150 nF
- 31. C64, 150 nF
- 32. C82, 150 nF
- 33. C83, 150 nF
- 34. C84, 150 nF
- 35. T60, SIC MOSFET SCT2750NY
- 36. T80, SIC MOSFET SCT2750NY
- 37. R62, 4.7 ohm
- 38. R82, 4.7 ohm
- 39. R64, 4k7
- 40. R84, 4k7
- 41. R63, 1Mohm
- 42. R83, 1Mohm
- 43. T61, MOSFET STD2NK100Z
- 44. T81, MOSFET STD2NK100Z
- 45. D63, UF4007, note polarity !
- 46. D83, UF4007, note polarity !
- 47. R74, 27 k / 3Watt, use 2 ceramic beads
- 48. R94, 27 k / 3Watt, use 2 ceramic beads
- 49. C60, 100 uF / 500 V, note polarity !
- 50. C61, 100 uF / 500 V, note polarity !
- 51. C80, 100 uF / 500 V, note polarity !
- 52. C81, 100 uF / 500 V, note polarity !
- 53. Insert IC60, IR25600, note orientation !
- 54. Insert IC80, IR25600, note orientation !
- 55. Insert IC61, OPA227, note orientation !
- 56. Insert IC81, OPA227, note orientation !

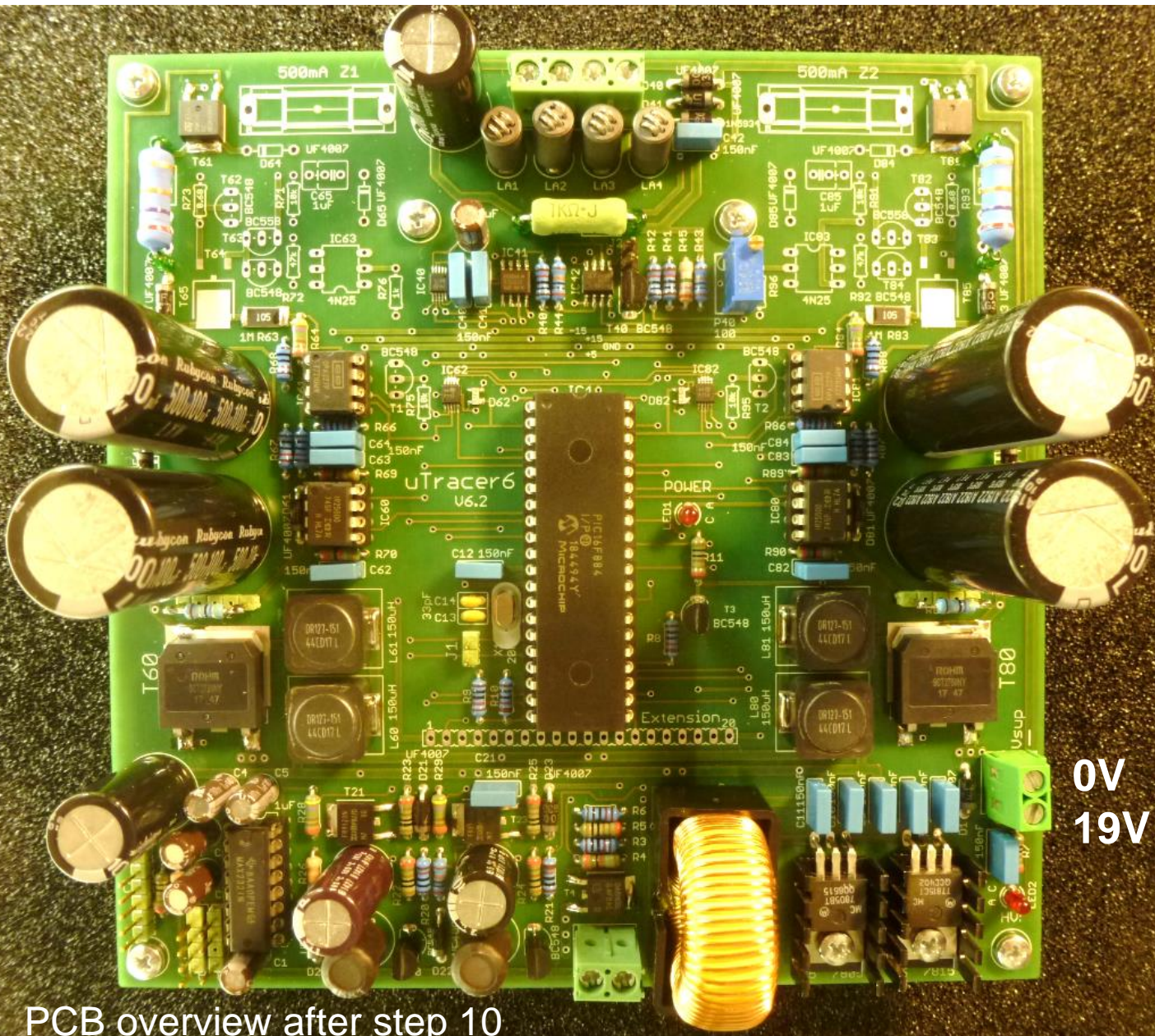
see Tip & Tricks 3

see Tip & Tricks 5

SMD type resistors !

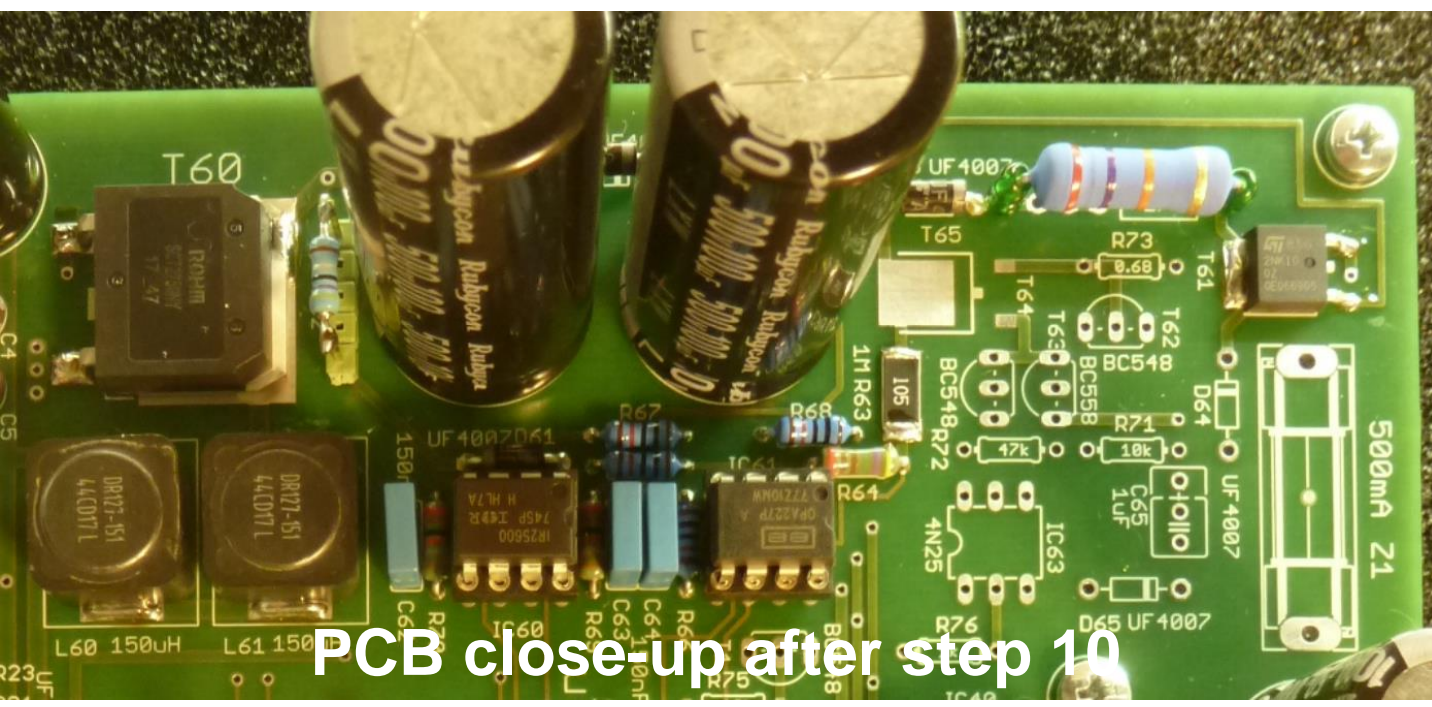
see Tip & Tricks 2

see Tip & Tricks 7



0V
19V

PCB overview after step 10



PCB close-up after step 10

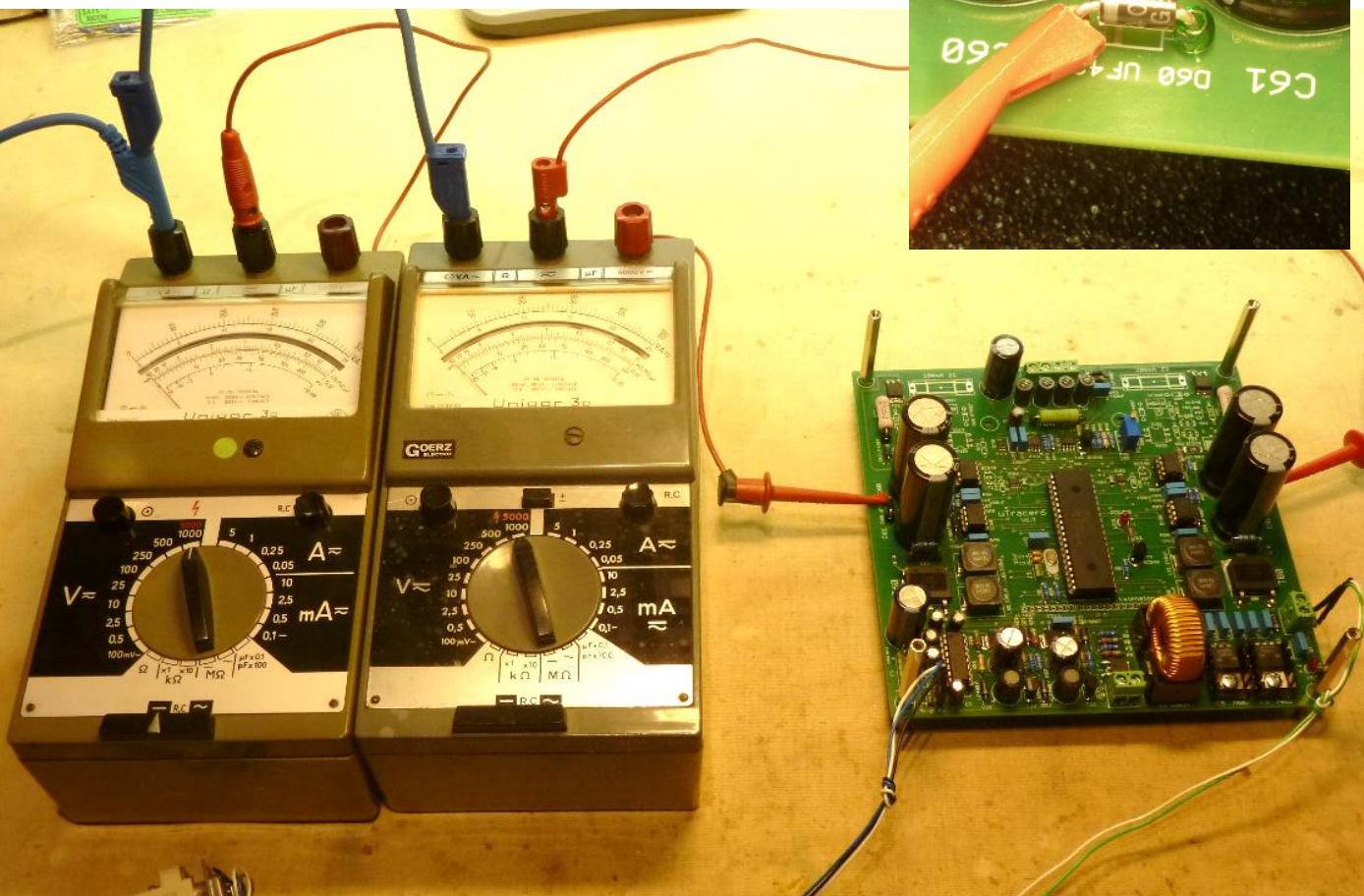
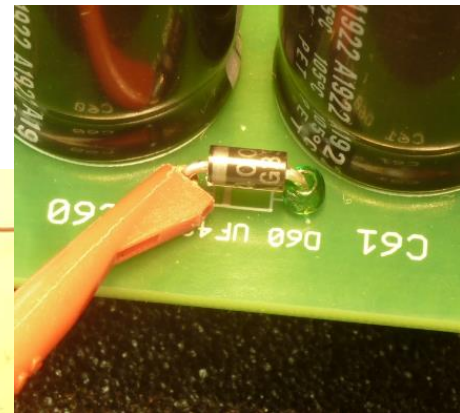
Part 10. Boost converters - testing

From this point onward there are potentially lethal voltages present on the board !!

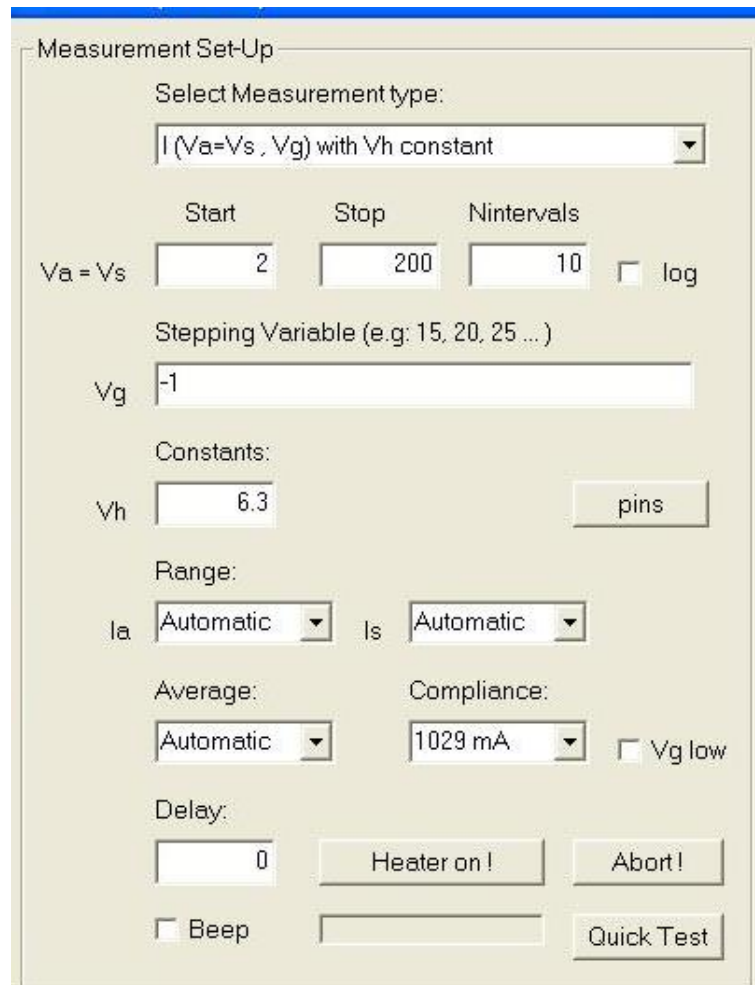
After a test / measurement, before working with the board:

1. Wait until the HV LED is off;
2. Disconnect the power supply;
3. Check that all four 100 μ F / 500 V caps are discharged !

- 1. Make sure that the GUI is properly installed and tested.
- 2. Connect the uTracer to the PC.
- 3. Connect a DVM or multimeter between the cathode of diode D60 and ground, and a second DVM or multimeter between the cathode of D80 cathode and ground. If you only have a single DVM or meter do the measurements after each other.



- 4. Connect the uTracer to the power supply, if possible set current limit to 1.5A, or if your power supply cannot deliver such high currents you can connect the uTracer directly to the laptop supply.
- 5. Switch on the power supply (supply current $\approx 120 \text{ mA @ } 19.5 \text{ V}$).
- 6. Select the following measurement.



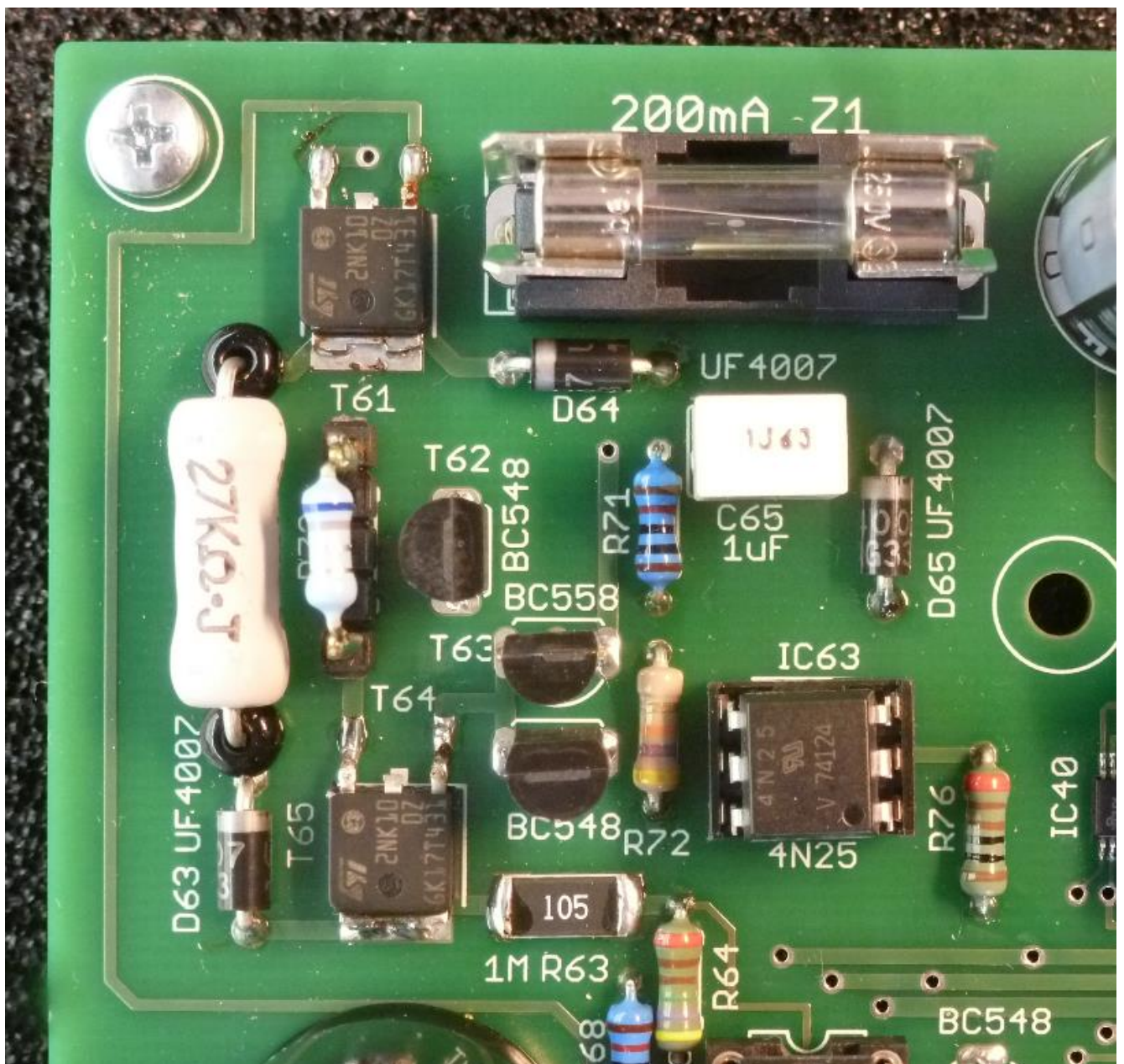
- 7. Press “Heater on!”
- 8. The text on the button now changes to “Heating.” Press the button again to skip the slow heating procedure, and finally press the button again to start a measurement.
- 9. The voltage on the caps should now increase in ten steps from approximately 20V (the supply voltage) to 220 V. If this does not happen stop here and first sort out what the problem is!
- 10. If this works you can enter a higher “Stop” voltage. Be careful, most modern DVMs can only handle a few hundreds of volts.

Part 11. HV switches - construction

- 1. R71, 10k
 - 2. R75, 10k
 - 3. R91, 10k
 - 4. R95, 10k
 - 5. R72, 47k
 - 6. R92, 47k
 - 7. R73, 0.56 ohm, use ceramic beads
 - 8. R93, 0.56 ohm, use ceramic beads
 - 9. R76, 1k
 - 10. R96, 1k
 - 11. D64, UF4007, note polarity !
 - 12. D65, UF4007, note polarity !
 - 13. D84, UF4007, note polarity !
 - 14. D85, UF4007, note polarity !
 - 15. 6 pin socket for IC63, note orientation !
 - 16. 6 pin socket for IC83, note orientation !
 - 17. T65, MOSFET 2NK100Z
 - 18. T85, MOSFET 2NK100Z
 - 19. T62, transistor BC548
 - 20. T64, transistor BC548
 - 21. T82, transistor BC548
 - 22. T84, transistor BC548
 - 23. T1, transistor BC548
 - 24. T2, transistor BC548
 - 25. T63, transistor **BC558**
 - 26. T83, transistor **BC558**
- } **see Tip & Tricks 2**
- } **Do not confuse BC548 with BC558**
- } **Do not confuse BC548 with BC558**

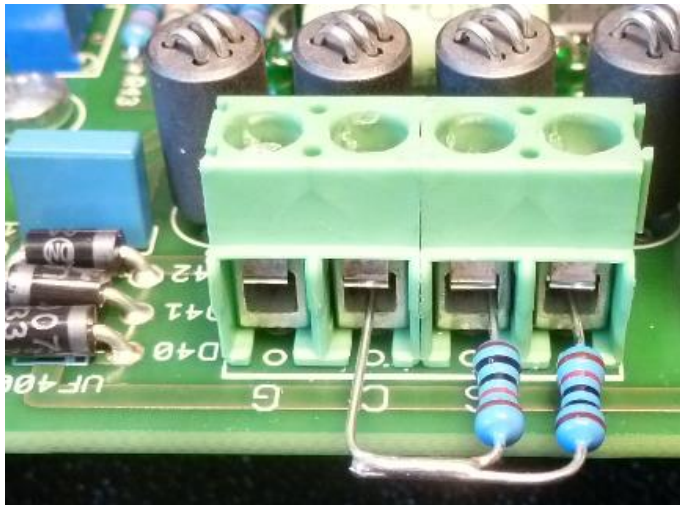


- ❑ 27. C65, 1 uF foil capacitor
- ❑ 28. C85, 1 uF foil capacitor
- ❑ 29. Fuse holder for Z1
- ❑ 30. Fuse holder for Z2
- ❑ 31. Insert IC63 optocoupler 4N25, note orientation !
- ❑ 32. Insert IC83 optocoupler 4N25, note orientation !
- ❑ 33. Insert fuse Z1
- ❑ 34. Insert fuse Z2



Part 11. HV switches - testing

- 1. Make sure that the GUI is properly installed and tested.
- 2. Connect the uTracer to the PC.
- 3. Connect a 10k ohm resistor between the anode and cathode terminals. and connect a second 10k resistor between the screen and cathode terminals. (see picture below).



- 4. Connect the uTracer to the power supply, if possible set current limit to 1.5A, or if your power supply cannot deliver such high currents you can connect the uTracer directly to the laptop supply.
- 5. Switch on the power supply (supply current ≈ 120 mA @ 19.5 V).
- 6. Select the measurement type as shown on the next page.
- 7. Plot the screen current on the second y-axis (see arrow).
- 8. Press “Heater on!”
- 10. The text on the button now changes to “Heating” Press the button again to skip the slow heating procedure and finally press the button again to start a measurement.
- 11. Verify that the uTracer6 measures two linear curves between 0 and approximately 20 mA (see picture next page).
- 12. Switch off the uTracer and keep the two 10kohm resistors for the calibration phase.

Measurement Set-Up

Select Measurement type:

I ($V_a = V_s$, V_g) with V_h constant

$V_a = V_s$ Start 2 Stop 200 Nintervals 10 log

Stepping Variable (e.g. 15, 20, 25 ...) V_g -1

Constants: V_h 6.3 pins

Range: I_a Automatic I_s Automatic

Average: Automatic Compliance: 1029 mA V_g low

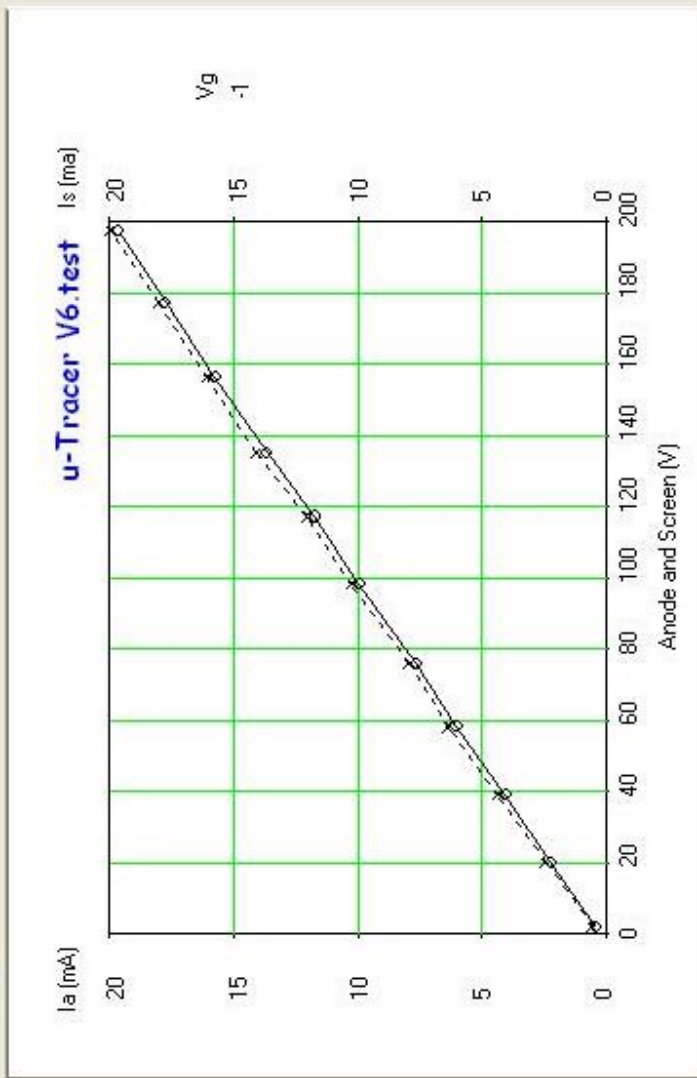
Delay: 0 Measure Curve Abort!

Beep Quick T

Miscellaneous

Ext. heater supply Debug Cal.

Curve Output



Axis: X $V_a = V_s$ Y1 I_a Y2 I_s Style: Dots Solid Track Scale: Track Min. 0 Max. 200 V Ticks 10

Keep Plot Distortion Pmax: 0 W Poly Degree: 3

Color Grid top graph off Title: Title

Part 12 Calibration

The calibration procedure of the uTracer6 is an iterative process. A measurement is started during which a parameter is measured. If the measured value does not correspond to the expected value, the corresponding slider in the calibration form is adjusted after which the measurement is started again. After a few iterations usually the measurement corresponds to the desired value.

The uTracer6 is a relatively simple device. Don't expect that it can generate an anode / screen voltage of a 1000V with millivolt accuracy!

The calibration of the anode and screen voltages in this manual is done at 250 V. The reason is that the maximum voltage allowed for my DVM is 300 V. If your DVM can handle higher voltages, feel free to do the calibration at the maximum voltage your DVM can handle. It will improve the accuracy!

Note that during the calibration of the grid section, jumper J2 needs to be installed! Remove it again after the calibration procedure.

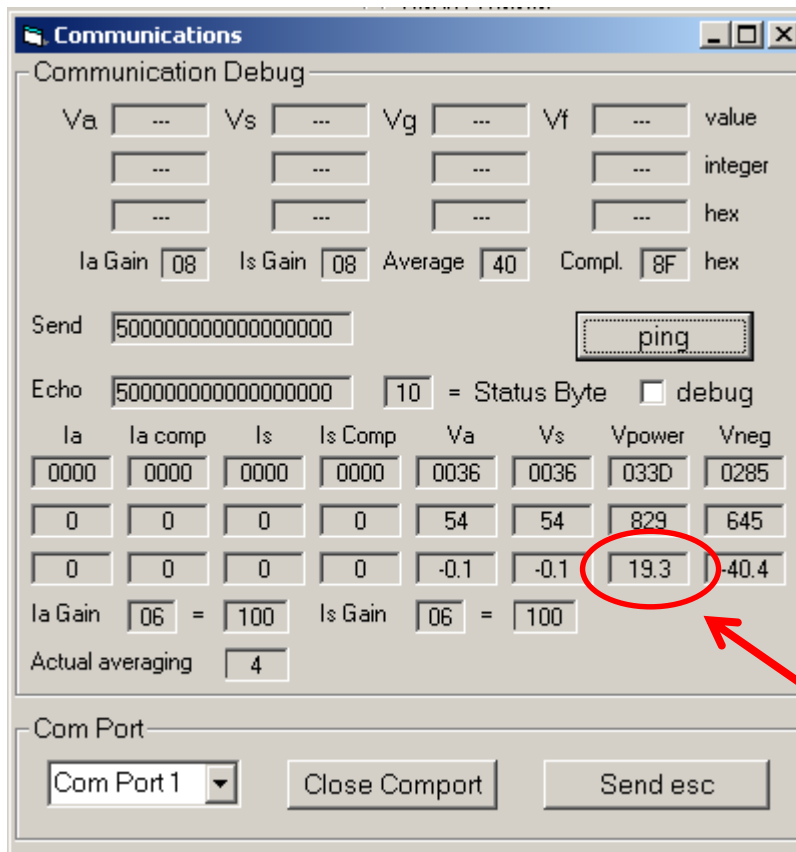
In between calibration steps, connections to the uTracer have to be changed. Please convince yourself that the high voltage LED is off, the power supply disconnected and the 100 uF caps discharged!

The calibration values are normally stored in the calibration file, however, please write down the final calibration values in this manual so that you can retrieve them in case your laptop crashes or other calamities.

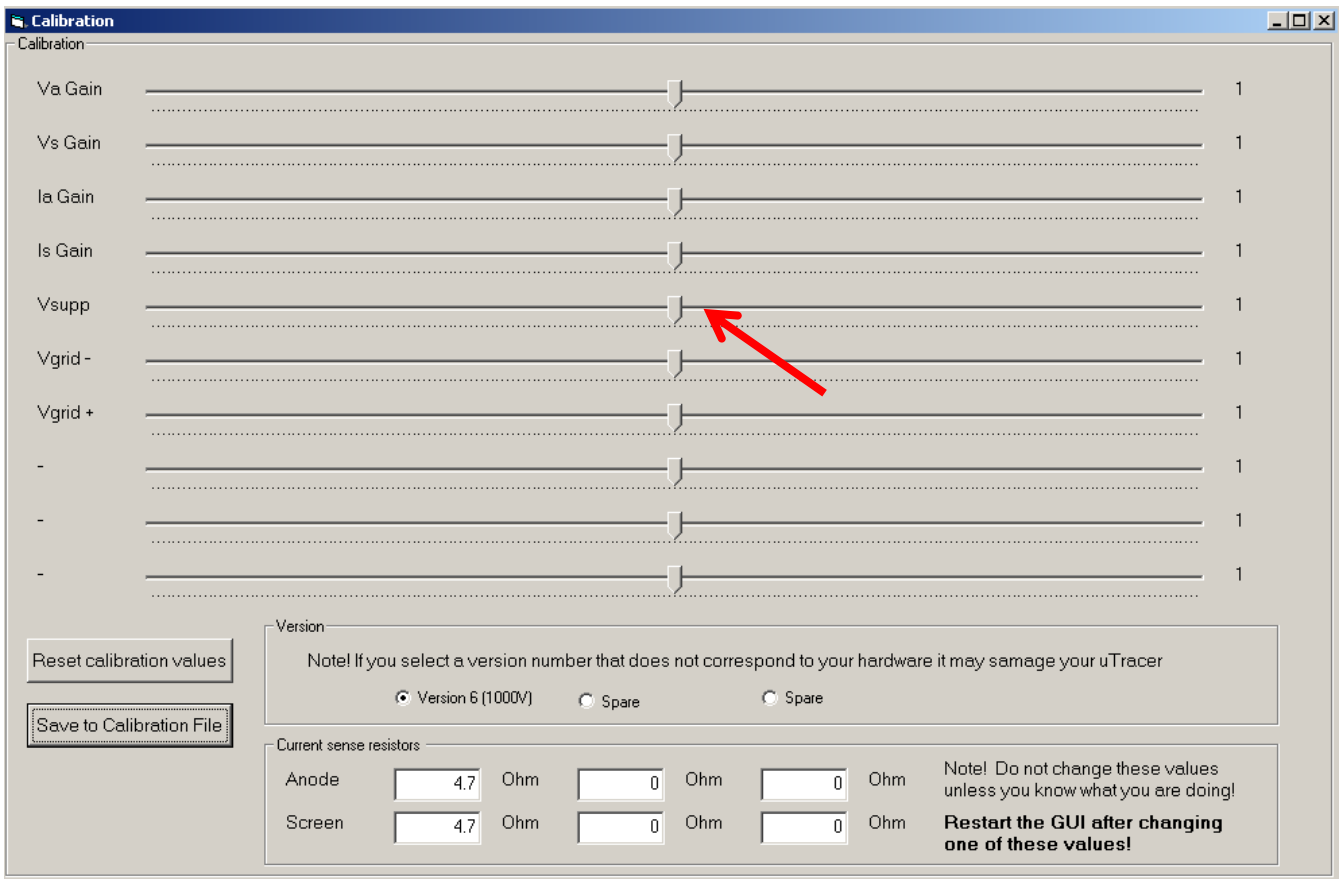


Part 12.1 Heater supply - calibration

- 1. Open the “Debug/Communications” window by pressing “Debug” in the “Miscellaneous” section of the GUI.
- 2. Press the “ping” command button.
- 3. The “Send string” and “Echo string” should now display:



- 4. No error message should appear !
- 5. The value in the circle is the supply voltage as measured by the PIC Microcontroller.
- 6. Measure the exact supply voltage using a multi-meter or DVM.
- 7. Open the Calibration form by pressing “Cal.” in the “Miscellaneous” section of the main GUI form.



- 8. Adjust the Vsuppl slide bar. (to the left to decrease the measured value, to the right to increase it)
- 9. Press “ping” on the “Debug/Communications” form.
- 10. Check if the supply voltage measured by the uTracer is equal to the exact supply voltage.
- 11. If not, repeat steps 8, 9, 10
- 12. When equal, press “Save to Calibration File” on the Calibration form to save the calibration value to the calibration file.

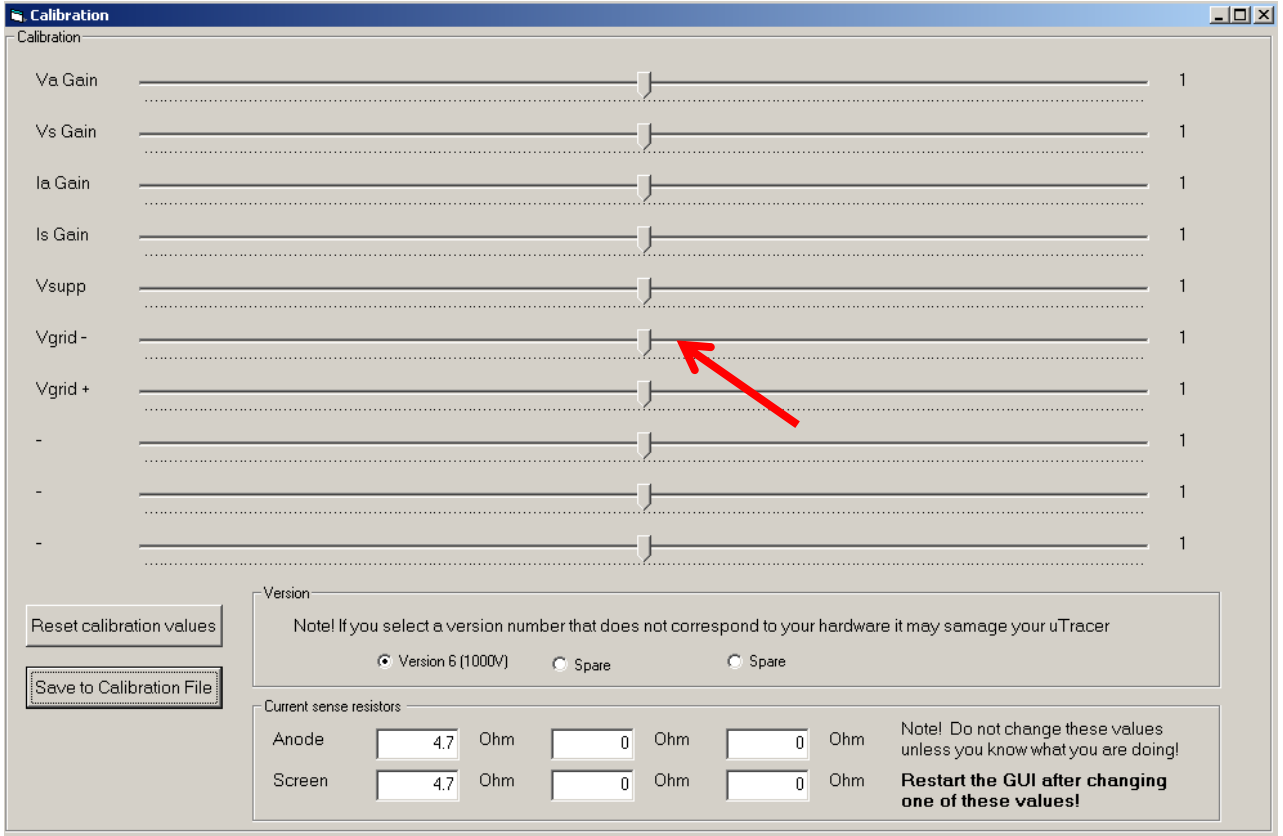
Part 12.2 Grid bias circuit – calibration

For the grid bias circuit calibration it is recommended to have the uTracer6 **switched on for at least half an hour** so that all components have reached their final operation temperature.

- 1. Make sure jumper J2 is removed.
- 2. For the calibration connect a DVM between grid and cathode terminals of the uTracer as shown in the figure in “Part 9. Grid Bias Circuit – testing”
- 3. At power up, the uTracer6 will reset the grid bias circuit to 0 V. Adjust potentiometer P40 until the grid bias output is 0 V.
- 4. In the main form of the GUI select the measurement type as indicated below and fill in -80 V as start value for the grid bias.

The image shows a screenshot of the 'u-Tracer (V3.12.6)' software interface, specifically the 'Measurement Set-Up' dialog box. The dialog box is titled 'u-Tracer (V3.12.6)' and has a 'Measurement Set-Up' section. The 'Select Measurement type:' dropdown menu is set to 'I (Va, Vg) with Vs, Vh constant'. Below this, there are fields for 'Start', 'Stop', and 'Nintervals'. The 'Start' field is set to 2, 'Stop' is set to 200, and 'Nintervals' is set to 30. There is a checkbox for 'log' which is unchecked. Below these fields is a 'Stepping Variable (e.g: 15, 20, 25 ...)' field, which is circled in red and contains the value '-80'. Below this is a 'Constants:' section with fields for 'Vs' (set to 200) and 'Vh' (set to 6.3), and a 'pins' button. Below that is a 'Range:' section with dropdown menus for 'Ia' (set to Automatic) and 'Is' (set to Automatic). Below that is an 'Average:' section with a dropdown menu set to Automatic, and a 'Compliance:' section with a dropdown menu set to 1029 mA and a checkbox for 'Vg low' which is unchecked. At the bottom, there is a 'Delay:' field set to 0, a 'Measure Curve' button, an 'Abort!' button, a 'Beep' checkbox which is unchecked, a blue progress bar, and a 'Quick Test' button.

- 5. Open the calibration form by pressing “Cal.” in the “Miscellaneous” section of the main GUI form.



- 6. Press “Heater On!” on the main form.
- 7. To skip the “slow heating” feature press again on “Heating ...”
- 8. Start a measurement by pressing “Measure Curve.”
- 9. Read the actual grid voltage on the DVM or multimeter.
- 10. Wait till the measurement is finished or press “Abort!”
- 11. Adjust the Vgrid slider on the “Calibration” form if the measured grid voltage is not equal to -80 V. (to the left if it was less negative than -80V, to the right if it was more positive than -80V)
- 12. Repeat steps 6 to 11 till the measured grid voltage is -80V.
- 13. When the measured grid voltage is equal to -80 V press the “Save to Calibration” button on the “calibration” form to save the calibration value to the calibration file.
- 14. **POSITION JUMPER J2!**

Part 12.3 Boost converters – calibration

In the procedure below first the anode boost converter will be calibrated, followed by the screen boost converter..

In this example the boost converters are calibrated at 250V because my DVM can only handle voltages up to 300V. If your DVM can handle higher voltages it is better to calibrate at a higher voltage.

- 1. Connect a DVM between the cathode of D60 and ground like shown in section 10 “Boost converters – testing.”
- 2. Configure the measurement setup as shown below, note the 30 second delay!

Measurement Set-Up

Select Measurement type:
I (V_g, V_a=V_s) with V_h constant

Start Stop Nintervals
V_g -10 0 1 log

Stepping Variable (e.g: 15, 20, 25 ...)
V_a=V_s 250

Constants:
V_h 6.3 pins

Range:
I_a Automatic I_s Automatic

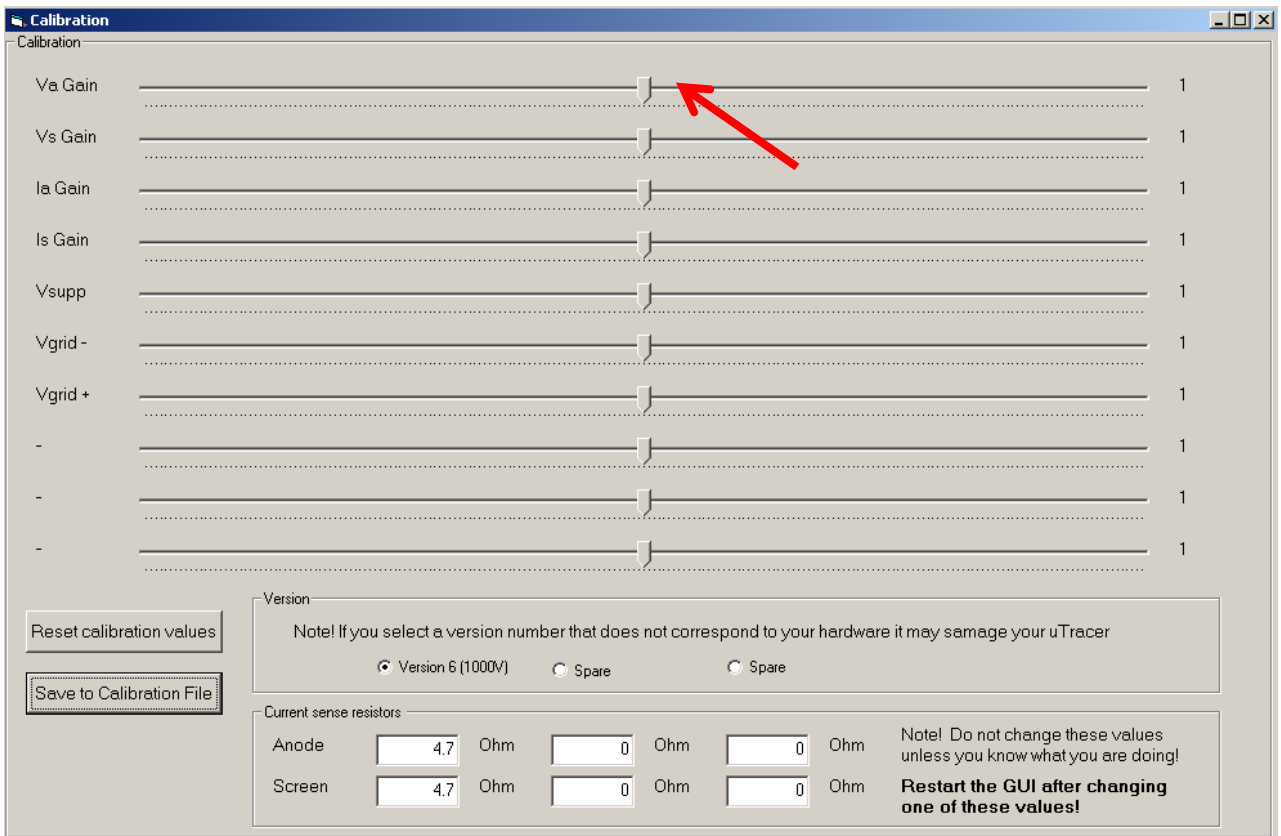
Average: Compliance:
Automatic 1029 mA V_g low

Delay: 30 Measure Curve Abort!

Beep Quick Test

When a tube would be connected to the uTracer this unusual configuration would measure the tube at a grid voltage of -10V and 0V at anode and screen voltages of 250V. Before the actual measurement the uTracer is instructed to wait 30 seconds. We use this configuration here to be able to read out the voltages during the 30 second stabilization phase.

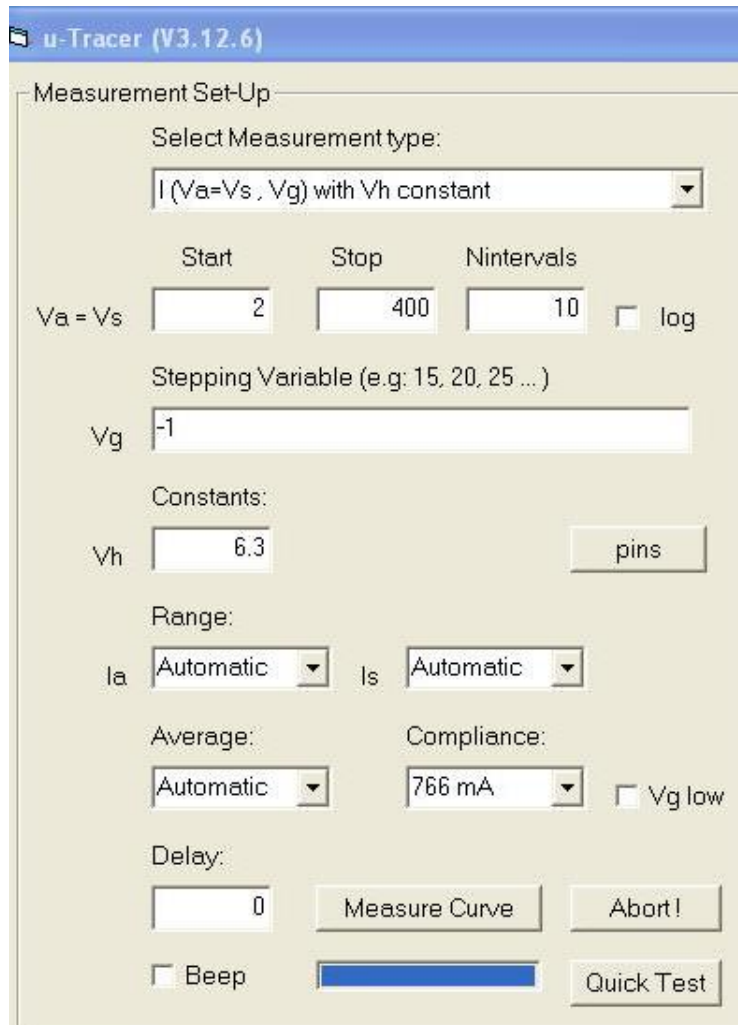
- 3. Measure the supply voltage = V_{supl} .
- 4. Open the calibration form by pressing “Cal.” in the “Miscellaneous” section of the main GUI form.
- 5. Press “Heater on!”
- 6. The text on the button now changes to “Heating” Press the button again to skip the slow heating procedure and finally press the button again to start a measurement.
- 7. Read the voltage on your DVM or multimeter.
- 8. Interrupt the measurement by pressing “Abort!”
- 9. If the measured voltage does not equal $250 + V_{supl}$, then adjust the V_aGain slider in the calibration form. (to the left of the voltage was too low, to the right if it was too high).



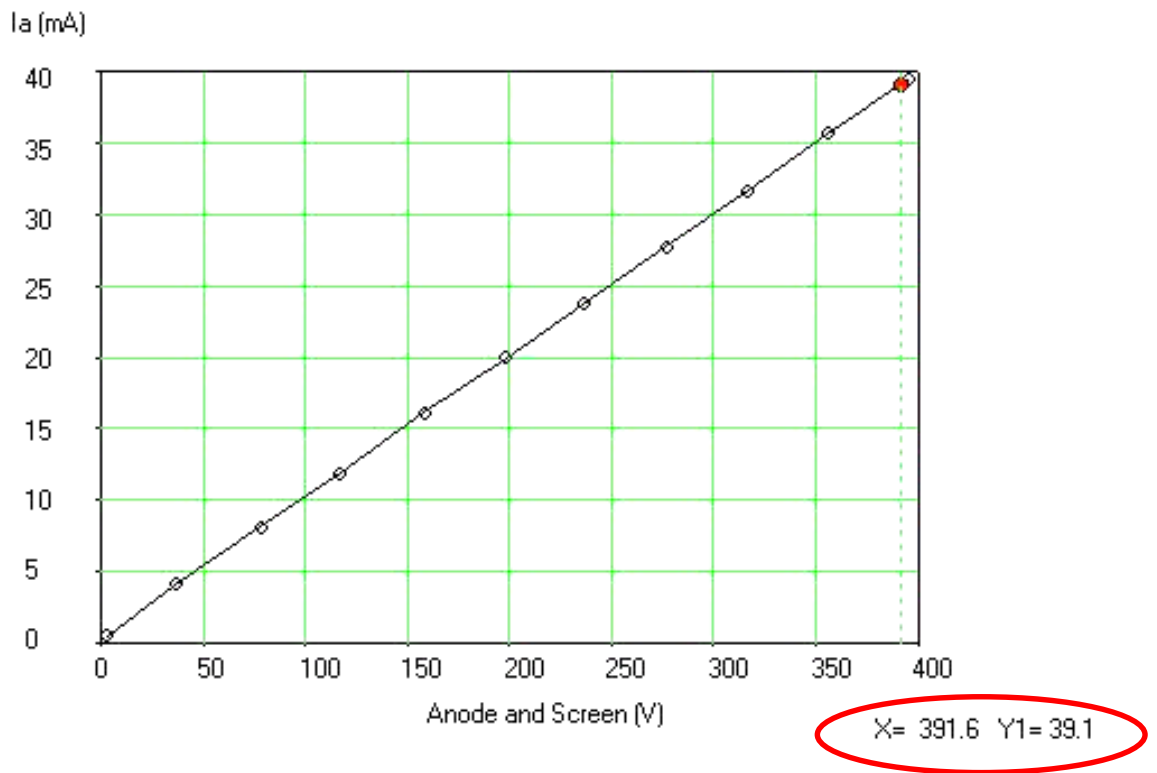
- 10. Repeat steps 6 to 9 until the measured voltage is $250V + V_{supl}$.
- 11. Connect the DVM or multimeter to the cathode of D80 and repeat the whole procedure (steps 6 to 10) but now adjust V_s Gain.

Part 12.4 Current amplifiers – calibration

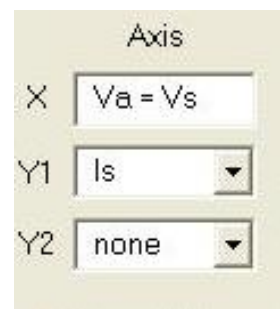
- 1. Connect a 10k ohm resistor between the anode and cathode terminals. and connect a second 10k resistor between the screen and cathode terminals. (see picture in Section 11 “HV Switches – testing.”)
- 2. Select the measurement shown below, and set the stop voltage to 400 V.



- 3. Open the calibration form by clicking on “Cal.” in the miscellaneous section of the main GUI form.
- 4. Press “Heater on!”
- 5. The text on the button now changes to “Heating.” Press the button again to skip the slow heating procedure, and finally press the button again to start a measurement.



- 6. Since a resistor is connected to the anode output, the measurement will produce a straight line (above).
- 7. Click with the mouse on the curve somewhere around 400 V. At the bottom of the curve two numbers appear. X is the voltage applied to the 10 k resistor, Y is the current flowing through the resistor. Note that the current should be $Y = X / 10k$. So when $X = 391.6$, ideally $Y = 391.6 / 10.000 = 39.1$ mA
- 8. If Y is too low according to the formula above, shift slider “Ia Gain” to the right. If the current is too high, shift it to the left.
- 9. When $Y = X / 10k$ press “Save to Calibration File”
- 10. Now plot the screen current on the Y-axis →
- 11. Repeat steps 4 to 10, but now adjust slider “Is Gain”
- 12. press “Save to Calibration File”

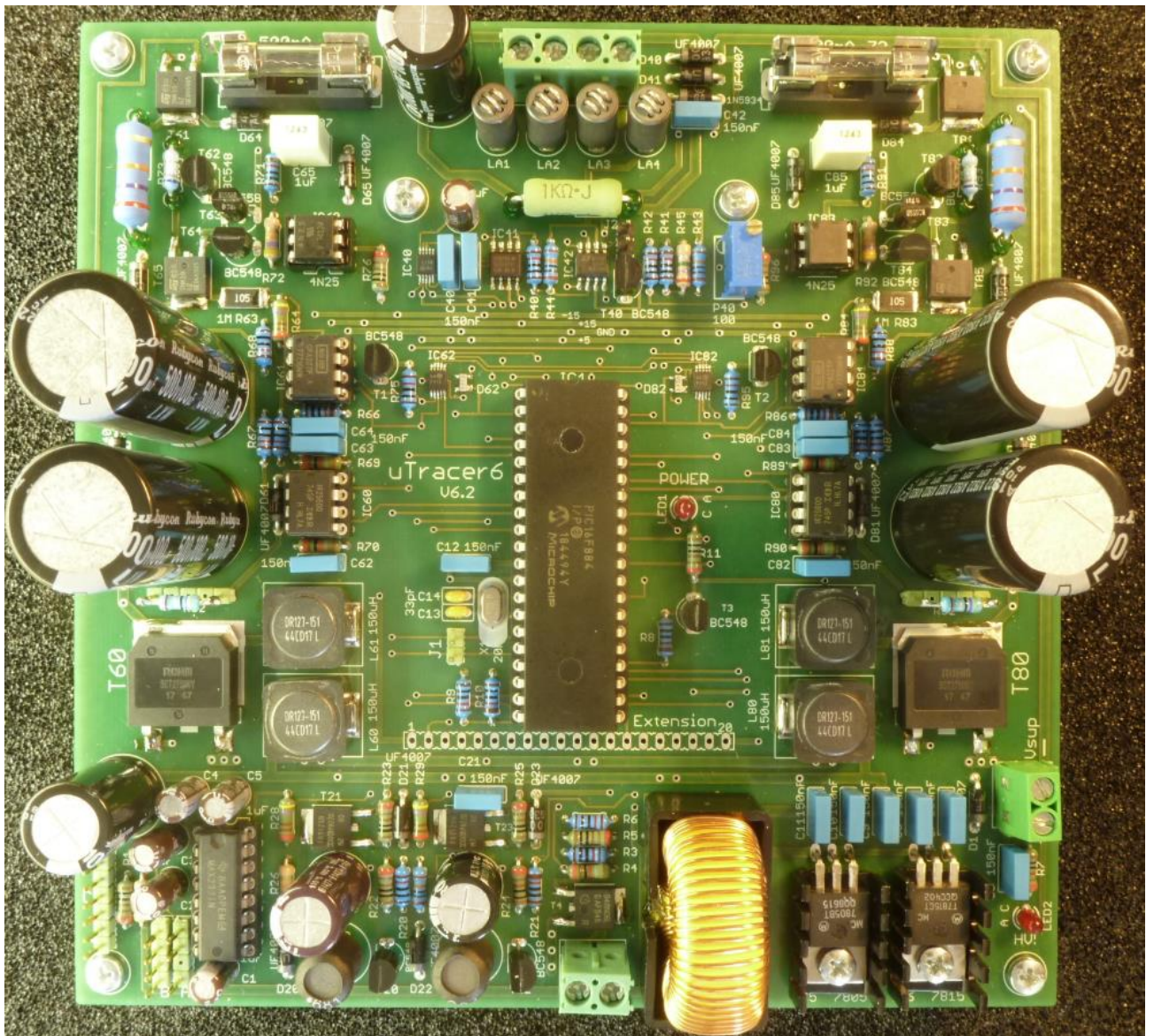


This completes the uTracer6 calibration!

Part 12.4 Backup calibration values

Write down the calibration values you have found in the table below:

Va Gain	
Vs Gain	
Ia Gain	
Is Gain	
Vsupl	
Vgrid -	
Vgrid +	



Configuration

The standard uTracer6 is configured for a hardware current limit slightly in excess of 1 A and a software current limit of approximately 750 mA.

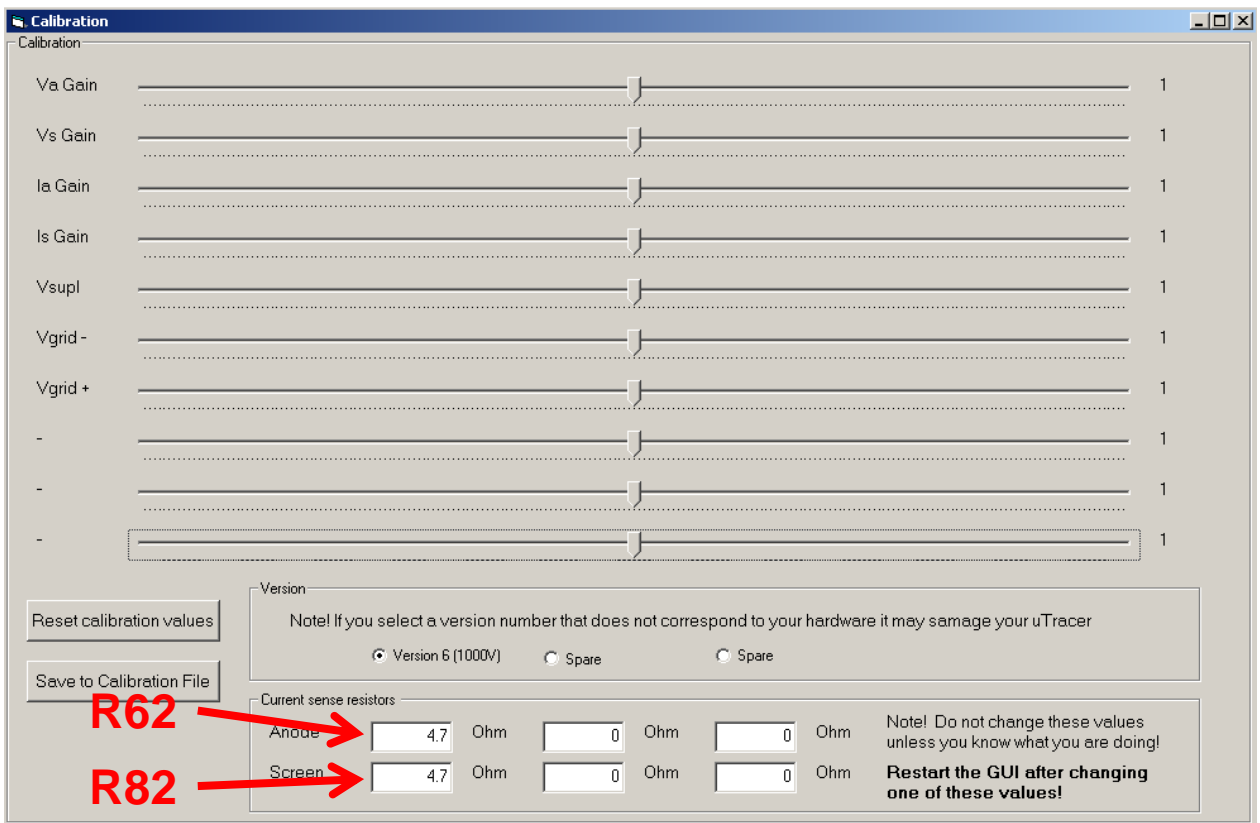
By changing the current sense resistors R62 and R82 you can either increase the current range at the expense of slightly higher noise level at low currents, or reduce the current range at the benefit of a better low current noise performance.

For more information please read:

www.dos4ever.com/uTracerlog6/tubetester6.html#current

The software current limit can easily be increased to 1 A by placing a 13.7 ohm resistor (provided with the kit) in parallel to the 4.7 ohm R62 and R82 resistors. This will lower the current sense resistance value to 3.5 ohm.

Any change in the current sense resistors has to be entered in the calibration form:

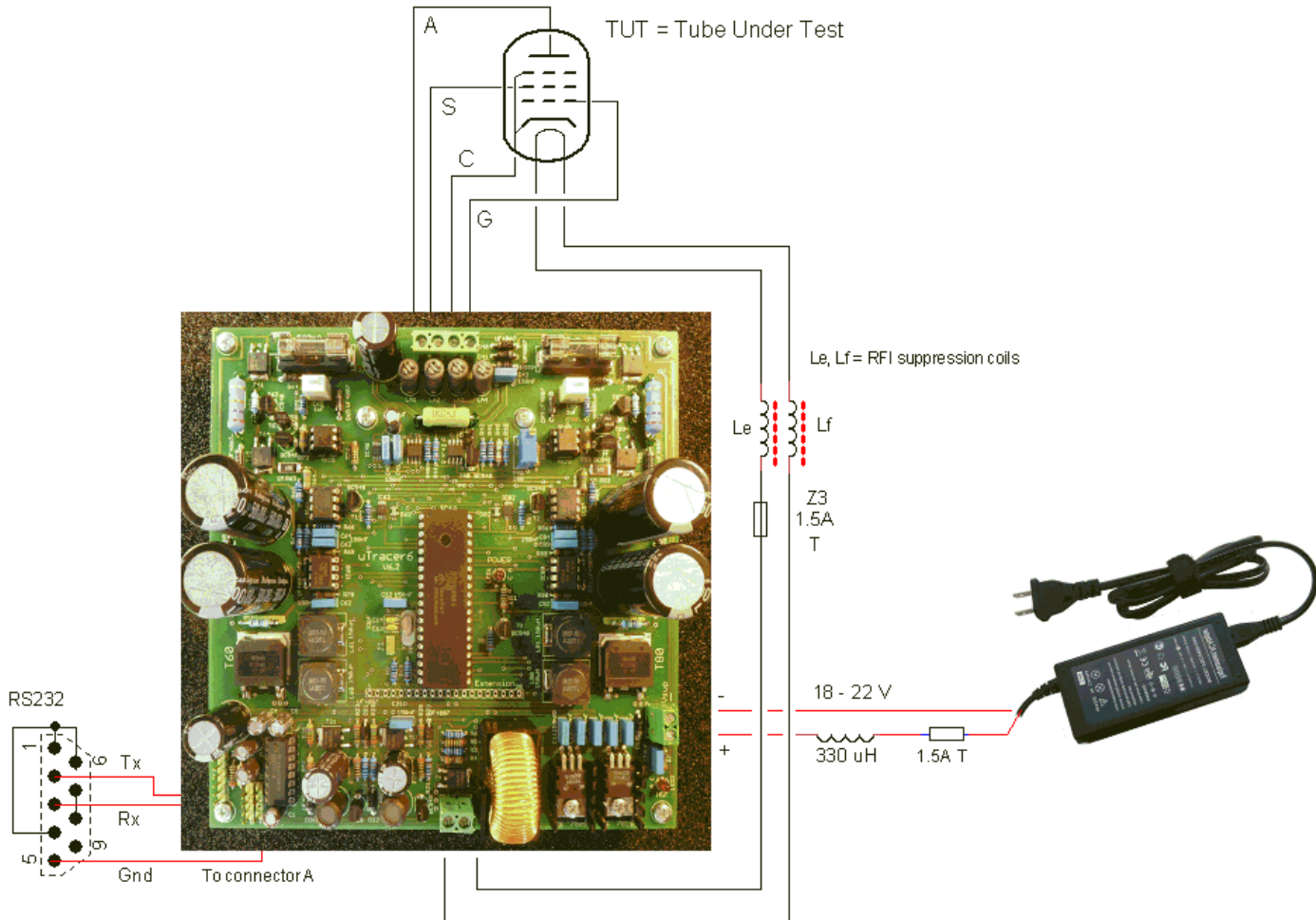


After making a change in the anode and screen current sense resistor values, press “save in Calibration File,” close the form and the GUI and then start the GUI again to make the new values active!

Wiring of the uTracer

The figure below shows how the uTracer is connected to the tube and the power supply.

The uTracer was designed to operate from an old laptop power supply with an output voltage in the range of 18 – 22 V. These power supplies combine a high output power with a small volume. It is strongly recommended to include a 330 μ H inductor and a 1.5 A fuse in series with the + lead of the power supply.



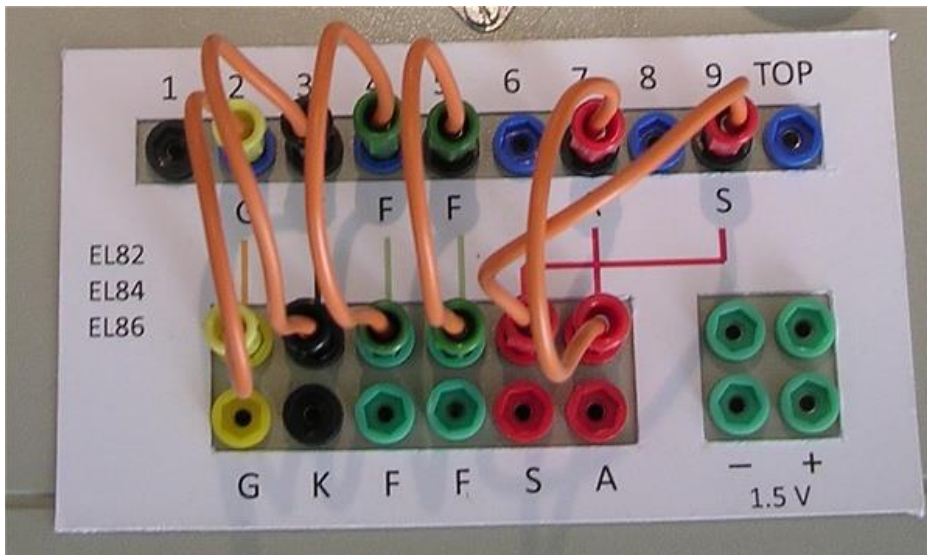
Additionally it is recommended to include a 1.5 A fuse and two RFI suppression beads in series with the heater connections. The connection of the other electrodes to the tubes will be discussed in the next section.

Tips for connecting the tube sockets

Probably everybody will have his (or hers!) on ideas on how to connect the uTracer to an array of tube sockets. In the many internet forums I read, many people opt for a relays matrix in combination with a database of tube sockets. It all sound very complicated and unnecessary to me.

More down to earth is a set of rotary switches, each switch connecting one of the terminals of the uTracer to a tube socket pin.

Personally I use a set of miniature banana plugs to make the required connections.



The top row of plugs is connected to the tube sockets. The bottom row is connected to the uTracer terminals. The bottom row contains a double set of plugs so that it is possible to connect two tube pins to one terminal e.g. to connect a pentode as triode (by connecting both the screen grid as well as the anode to the anode terminal). A separate set of plugs is connected to a 1.5 V battery for battery operated tubes. Color coding is used to reduce the risk of mistakes: red for screen and anode (high voltage), green for the heater, black for the cathode (ground), and yellow for the control grid. I made overlay cards with the wiring scheme for the most common tubes.

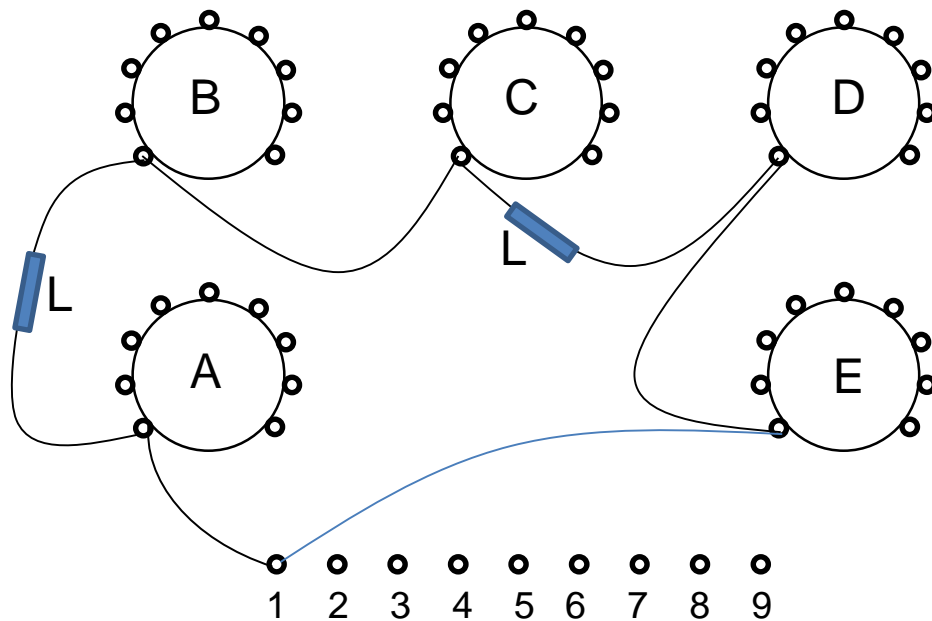
For conventional tube testers, such a plug-and-wire solution would be rather dangerous because high voltages can be present on the terminals. In this case the danger is very limited because the high voltages are only present for 1 millisecond, and even then the amount of charge stored in the capacitors is limited, or at least less dangerous than the power an 80 W / 300 V power supply can deliver.

One of the biggest threats to any tube tester are unwanted oscillations which can occur because the tubes are tested under realistic bias conditions while they are connected by long wires which contain many parasitic resonance circuits.

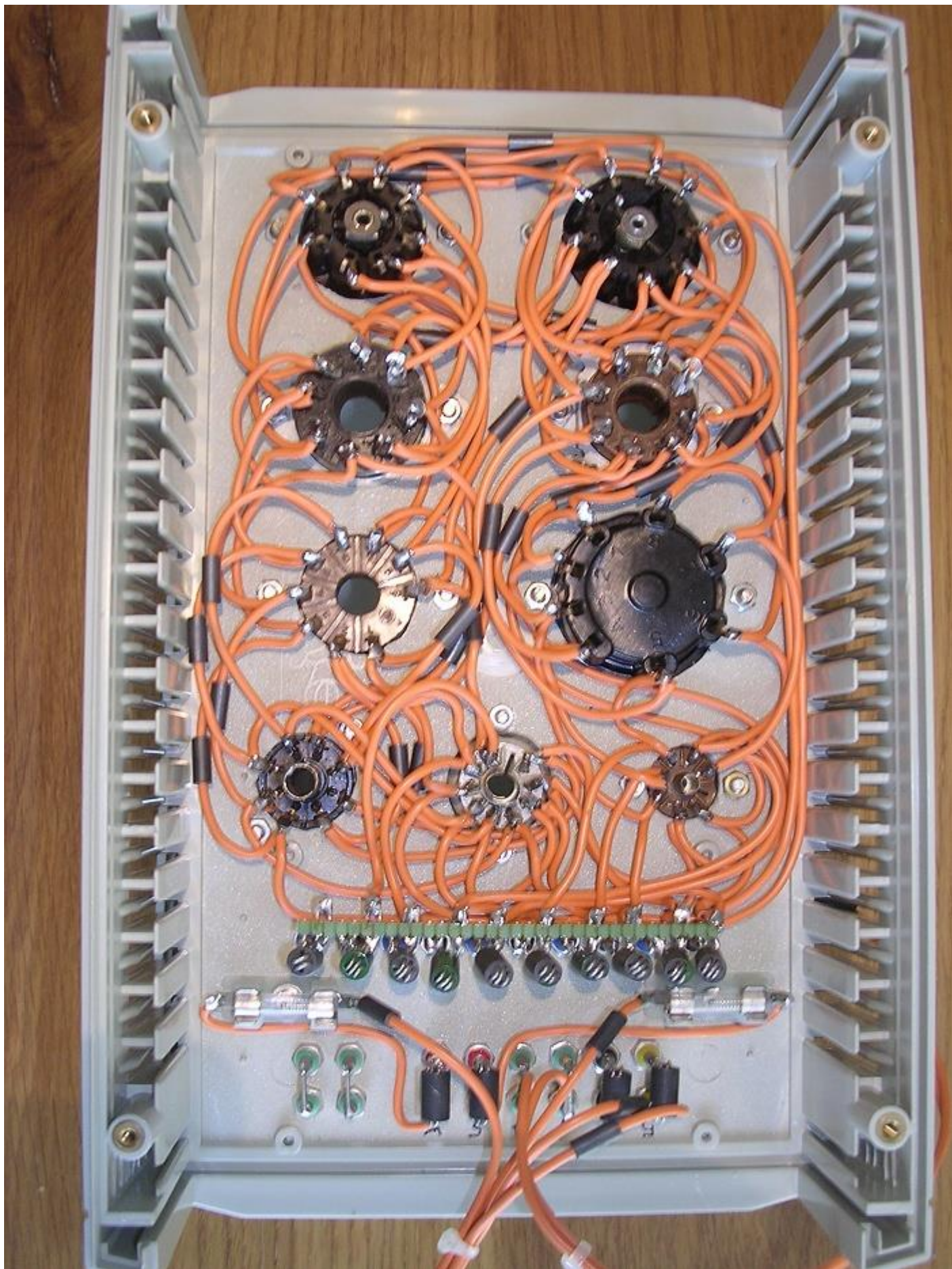
With conventional tubetesters, it can very well happen that a tube is destroyed because a destructive oscillation occurs! A big advantage of the uTracer is that the maximum energy stored in the reservoir capacitors is very small, too small to destroy a tube. The oscillations can however disturb the measurement so it is important that they are prevented.

One of the most effective ways to achieve this is to include RF suppression coils in the tube connections. At DC the resistance of these coils is almost zero, while the resistive losses quickly increase for increasing frequencies.

Furthermore, I strongly recommend using the wiring scheme that AVO uses (and patented) in their tube-testers. The schematic diagram below shows the basic idea for one terminal.

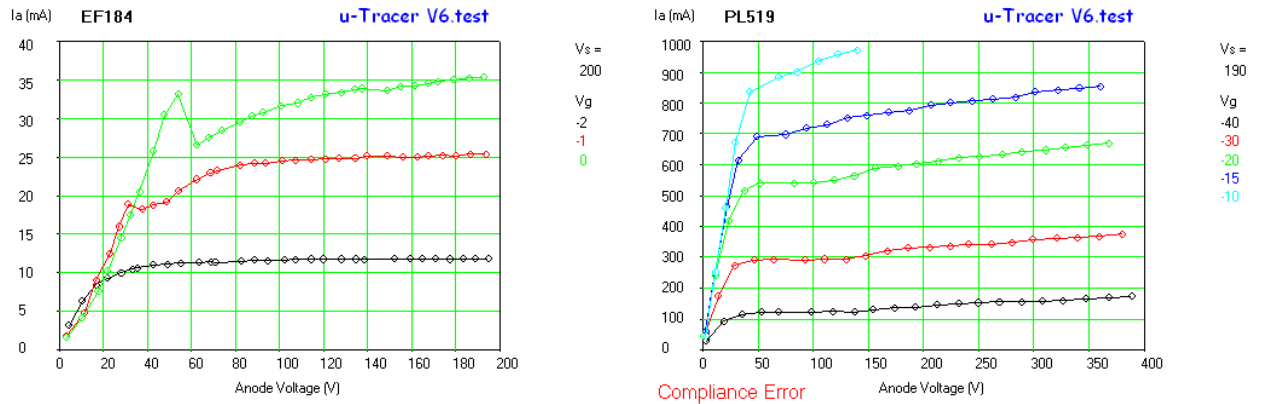


The wire connecting say pin one of each tube socket runs in a loop which also connects to the banana plug or rotary switch. At certain intervals an RF suppression bead (e.g. Wuerth no: 74270015) is shifted over the wires. The physical lengths of all the loops has to be approximately the same. The figure on the next page gives an impression of the wiring scheme of my version of the uTracer. Note that in this version the high voltage fuses were not yet integrated on the PCB,



Suppression of oscillations

Despite all precautions (careful wiring, inclusion of RFI suppression beads etc.) it is still possible that the tube will oscillate, especially for high slope (high transconductance) tubes. When a tube oscillates during tracing the curves appear ragged or noisy like these EF184 and PL519 examples:



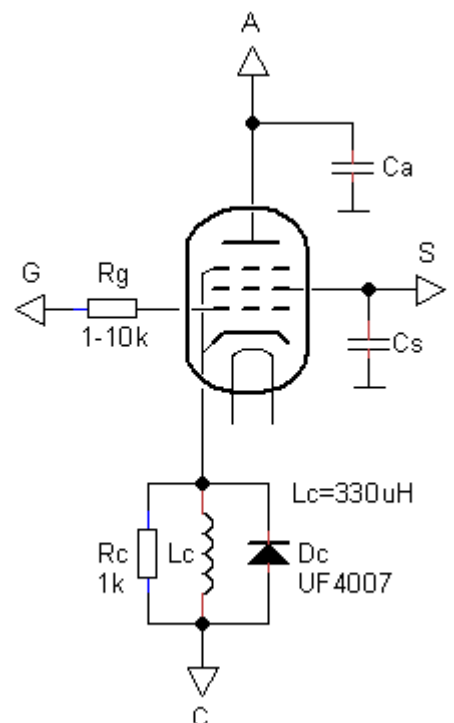
Oscillations are unpredictable and sometimes difficult to avoid. Since the cause for the oscillations may vary depending on the construction and wiring of the uTracer and the sockets, it is impossible to give a definite remedy that will work on all cases. Rather there are a number of tricks that might work individually or in combination with each other. All of them are aimed at reducing the gain of the tube at RF frequencies while leaving the DC characteristics unchanged.

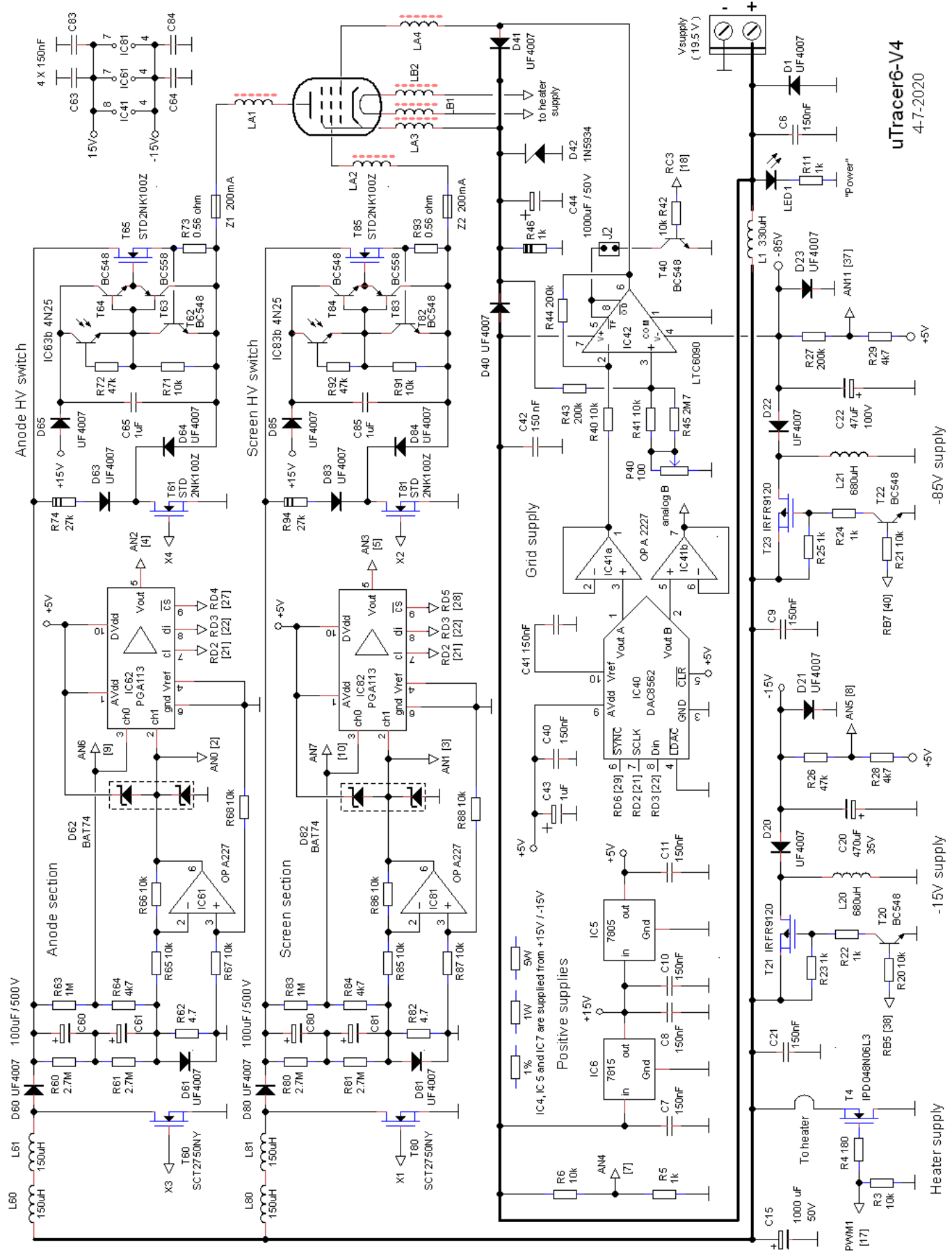
One of the most common ways to stop oscillations is the insertion of a "grid-stopper" resistors. Try values between 100 ohm and 10k.

Some tubes immediately stop oscillating when a small load capacitance (range 10 pF to 300 pF- high voltage!) is connected to the anode (or screen).

Very effective is the insertion of an RLD combination in the cathode lead. For DC the inductors is a short, while for RF the resistor kills the gain. The diode is just a safety precaution. (components supplied with the kit)

Sometimes a combination of the above measures is required.

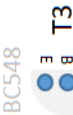
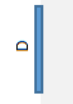
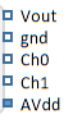
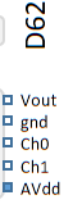
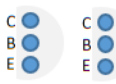
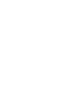
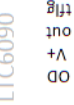
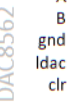




uTracer6-V4
4-7-2020

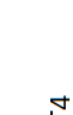
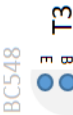
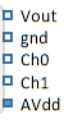
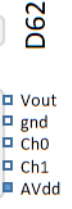
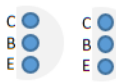
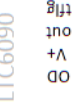
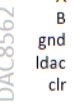
2NK100Z

2NK100Z



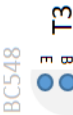
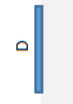
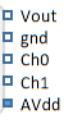
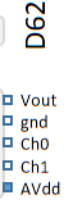
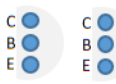
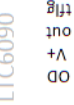
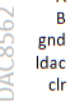
2NK100Z

2NK100Z



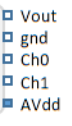
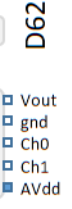
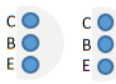
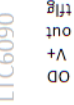
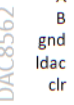
2NK100Z

2NK100Z



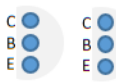
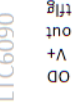
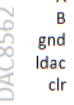
2NK100Z

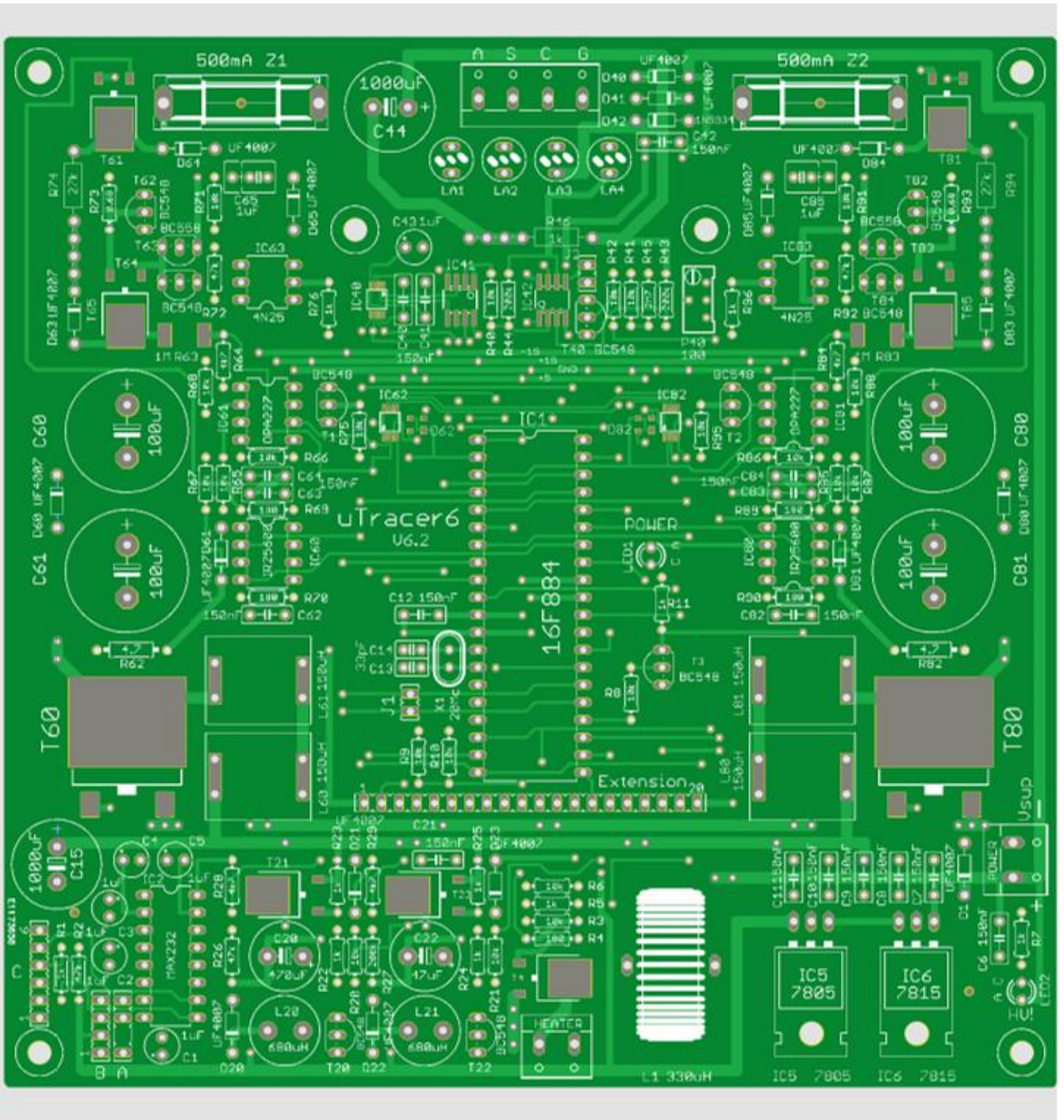
2NK100Z



2NK100Z

2NK100Z





Shin-ichi Haseno and David Johnston are acknowledged for carefully proof reading this manual