µTracer 3+

Construction Manual   V3.6

Important !

This kit is the result of many requests from the readers of my uTracer weblog pages to make this project available as a kit for those who do not have the means to build the circuit from scratch. The preparation of the kit has cost me a lot of time and effort, and I make little or no profit on it.

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 prevent disappointment, and indeed a good many of uTracers has now successfully been built! Nevertheless, small errors and or omissions may have occurred and I will be grateful if they are reported to me. A summary of the issues and problems collected so far can be found on the uTracer homepage www.dos4ever.com/uTracer3/uTracer3.html under "Bugs and Issues."

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

In case of any problems occurring during the construction of the kit, 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 that the voltages which 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 use of the uTracer circuit.

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

Before you begin

1. In order to build the μTracer 3 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 μTracer 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 when 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 uTracer 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 V, 500 mA max. In this way the damage done when an accidental short circuit occurs will be limited. This is by the way also more than enough to test an EL84!

6. Reversing the polarity of the power supply will certainly destroy the circuit! Reversing the cables can easily occur during the construction phase when you are building-testing-building-testing. I use a simple 90 degree clip to prevent accidentally connecting the power leads the wrong way (see next pages). 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.dekker@philips.com)

9. I will not and cannot be held responsible for any accidents or injuries which can occur as a result of improper use or defect of the uTracer circuit. The user is at all times responsible to take sufficient measures to prevent accidents or electrocution

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. 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. a small fretsaw

Tips and Tricks:

1. Bend the leads of the resistors in a sharp angle close to the body of the resistor
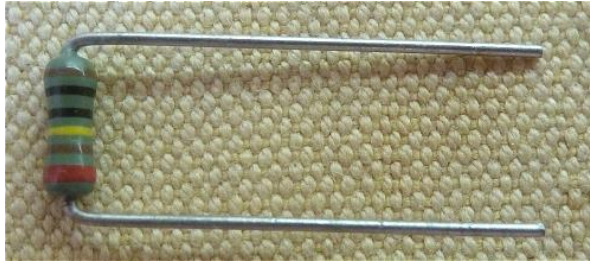
2. Trick to bend the leads of the reservoir capacitors into a nice 90 degree angle

3. Use a small box or the like to support the PCB on during component insertion and soldering

4. By using a 90 degree connector for the power supply connection, the accidental mix-up of power supply polarity can be prevented
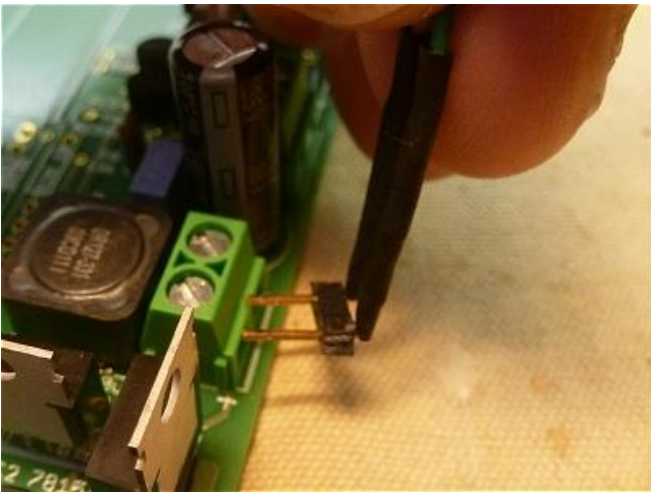

5. Since some of the components are located underneath the PIC microcontroller, the "separator spacers" in the center of the 40 pin socket need to be removed e.g. with a small fret saw.
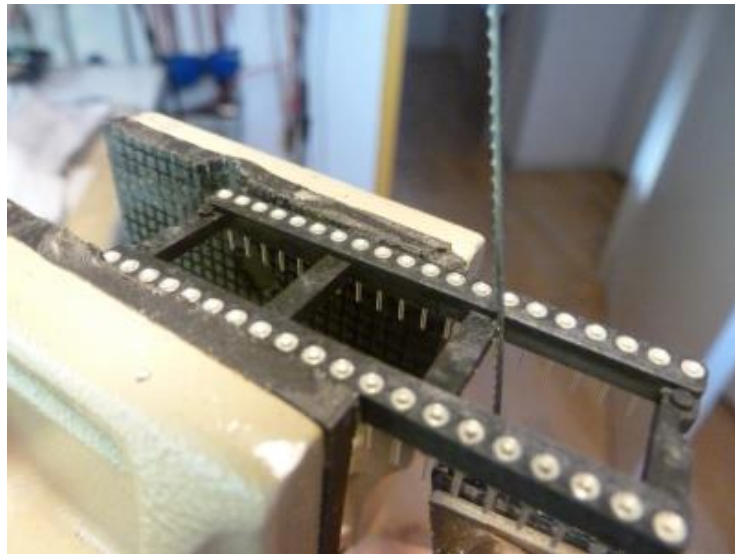
6. The inductors are relatively heavy components. When they are only soldered to the top copper layer, it is not unimaginable that they will (eventually) delaminate the copper layer. What I recommend to do is to first solder two pieces of "U" shaped wire in the holes in the contact pads, and then to solder the inductors to the wires. Additionally this will create a standof between the inductor (where high voltages can occur) and the tracks underneath the inductors
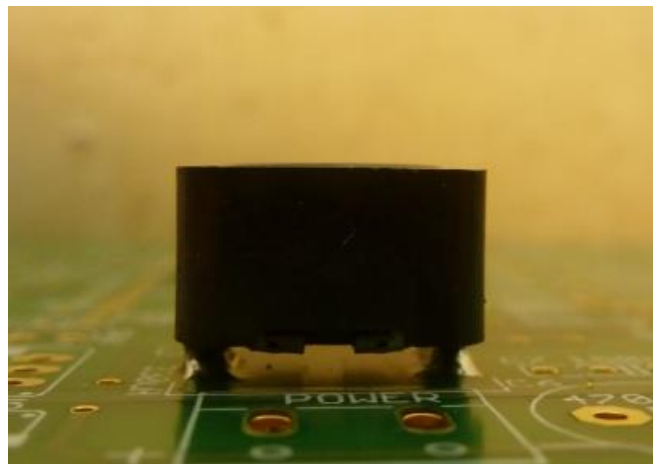
7. When soldering the components to the PCB, allow for some time to let the solder flow through the contact holes to the front side of the PCB. This will create an additional conductive path between front- and back-side contact pad.
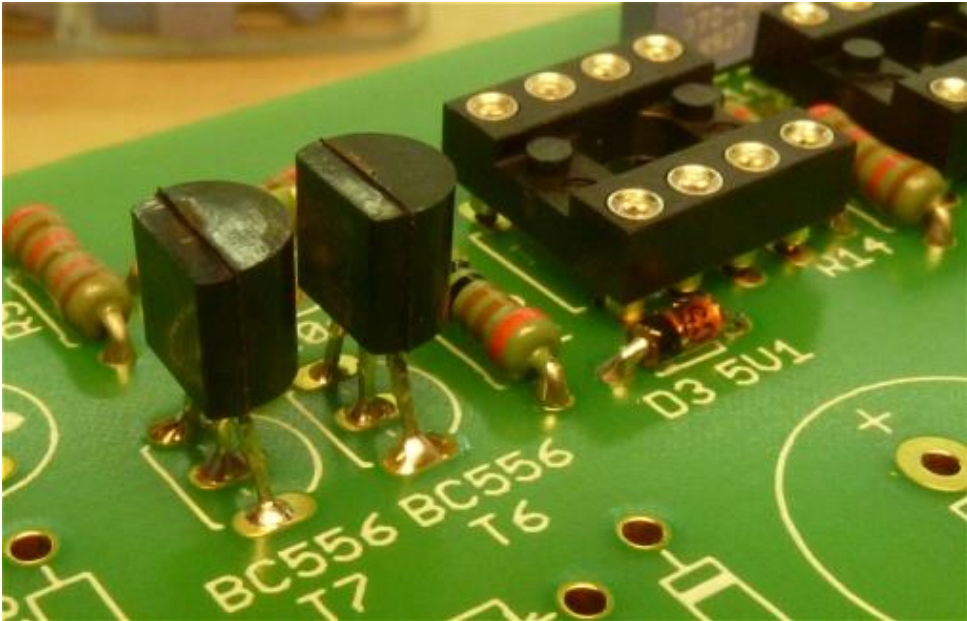

8. In order for the RFI suppression beads to fit on the PCB they need an additional turn. At the same time this will create some additional impedance.

9. Since earliest boyhood I use this crocodile clamp as a "mouth piece" to apply solder when I hold a PCB with my left hand (pressing the component to the PCB with the index finger) and hold the solder iron with my right hand. It must look rather stupid, but works great!

10. The PCB is designed for both standard size 5 mm pitch Xtals as well as miniature 2.5 mm pitch Xtals. In this kit the miniature Xtals are used. Be sure to insert the Xtal into the proper pads, and to use the insulating spacer (see picture below)
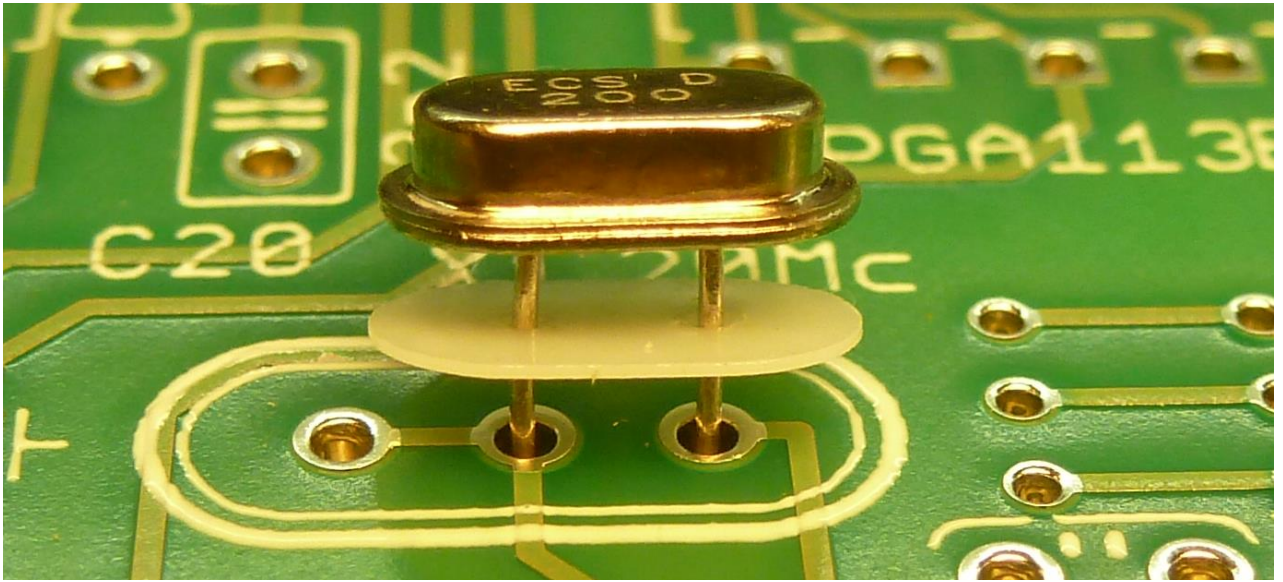
11. As usual the pins of the PIC processor are at an angle so that the PIC doesn't fit directly in the 40 pin socket. The best way to adjust the angle of the pins is to hold the processor at the sides and correct the angle of 20 pins at once by bending them against a flat surface e.g. the table

12. 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!

not properly inserted

properly inserted
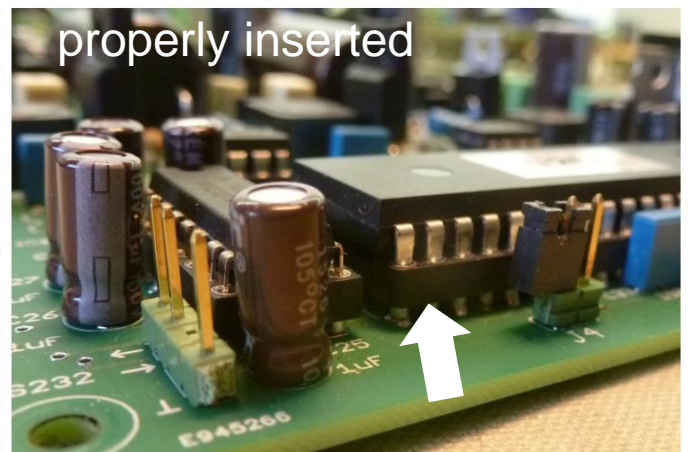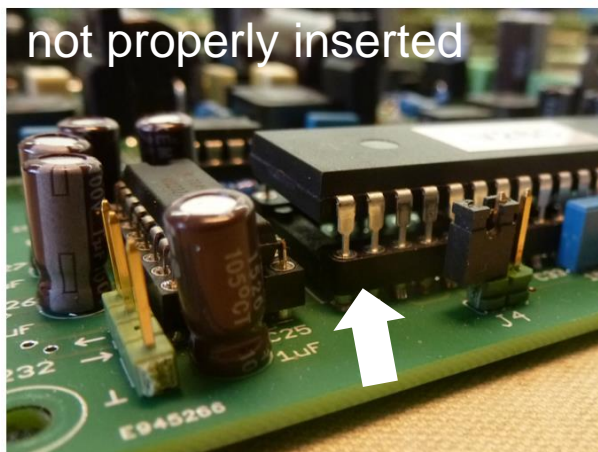
Step-by-step construction of the uTracer

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

The uTracer consists of the following parts:

1. the Graphical User Interface (GUI)
2. the RS232 cable
3. the voltage regulators
4. RS232 interface
5. microcontroller
6. heater supply
7. negative power supply
8. grid bias circuit
9. current amplifiers
10. high voltage boost converters and switches

The circuit is built in this order

Each part consists of a "construction" and a "testing / calibration" part. 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 than 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 PGA113s come in an impossible small MSOP-10 package which is almost impossible to handle without special tools. They are therefore already assembled on the PCB

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

During most of the tests the HV LED will be on, although no high voltages are present. This is normal and the result of the fact that the ADC input is not connected.

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 on the board.

In some cases component values used differ from the values printed on the PCB. This is due to the fact that for the uTracer3+ the same board is used as for the uTracer3. In those cases the warning sign (right) is used

For the same reason in a few cases the components need to be placed On the backside of the PCB or "piggy bag" on another component (part10).

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 which are perhaps difficult to understand from the text

Note that the uTracer3+ can only be used in combination with the GUI version 3p11 or higher !

High resolution photographs of the PCB can be downloaded from:
www.dos4ever.com/uTracer3/3+Front.jpg
www.dos4ever.com/uTracer3/3+Back.jpg

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

Component in circuit diagram	Replacement type(s)
HER108	UF1J or UF4007

Part 1. The GUI – installation

☐   1.   check if your PC has a COM port, if so skip steps 2 – 5

         ☐   2.   buy a USB-to-serial adapter, make sure it uses the FTDI chipset (see weblog)

         ☐   3.   install the USB-to-serial driver if needed

         ☐   4.   make sure the communication protocol is set to 9600 baud, 1 stop bit, no parity check

         ☐   5.   be sure you know the COM port assigned to the adapter

☐   6.   create a folder to download the zipped file into

☐   7.   download the GUI into the folder from:

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

Note that the uTracer3+ can only be used in combination with GUI 3p11 or higher!

☐   8. unzip the downloaded file into that folder

☐   9. double click "setup"

☐   10. follow the instructions of the setup program

☐   11. start the GUI from your program list

If the program is run for the first time it will generate a calibration file in the directory from which the program is run. This file will hold the calibration values.
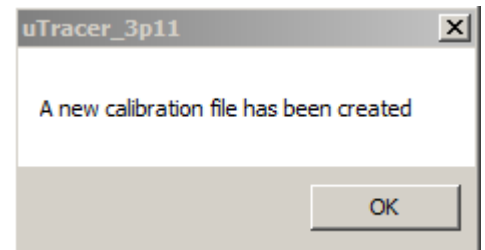
uTracer_3p11   ✕
A new calibration file has been created
OK

☐   12. Open the Calibration form by pressing "Cal." in the "Miscellaneous" section of the main GUI form

| Save to Calibration File	Version
	○ Version 3 (300V) ⦿ Version 3+ (400V)
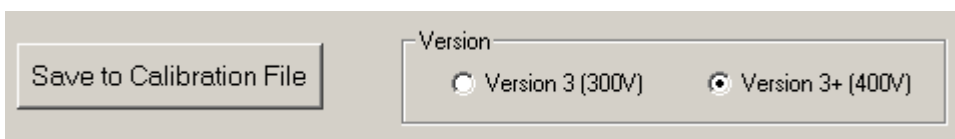
☐   13. Select Version 3+ (400V). The main form will now pop to the front because the GUI has been reset.

☐   14. Bring the Calibration form to the front and press "Save to Calibration File" to store the Version3+ configuration selection in the Cal file.

Part 1. 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:
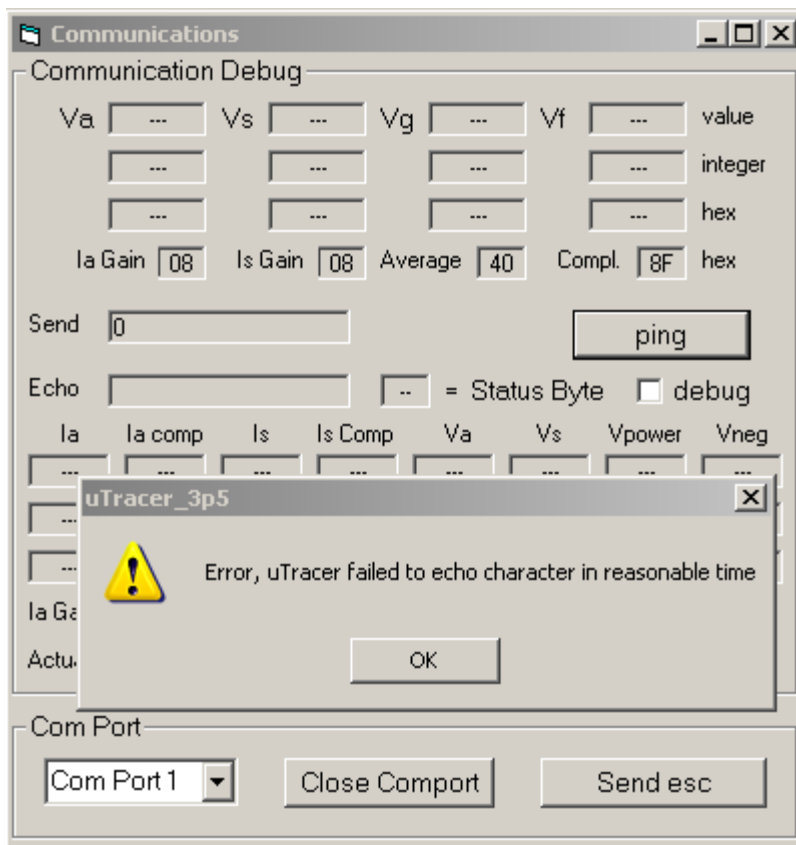
explanation:
The GUI tries to send a "0" this zero is not echoed. After a few seconds the GUI detects that no character is returned resulting in the timeout message.

☐   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 2. 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
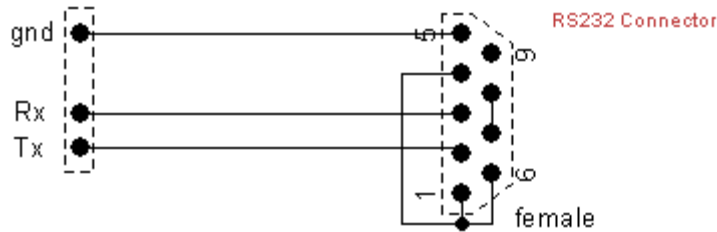

On the D-SUB connector pins 1,4,6 and pins 7,8 have been connected to "fool" the handshake signals. Strictly speaking this should not be necessary because the GUI does not check for these signals, but I am never sure what USB to serial adapters do
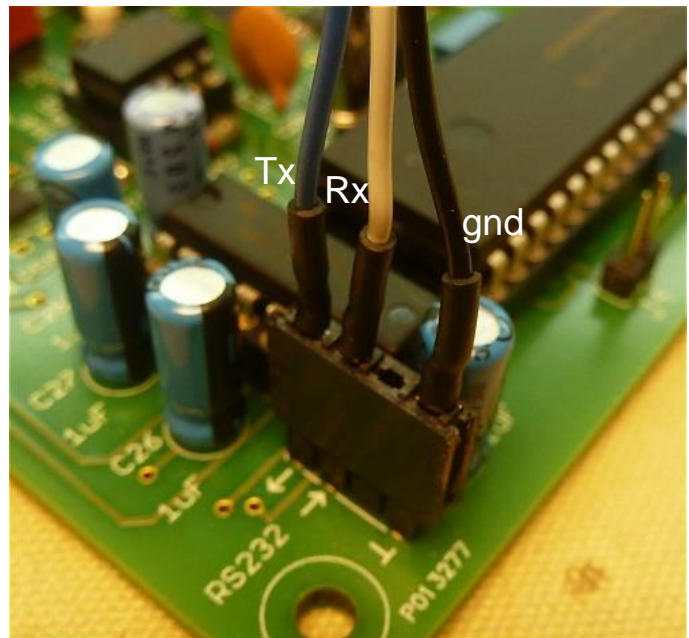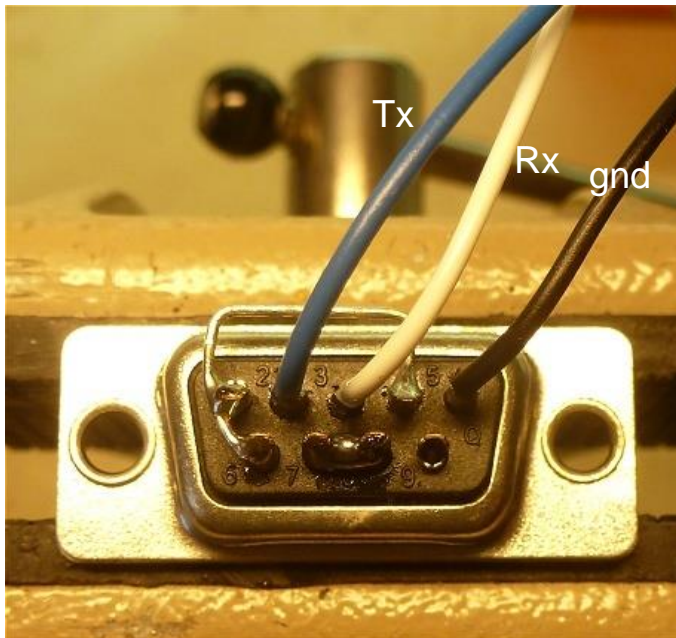
Part 2. The RS232 connector cable – testing

☐   1. make sure that the GUI is properly installed and tested

☐   2. connect the RS232 cable to the PC
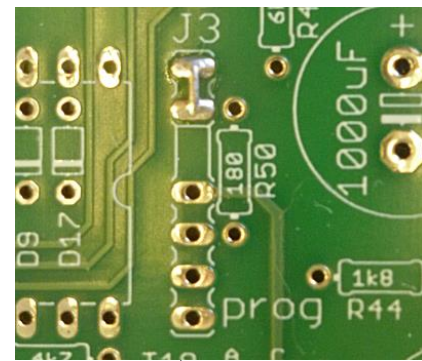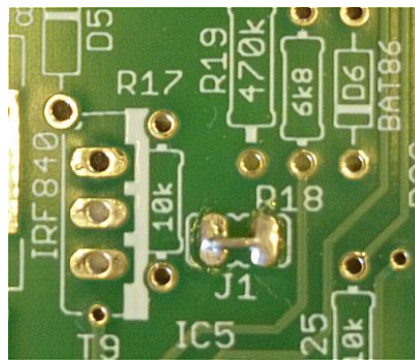
☐   3. connect the Tx and Rx pins   ➡️
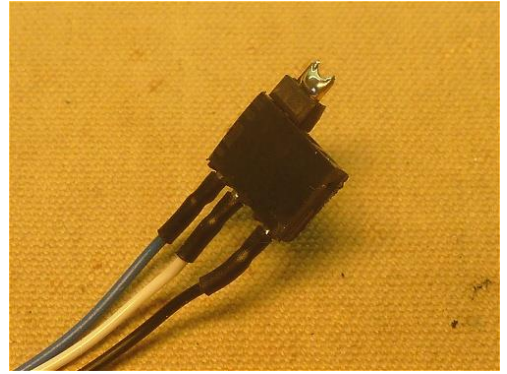
☐   4. start the GUI

☐   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:
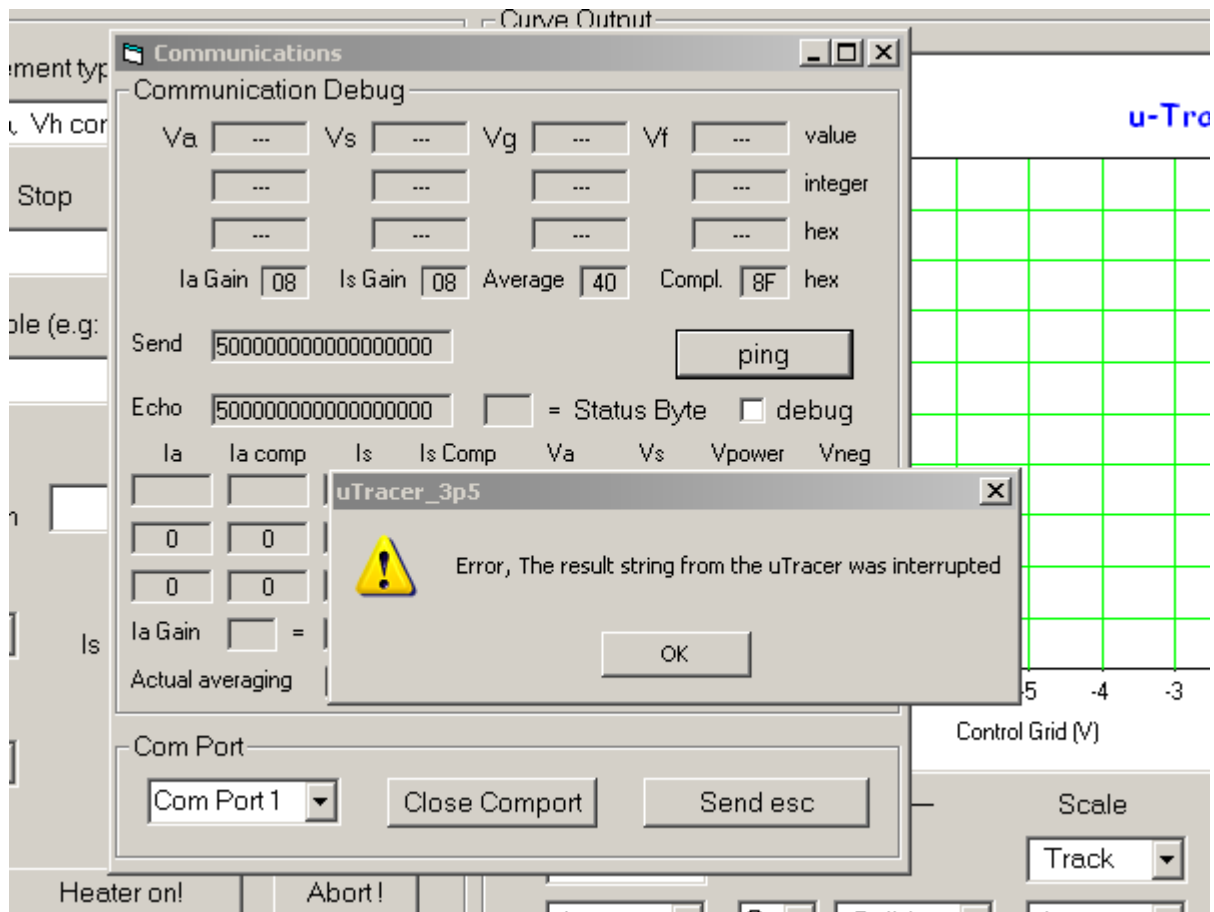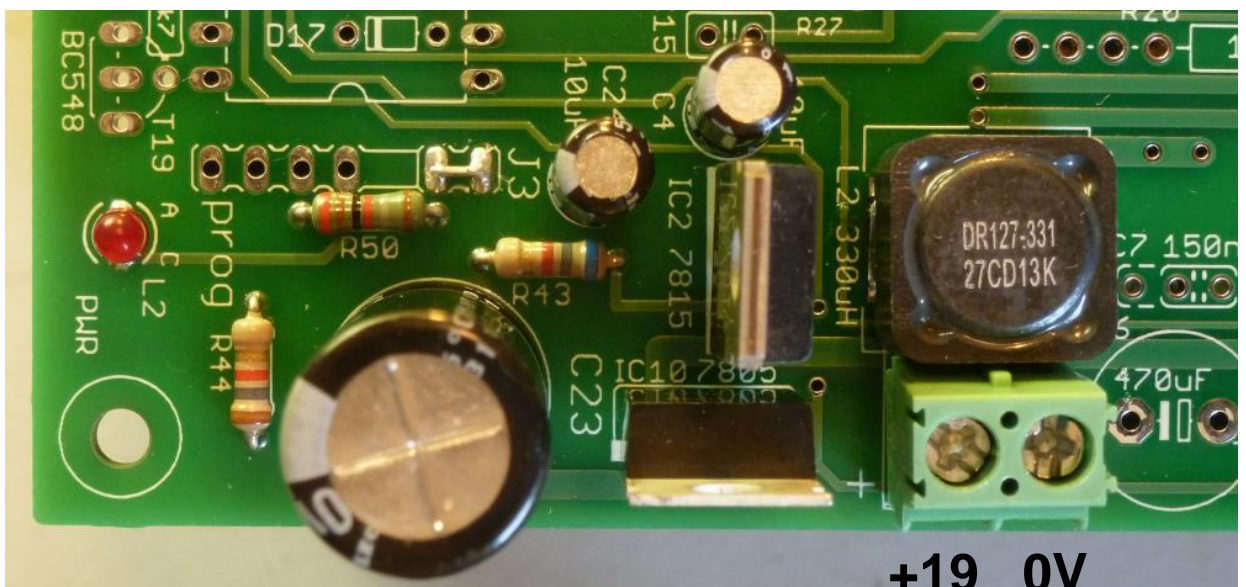

☐   8. after ca. 10sec the error message: "The result string from the

       uTracer was interrupted" should appear . This is the proper

       response

Part 3. The voltage regulators - construction

☐   1.   wire jumper J1

☐   2.   wire jumper J2

☐   3.   wire jumper J3

☐   4.   Power screw connector

☐   5.   L2, 330 uH, see "Tips and Tricks"

☐   6.   IC10, LM7805 , TO220, +5V voltage regulator

☐   7.   IC2, LM7815, TO220, +15V voltage regulator

☐   8.   R43, 6k8

☐   9.   R44, 1k8

☐ 10.   R50, 180 ohm

☐ 11.   LED 2, note proper orientation !

☐ 12.   C23, 1000 uF, 50V note polarity !

☐ 13.   C4, 10 uF, note polarity !

☐ 14.   C5, 150 nF

☐ 15.   C24, 10 uF, note polarity !
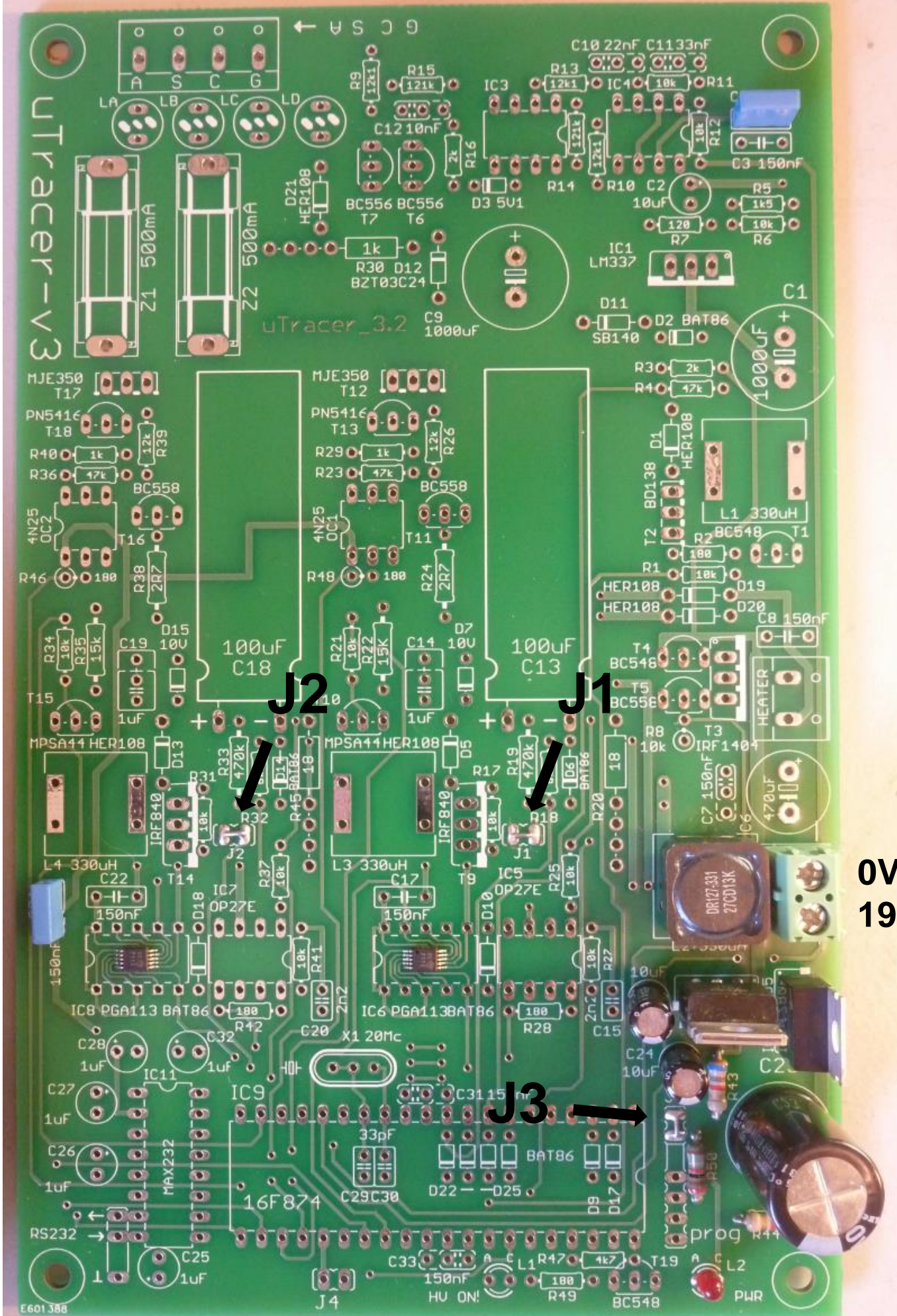
☐ 16.   C34, 150 nF

anode (long pin) cathode

# Part 3. The voltage regulators - testing

- ☐ 1. Connect the uTracer to a power supply ca. 19V, if possible limit the current to ca. 100 mA (see also "Tips and Tricks")
- ☐ 2. Switch the power supply on (supply current ≈ 30 mA @ 19 V)
- ☐ 4. Led L2 should be on, if not switch off and check circuit
- ☐ 5. check +15 V on pin 7 IC5 with respect to ground
- ☐ 6. check + 5 V on pin 16 IC11 with respect to ground

Part 4. RS232 Interface - construction

- ☐ 1. socket for IC11 , note orientation!
- ☐ 2. connector for J4 (male pin)
- ☐ 3. connector for connection to RS232 connector (three pin male)
- ☐ 4. C25, 1 uF, note polarity!
- ☐ 5. C26, 1 uF, note polarity!
- ☐ 6. C27, 1 uF, note polarity!
- ☐ 7. C28, 1 uF, note polarity!
- ☐ 8. C32, 1 uF, note polarity!
- ☐ 9. insert IC11, note orientation !
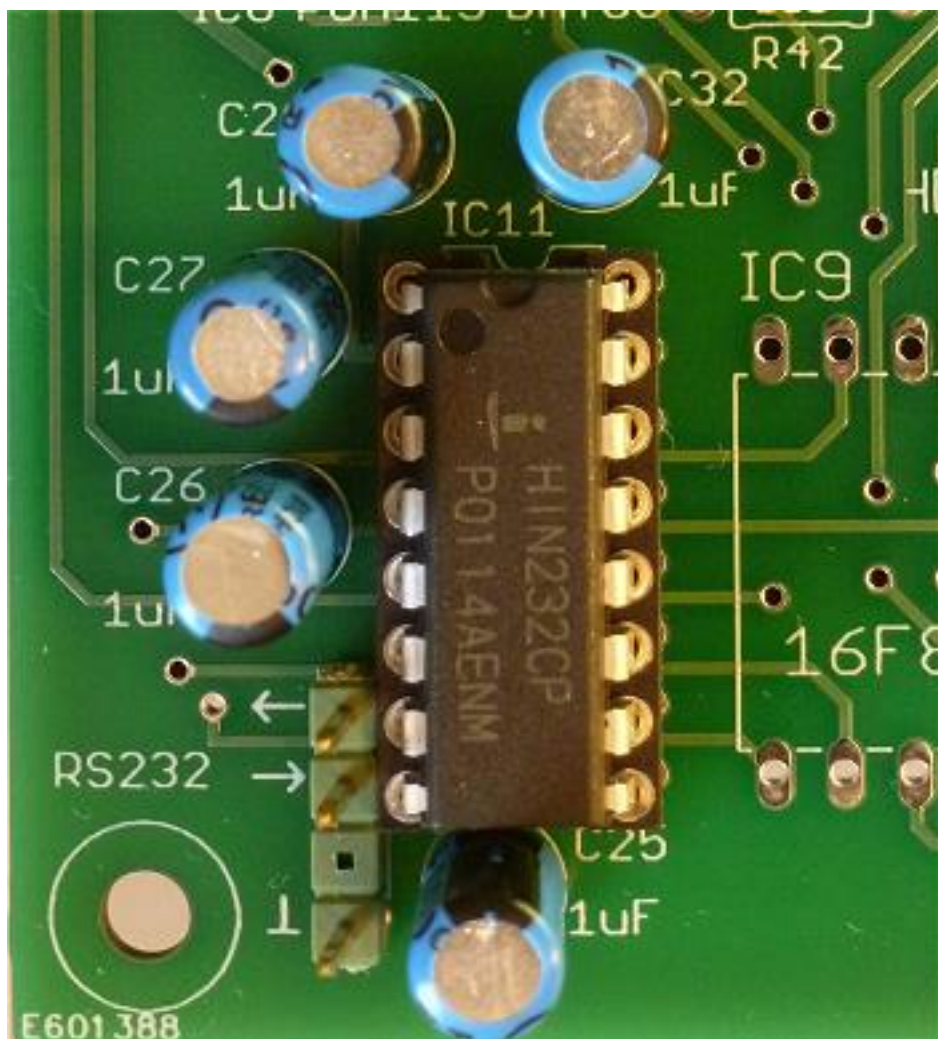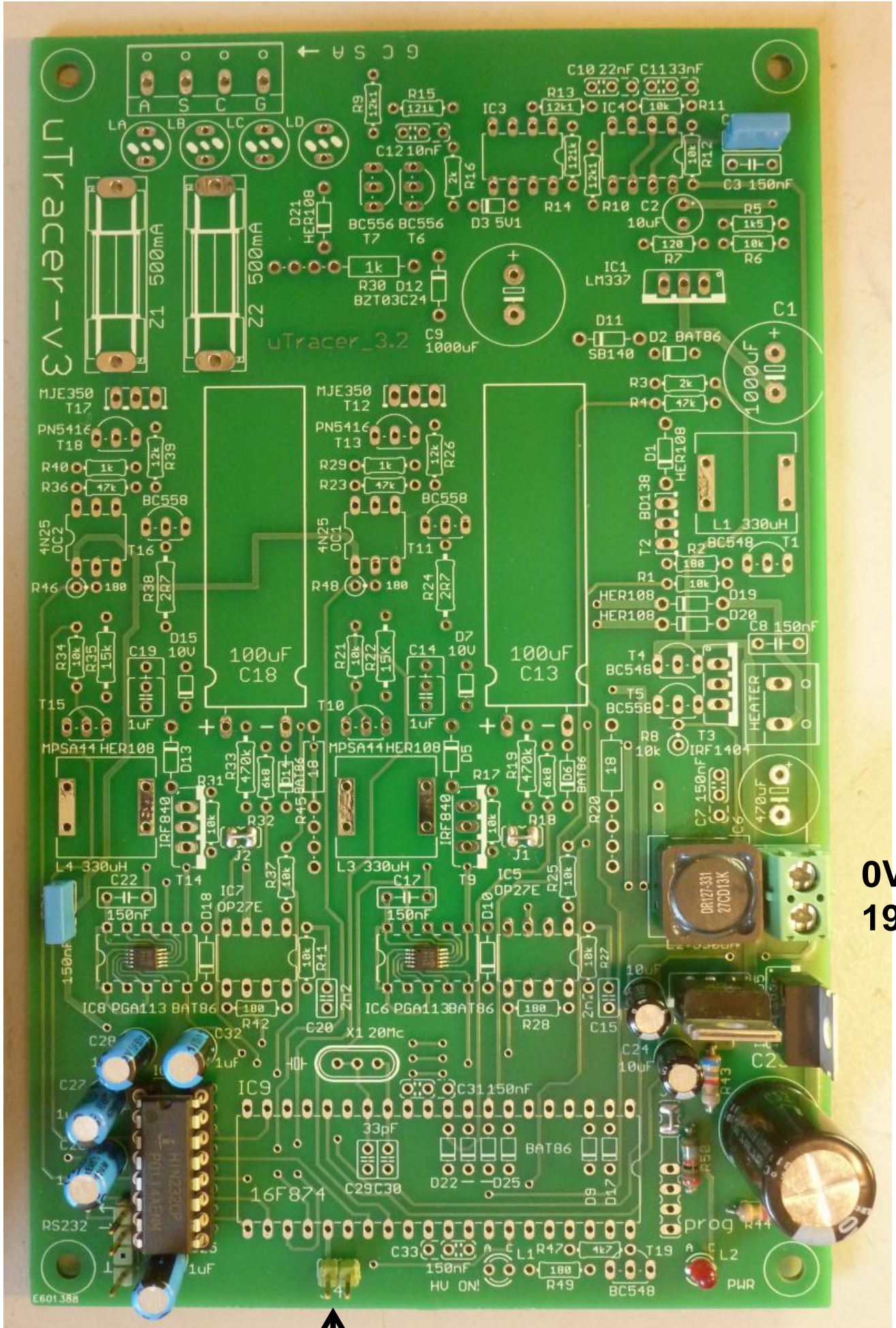
J4 only used during testing !
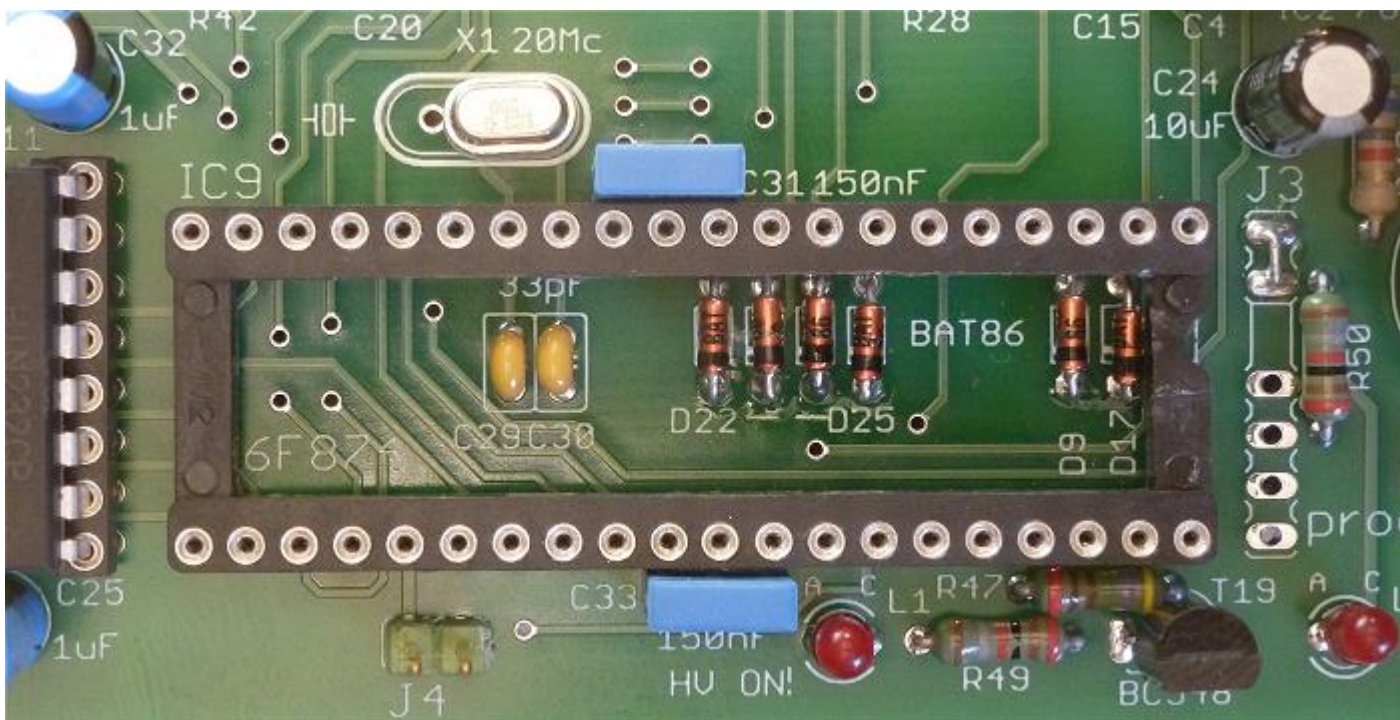
Part 4. RS232 Interface - testing

- ☐ 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
- ☐ 4. make sure jumper J4 is in place
- ☐ 5. start the GUI
- ☐ 6. switch on the power supply (supply current ≈ 35 mA @ 19 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:
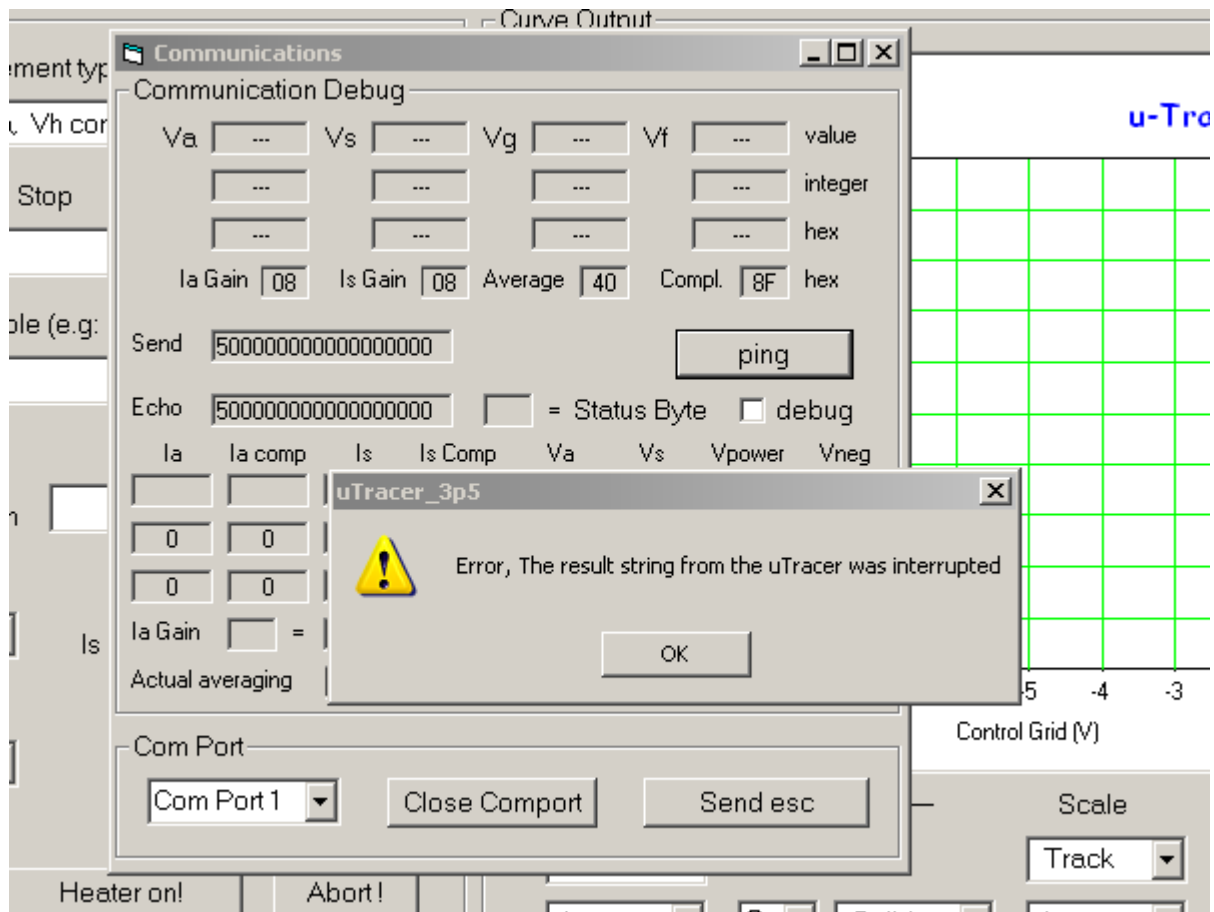
- ☐ 10. after ca. 10 sec the error message: "The result string from the uTracer was interrupted" should appear . This is the proper response
- ☐ 11. <u>be sure to remove jumper J4 !!</u>

Part 5. Micro-controller - construction

☑ 1. remove the center parts of the 40 pin IC socket (see picture)

☐ 2. C29, 33 pF

☐ 3. C30, 33 pF

☐ 4. D22, BAT86, note polarity !

☐ 5. D23, BAT86, note polarity !

☐ 6. D24, BAT86, note polarity !

☐ 7. D25, BAT86, note polarity !

☐ 8. D17, BAT86, note polarity !

☐ 9. D9, BAT86, note polarity !

☐ 10. C31, 150 nF

☐ 11. C33, 150 nF

☐ 12. X1,   20 MHz Xtal,

The Xtal is located on the conductive foam.
Insert plastic isolator between Xtal and PCB,
see "Tips and Tricks" section !

☐ 13. R47, 6k8

☐ 14. R49, 180 ohm

Note: R47 is marked
4k7 on the PCB

⚠

☐ 15. T19, BC548

☐ 16. LED1 "High Voltage On" (anode = long pin)

☐ 17. place 40 pin IC socket for PIC processor , NOTE ORIENTATION !

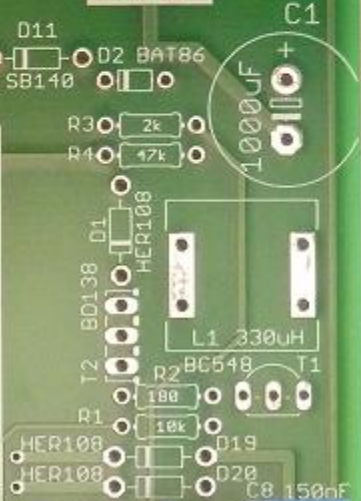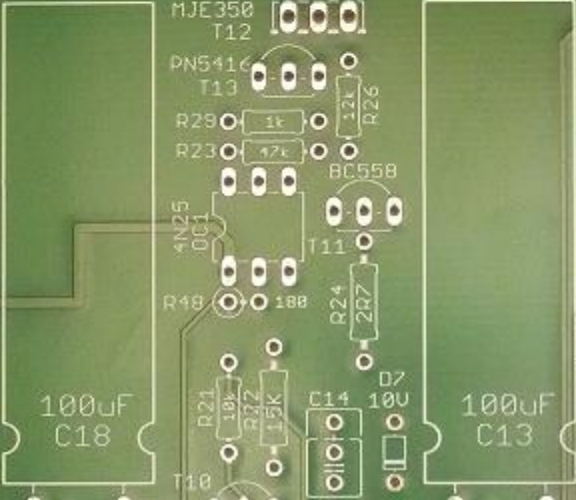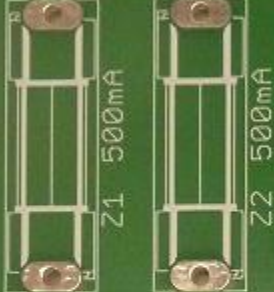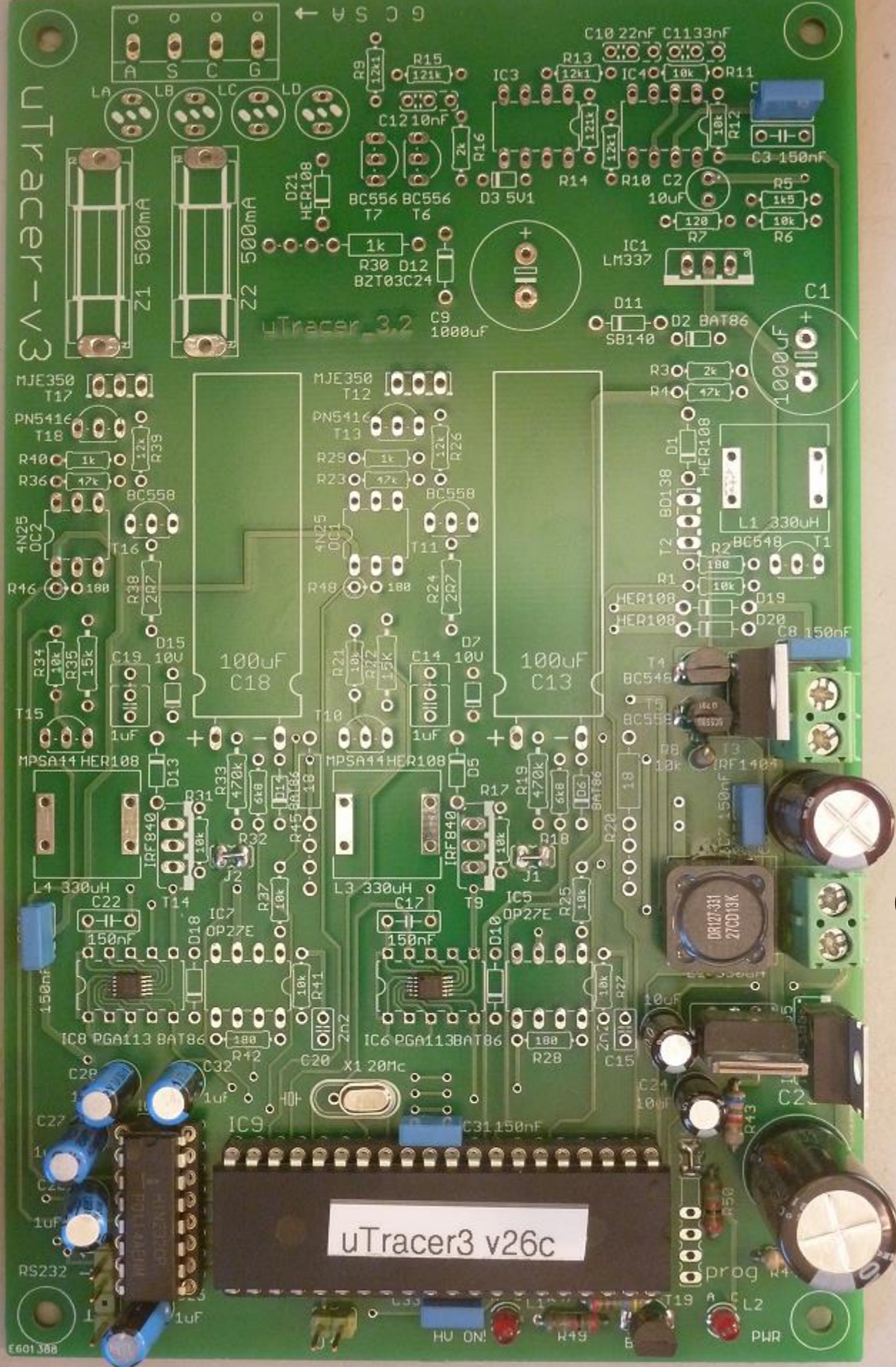☐ 20. insert PIC processor 16F874, NOTE ORIENTATION !

Part 5. Micro-controller - testing

Note: During this test, and the following test the HV LED will be on!
Only when the construction of the uTracer is completed, the HV LED will function correctly!

☐ 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

☐ 5. start the GUI

☐ 6. switch on the power supply (supply current ≈ 60 mA @ 19 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:

Vsupl
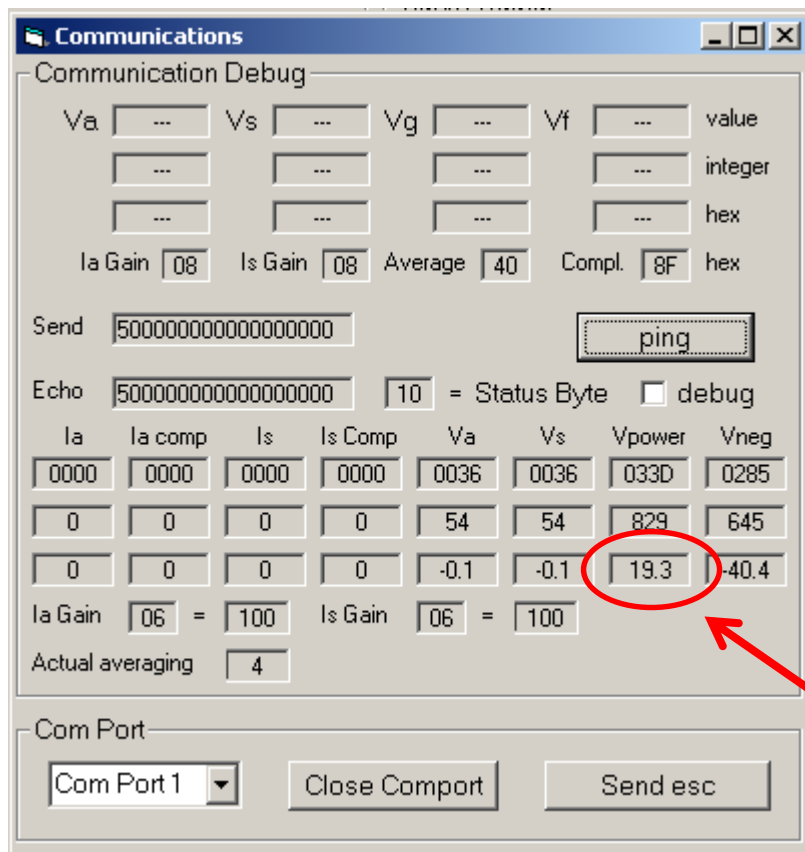
☐ 10. No error message should appear !

☐ 11. The value in the circle is the supply voltage as measured by the controller

Part 5. Micro-controller - calibration

☐ 1. continue from the situation on the previous page

☐ 2. measure the exact supply voltage using a multi-meter or DVM

☐ 3. if the measured supply voltage is not equal to the supply voltage
        measured by the uTracer displayed on the
        "Debug/Communications" form (see previous page) , continue ….

☐ 4. open the Calibration form by pressing "Cal." in the "Miscellaneous"
        section of the main GUI form

☐ 5. adjust the Vsupl slide bar

☐ 6. press "ping" on the "Debug/Communications" form

☐ 7. check if the supply voltage measured by the uTracer is now equal
        to the exact supply voltage

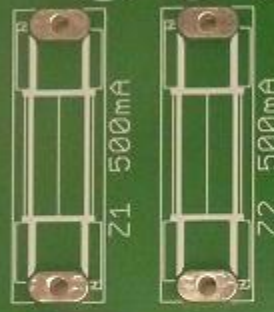☐ 8. if not ,repeat steps 5, 6, 7

☐ 9. when equal, press "Save to Colibration File" on the Calibration form
        to save the calibration value to the calibration file

Part 6. Heater supply - construction

☐ 1. R8, 10 k

☐ 2. T5, BC558

☐ 3. T4, BC548

☐ 4. C7, 150 nF

☐ 5. C8, 150 nF

☐ 6. C6, 470 uF, note polarity !

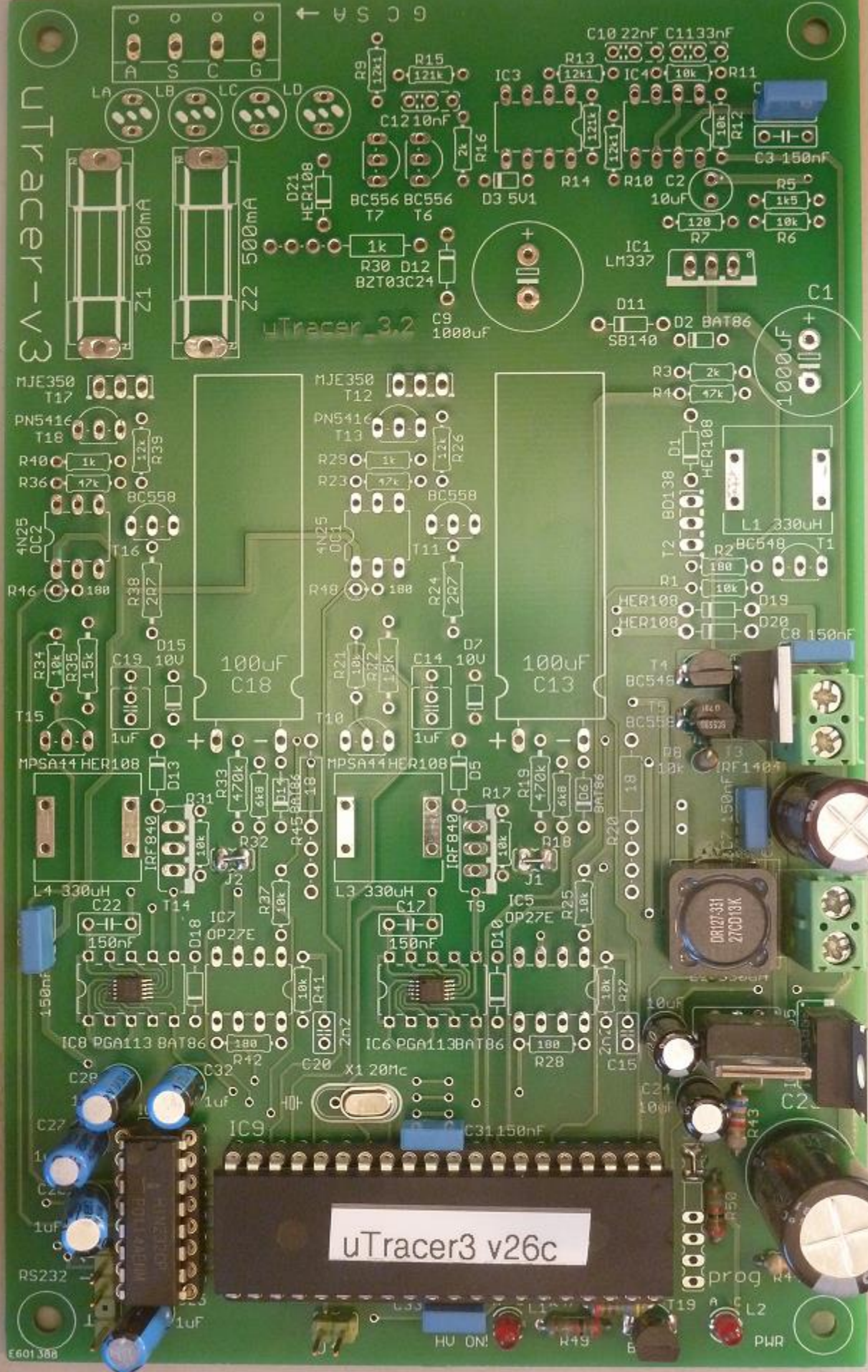☐ 7. screw terminal for heater

☐ 8. T3, IRF1404Z
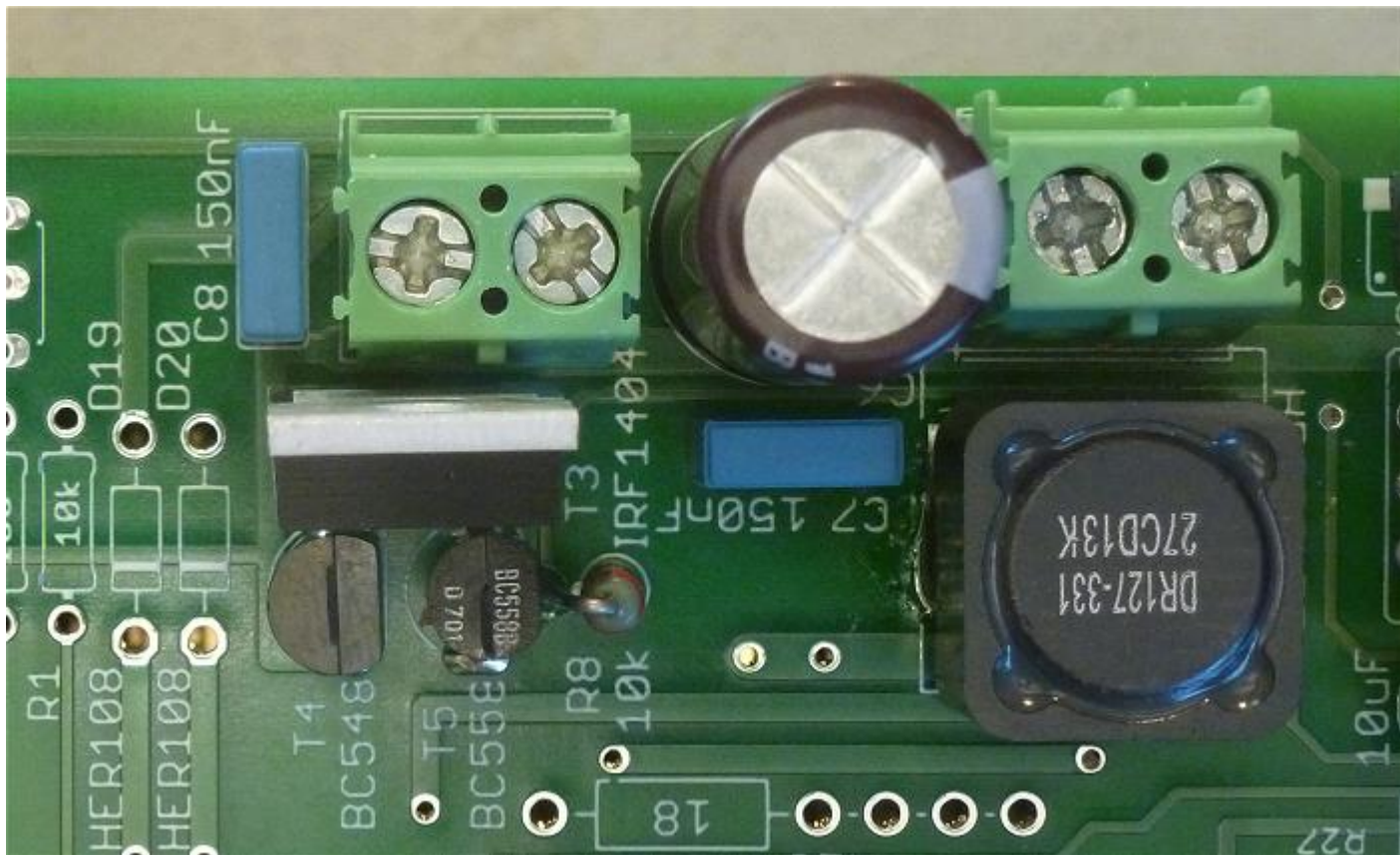
Part 6. Heater supply - testing

- ☐ 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
- ☐ 4. Connect a lamp 6 or 12 volts / ca. 100 mA to the heater terminals
- ☐ 5. start the GUI
- ☐ 6. switch on the power supply of the uTracer
- ☐ 7. enter the voltage of the lamp in the designated field (arrow)
- ☐ 8. press "Heater on!"
- ☐ 9. the lamp should now be switched on in 10 sec
- ☐ 10. Press "Abort!" to switch off the lamp
- ☐ 11. Replace the lamp for "a heavier" lamp of ca. 1 to 1.5 A
- ☐ 12. Repeat steps 7 to 10
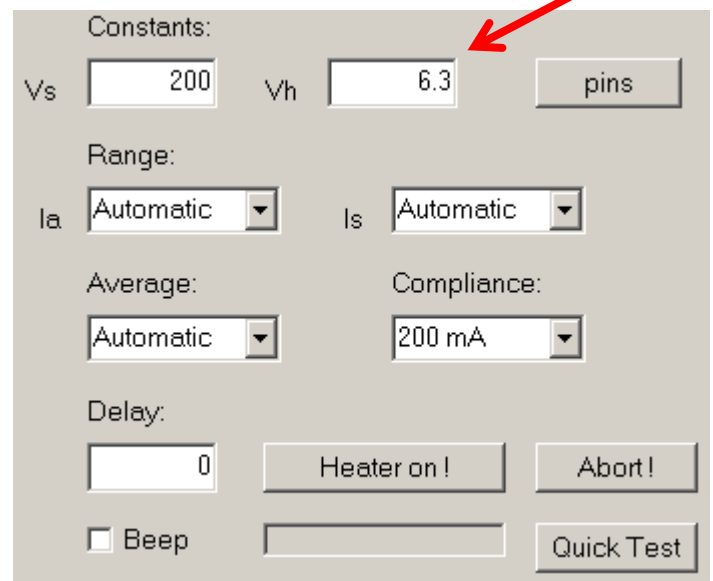
The power MOSFET should remain cool even for a 1.5 A lamp

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 7. Negative Power Supply - construction

- ☐   1. R1, 10k
- ☐   2. R2, 180 ohm
- ☐   3. L1, 330 uH, see mounting tips for inductors in "Tips and Tricks"
- ☐   4. T1, BC548
- ☐   5. T2, BD138 note orientation (see figures)
- ☐   6. D1, HER108
- ☐   7. R4, 47 k
- ☐   8. R3, 2 k
- ☐   9. D2, BAT86
- ☐ 10. C1, 1000 uF, 50 V, note polarity !
- ☐ 11. R5, 1k5
- ☐ 12. R6, 10k
- ☐ 13. R7, 120 ohm
- ☐ 14. C2, 10 uF, note polarity !
- ☐ 15. C3, 150 nF
- ☐ 16. IC1, LM337, note orientation !
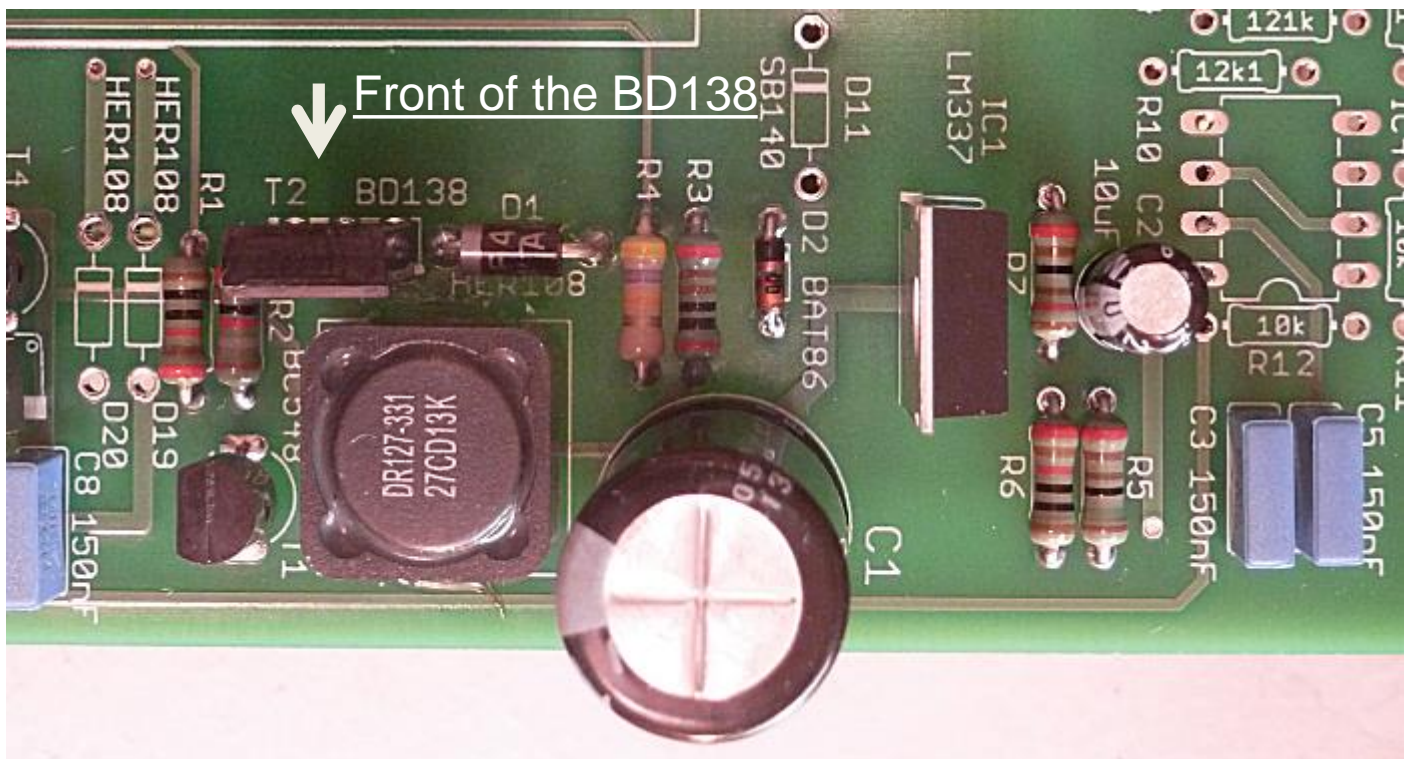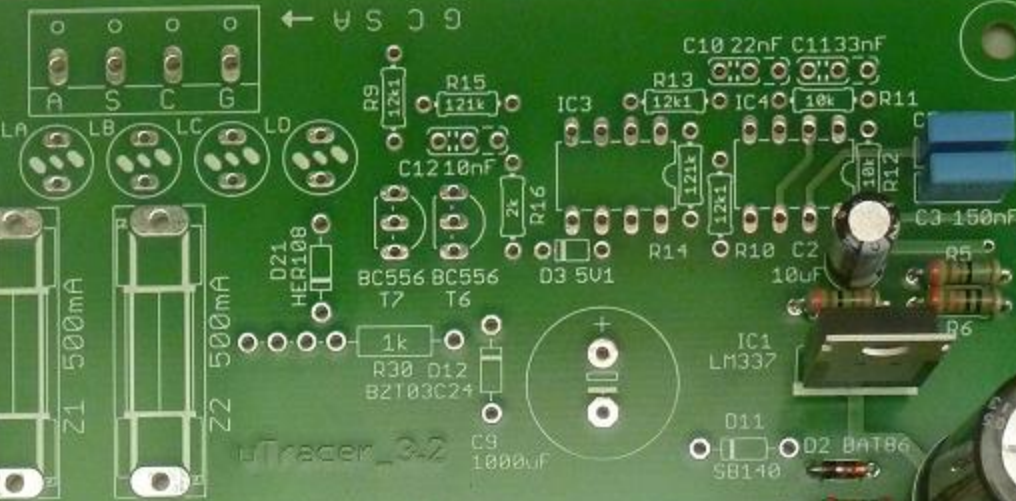
Frontside of the BD138
(the side with text)

Part 7. Negative Power Supply - 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 ≈ 95 mA @ 19 V)

☐   2. measure the - 40 V output voltage of the inverting boost converter
        on the test point indicated in the photograph

☐   3. measure the - 15 V output voltage of the negative voltage regulator
        on the test point indicated in the photograph

-15 V →

-40 V →
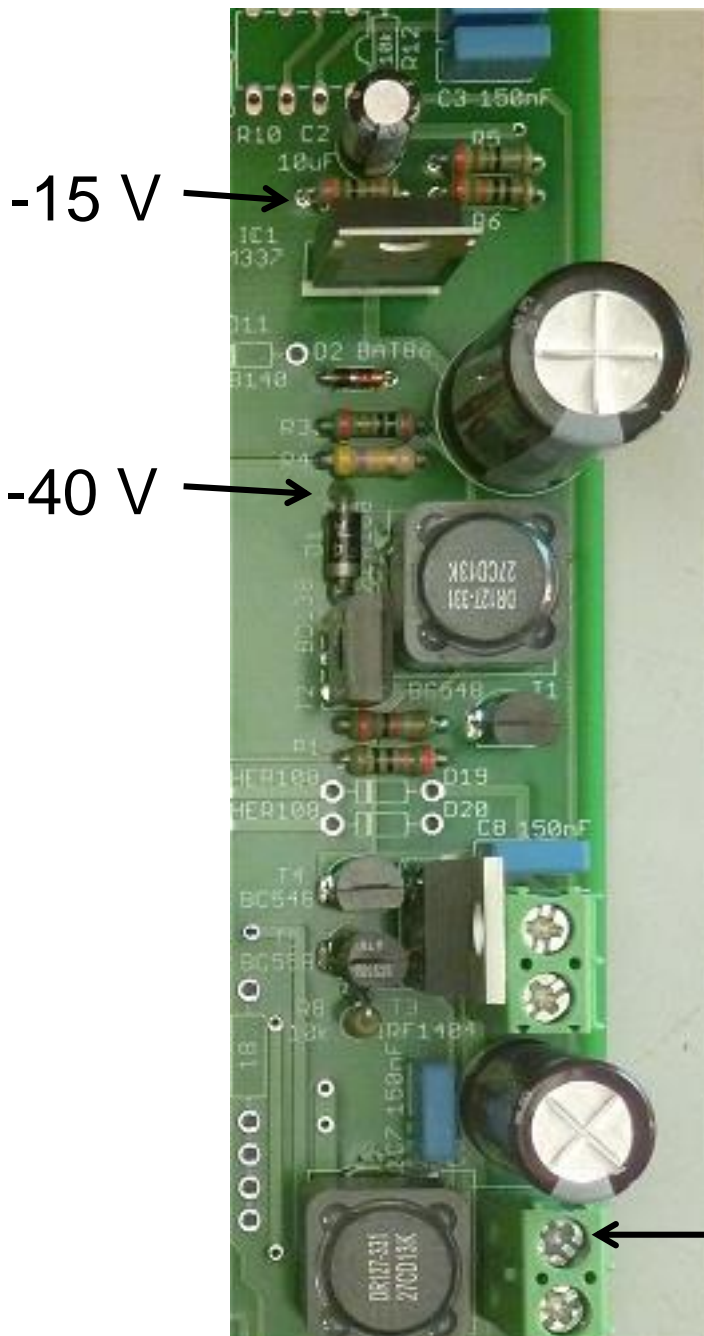
be careful
not to slip
with the
test pins!

During operation the
negative power supply can
produce a high hissing sound

Part 8. Grid Bias Circuit - construction

- ☐ 1. R12, 10 k
- ☐ 2. R11, 10 k
- ☐ 3. R9, 12k1
- ☐ 4. R10, 12k1
- ☐ 5. R13, 12k1
- ☐ 6. R14, 121k
- ☐ 7. R15, 121k
- ☐ 8. R16, 2k

Note!
There are are 12k1 (12.1 kohm) and 121k (121 kohm) resistors.
Do not mix them up!

- ☐ 9. D3, 5V1 zener diode, note polarity !
- ☐ 10. C10, 220 nF (Note C10 is marked 22 nF on the PCB !)
- ☐ 11. C11, 330 nF   (Note C11 is marked 33 nF on the PCB !)
- ☐ 12. C12, 10 nF
- ☐ 13. 8 pin socket for IC3, note orientation !
- ☐ 14. 8 pin socket for IC4, note orientation !
- ☐ 15. T6, BC556
- ☐ 16. T7, BC556
- ☐ 17. modify La, Lb, Lc, Ld (see "Tips and Tricks" section)
- ☐ 18. place La, Lb,Lc, Ld
- ☐ 19. fuseholder for Z1
- ☐ 20. fuseholder for Z2
- ☐ 21. mount screw terminal for tube connection
- ☐ 22. D19, HER108, note polarity !
- ☐ 23. D20, HER108, note polarity !
- ☐ 24. D21, HER108, note polarity !
- ☐ 25. D12, 24V zener diode , 1N5934 ⟶

Note: D12 is marked BZT03C24 on the PCB

- ☐ 26. R30, 1 k / 5 Watt, use ceramic spacer beads (2 per side)
- ☐ 27. insert IC3, 741
- ☐ 28. insert IC4, NE5534
- ☐ 29. C9, 1000uF 50 V, note polarity !
- ☐ 30. D11, SB140, note polarity !

Part 8. Grid Bias Circuit – testing and calibration

☐   1. make sure that the GUI is properly installed and tested

☐   2. connect the uTracer to the PC
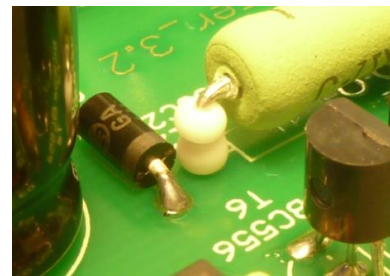
☐   3. connect the uTracer to the power supply, if possible set current limit
to 500 mA.

☐   4. Connect a DVM or multimeter between the cathode and grid
terminals

☐   5. start the GUI

cathode to
- DVM

grid to
+ DVM

☐   6. in the main form of the GUI fill in -40 V as start value for the grid bias

☐ 8. open the calibration form by pressing "Cal." in the "Miscellaneous" section of the main GUI form

☐ 9. switch on the power supply (supply current ≈ 200 mA @ 19 V)

☐ 10. press "Heater On!" on the main form

☐ 11. to skip the "slow heating" feature press again on "Heating …"

☐ 12. start a measurement by pressing "Measure Curve"

☐ 13. read the actual grid voltage on the DVM or multimeter

☐ 14. At this point a "time out" error message will appear because the GUI is expecting a measurement. Remove the error messages by clicking "ok"

☐ 15. Switch off the power of the uTracer to reset the uTracer

☐ 16. Adjust the VgridGain slider on the "Calibration" form if the measured grid voltage is not equal to -40 V

☐ 17. Repeat steps 9 to 15

☐ 18. When the measured grid voltage is equal to -40 V press the "Save to Calibration" button on the "calibration" form to save the calibration value to the calibration file

At the end of the construction of the uTracer an extended calibration of the grid bias will be done (part 11)

Part 9. Current Amplifiers - construction

- ☐ 1. D10, BAT86, note polarity !
- ☐ 2. D18, BAT86, note polarity !
- ☐ 3. R42, 180 ohm
- ☐ 4. R28, 180 ohm
- ☐ 5. R41, 10k
- ☐ 6. R27, 10k
- ☐ 7. R37, 10k
- ☐ 8. R25, 10k
- ☐ 11. 8 pin socket for IC5
- ☐ 12. 8 pin socket for IC7
- ☐ 13. D6, BAT86
- ☐ 14. D14, BAT86
- ☐ 15. C17, 150 nF
- ☐ 16. C22, 150 nF
- ☐ 17. L3, 330 uH
- ☐ 18. insert IC5, OPA227
- ☐ 19. insert IC7, OPA227
- ☐ 20. R45, 18 ohm, use 2 ceramic spacer beads /side ⎤ Do not cut
- ☐ 21. R20, 18 ohm, use 2 ceramic spacer beads /side ⎦ wires yet

note: the 18 ohm resistors shown on the photo are larger than
the ones supplied with the kit !

⚠️ C15 and C20 are not used
In the uTracer3+

Part 9. Current Amplifiers - testing

Explanation: To test the current amplifiers a trick is used.
By connecting two resistors of 12k1 between the current sense resistors
R20 and R45 and the -15 V supply, a anode / screen current of 15/12k1 =
1.24 mA is simulated.

☐   1. connect two 12k1 resistors as shown below
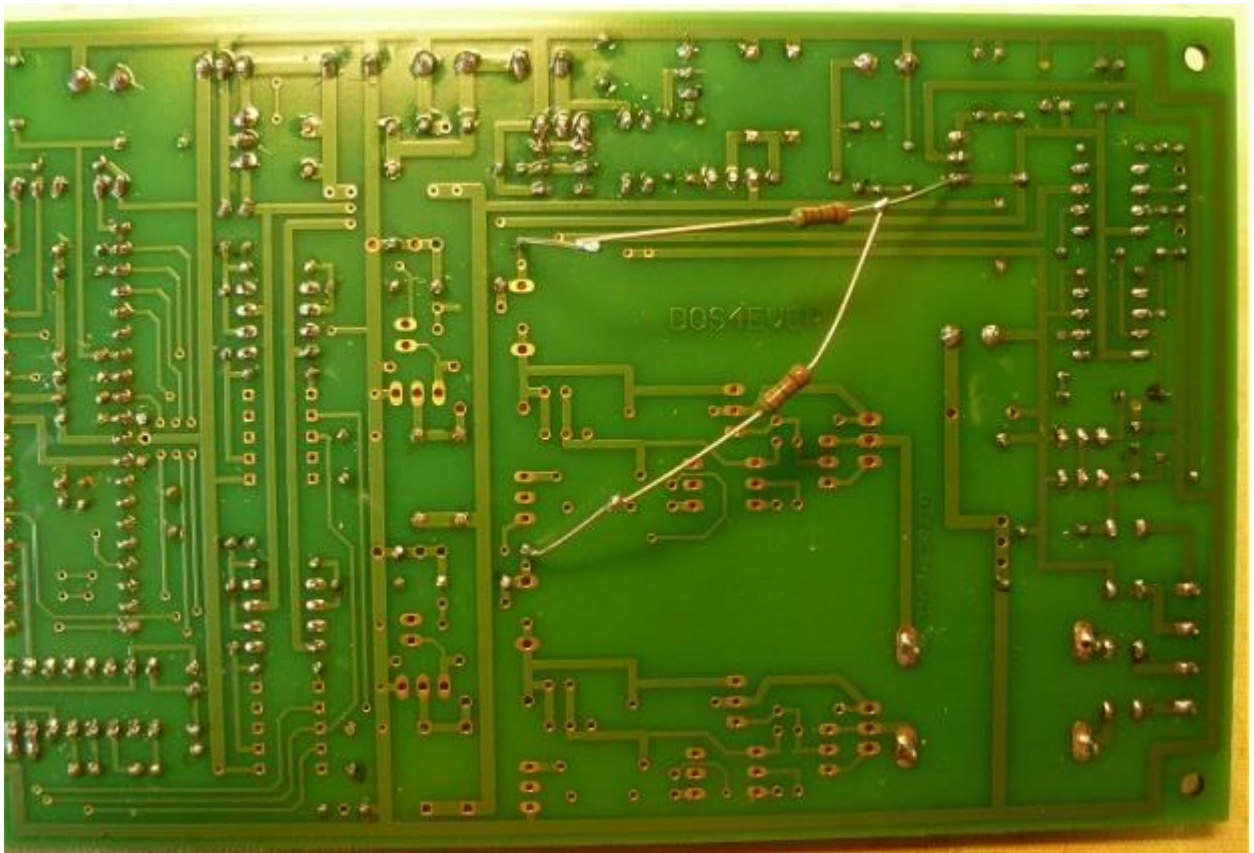
☐   2. make sure that the GUI is properly installed and tested

☐   3. connect the uTracer to the PC

☐   4. connect the uTracer to the power supply, if possible set current limit
           to 500 mA

☐   5. start the GUI

☐   6. switch on the power supply (supply current ≈ 225 mA @ 19 V)

☐ 7. Switch off auto-ranging and auto-averaging functions by selecting
the 0 – 2 mA ranges for both the anode as well as the screen
currents, and by setting average to "none " (see picture below)
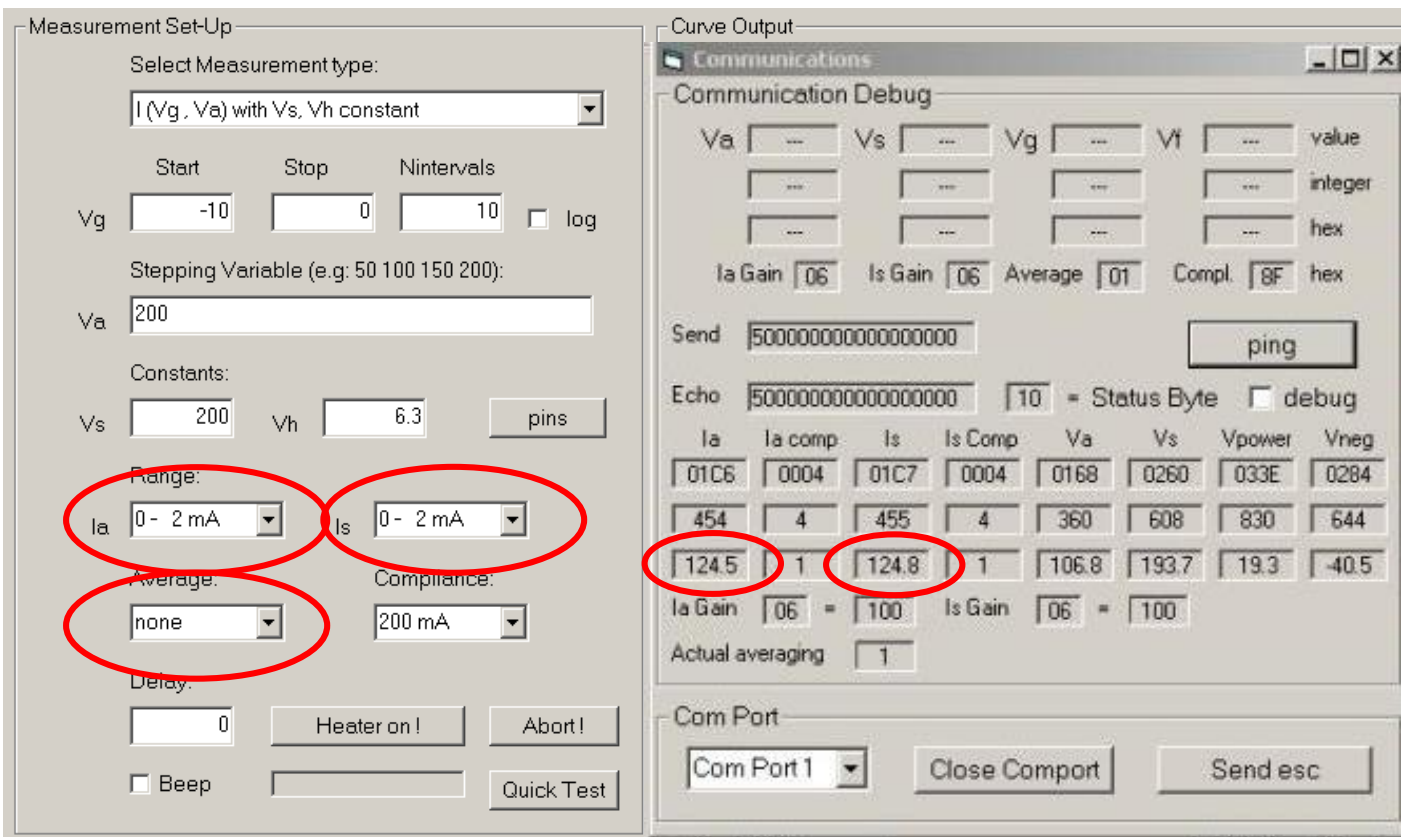
☐ 8. open the "Debug / Communications" form by pressing the "Debug"
command button in the "Miscellaneous" section of the main form
of the GUI

☐ 9. press the "ping" command button on the "Debug / Communications"
form

☐ 10. The returned values for the anode and screen currents on the
Debug / Communications" form should now display a value
of approximately 124 which is the current (1.24) times the gain
factor (100x). The exact calibration will be done later on

☐ 11. Remove the resistors and clip the ends of R20 and R45

Part 10. Boost converters and HV switches - construction

- [ ] 1. R40, 1k
- [ ] 2. R29, 1k
- [ ] 3. C36, 2n2
- [ ] 4. C38, 2n2
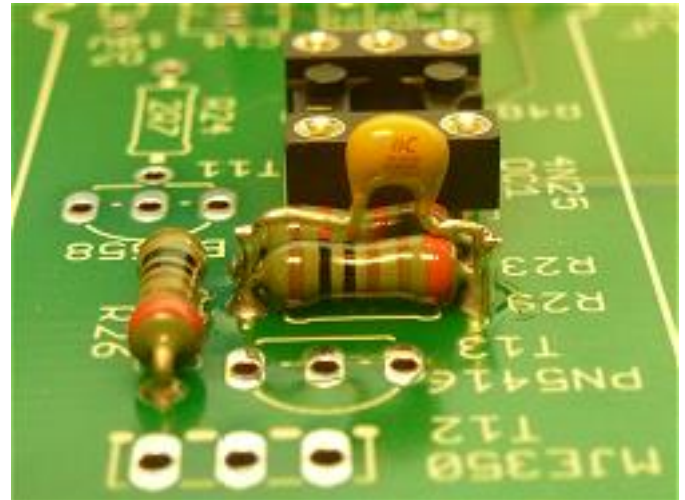
C36 and C38 are soldered
"piggy back" on R29 and R40
(The photo on the right was taken at a later stage
in the assembly when more
components were present on the board)

- [ ] 5. R23, 12k1 ⎤
- [ ] 6. R36, 12k1 ⎦ Note: R23 and R36 are marked 47k on the PCB ⚠
- [ ] 7. R26, 1k ⎤
- [ ] 8. R39, 1k ⎦ Note: R26 and R39 are marked 12k on the PCB ⚠
- [ ] 9. 6 pin socket for OC1
- [ ] 10. 6 pin socket for OC2
- [ ] 11. R46, 100 ohm ⎤
- [ ] 12. R48, 100 ohm ⎦ Note: R46 and R48 are marked 180 on the PCB ⚠
- [ ] 13. R21, 10k
- [ ] 14. R34, 10k
- [ ] 15. R22, 15k, 0.5W, use ceramic spacer beads
- [ ] 16. R35, 15k, 0.5W, use ceramic spacer beads
- [ ] 17. D7, 10V zener, note orientation !
- [ ] 18. D15, 10V zener, note orientation !
- [ ] 19. C14, 1uF, foil capacitor
- [ ] 20. C19, 1uF, foil capacitor
- [ ] 21. D5, HER108, note polarity !
- [ ] 22. D13, HER 108, note polarity !
- [ ] 23. R18, 5.23K ⎤
- [ ] 24. R32, 5.23K ⎦ Note: R18 and R32 are marked 6k8 on the PCB ⚠

- ☐ 25. R19, 470k, 0.5W, use ceramic spacer beads
- ☐ 26. R33, 470k, 0.5W, use ceramic spacer beads
- ☐ 27. L4, 330 uH, see mounting tips for inductors in "Tips and Tricks"
- ☐ 28. R24, 2.7 ohm, 0.5W , use ceramic spacer beads
- ☐ 29. R38, 2.7 ohm, 0.5W , use ceramic spacer beads
- ☐ 30. T11, BC558
- ☐ 31. T16, BC558
- ☐ 32. T10, KSP44BU
- ☐ 33. T15, KSP44BU
- ☐ 34. T13, KSP94BU
- ☐ 35. T18, KSP94BU

Note: T13 and T18 are marked PN5416 on the PCB ⚠

- ☐ 36. T9, SPA07N60
- ☐ 37. T14, SPA07N60

Note: T9 and T14 are marked IRF840 on the PCB ⚠

- ☐ 38. T12, KSA1156
- ☐ 39. T17, KSA1156

Note: T12 and T17 are marked MJE350 on the PCB ⚠

- ☐ 40. insert OC1, 4N25
- ☐ 41. insert OC2, 4N25
- ☐ 42. C13, 100uF, 450V, see tips for bending lead in "Tips and Tricks"
- ☐ 43. C18, 100uF, 450V, see tips for bending lead in "Tips and Tricks"
- ☐ 44. insert fuse Z1
- ☐ 45. insert fuse Z2

Continue next page

⚠ On the board there is space for two 10 k resistors marked R17 and R31. These resistors are not used.

☐ 46. C35, 10 nF
☐ 47. C37, 10 nF

These components are placed on the bottom side of the board see photo below ⚠
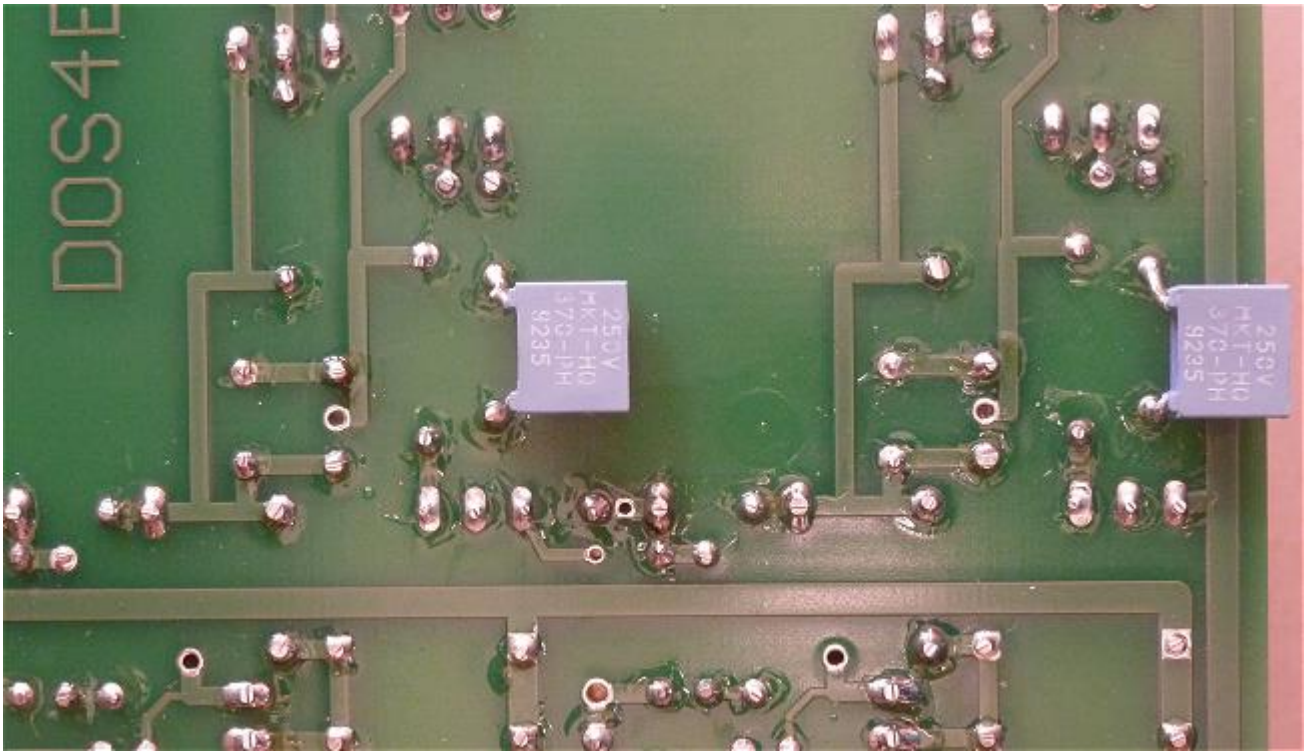
☐ 48. R51, 470
☐ 49. R52, 470

These components are placed on the bottom side of the board see photo below ⚠
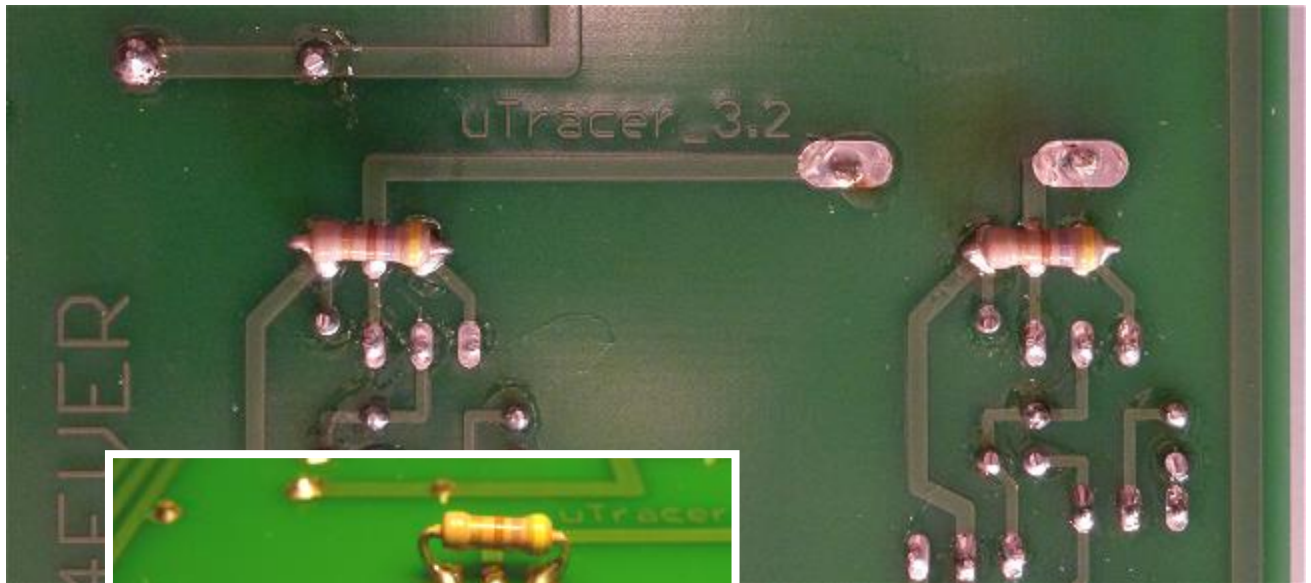
☐ 50. heatsink on IC1 (LM337)

bend the lower fins away from resistors

☐ 51. heatsink on IC10 (7805)

make sure screw does not touch the 7815!
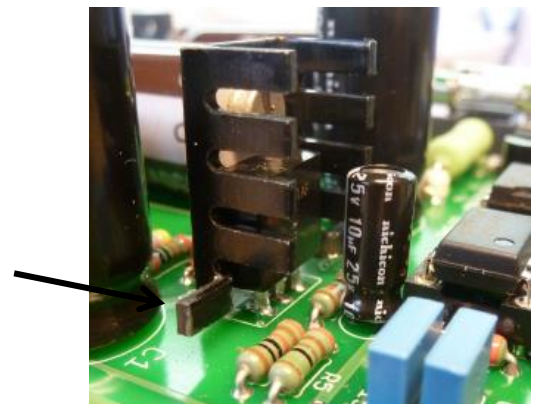
Frontside of the KSA1156
(the side with text)

0V
19V

A high resolution photograph of the PCB can be downloaded from:
www.dos4ever.com/uTracer3/3+Front.jpg

Part 10. Boost converters and HV switches -

testing and calibration

- ☐ 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.
- ☐ 5. start the GUI
- ☐ 6. switch on the power supply (supply current ≈ 225 mA @ 19 V)
- ☐ 7. measure the exact supply voltage with a multimeter or DVM: Vsupl
- ☐ 8. switch the power supply off
- ☐ 9. connect the multimeter / DVM to the C18, the anode reservoir capacitor , using two test cables with suitable clips

- ☐ 10. switch on the power supply

The testing and calibration will be done in three steps:

A. functionality test of the boost converters

B. calibration of the boost converters

C. testing and calibration of the high voltage switches and current amplifiers

Part A. functionality test boost converters

☐   11. continue from line ten on the previous page

☐   12. set up the measurement as shown in the screen dump below.

This will cause a "sweep" of the anode and screen voltages

consisting of two values: 50 and 100 V. The delay of 5 seconds

causes the uTracer to pause when the set point is reached

☐   13. start a measurement by clicking on "Heater On" followed by

clicking on "Measure Curve"

☐   14. verify on the DVM that the voltages reached are approximately:

50 + Vsupl and 100 + Vsupl

☐   15. In case this test is passed slowly increase the maximum voltage by

testing with the following sequences:

start: 50   stop: 150   Nint: 2

start: 50   stop: 200   Nint: 3

start: 50   stop: 250   Nint: 4

start: 50   stop: 300   Nint: 5

□ 16. switch off the power supply

□ 17. connect the measurement clips to the screen reservoir capacitor
        C13

□ 18. switch on the power supply

□ 19. repeat steps 12, 13, 14

Part B. calibration of the boost converters

Warning ! During this test lethal voltages present

□ 20. continue from step 19, assume the DVM is still connected to the
        screen reservoir capacitor C13 and that the power is on

□ 21. set up the measurement sequence shown below (Va= Vs = 100 and
        200 V with 5 sec delay)

□ 22. calculate   X = 200 + Vsupl   (Vsupl measured in step 7)

□ 23. open the calibrate form by clicking "Cal." in the miscellaneous
        section of the main GUI form

☐ 24. start a measurement by clicking on "Heater On" followed by
clicking on "Measure Curve"

☐ 25. in case the measured voltage during the 200 V phase of the
measurement does not equal X, adjust the VsGain slide bar
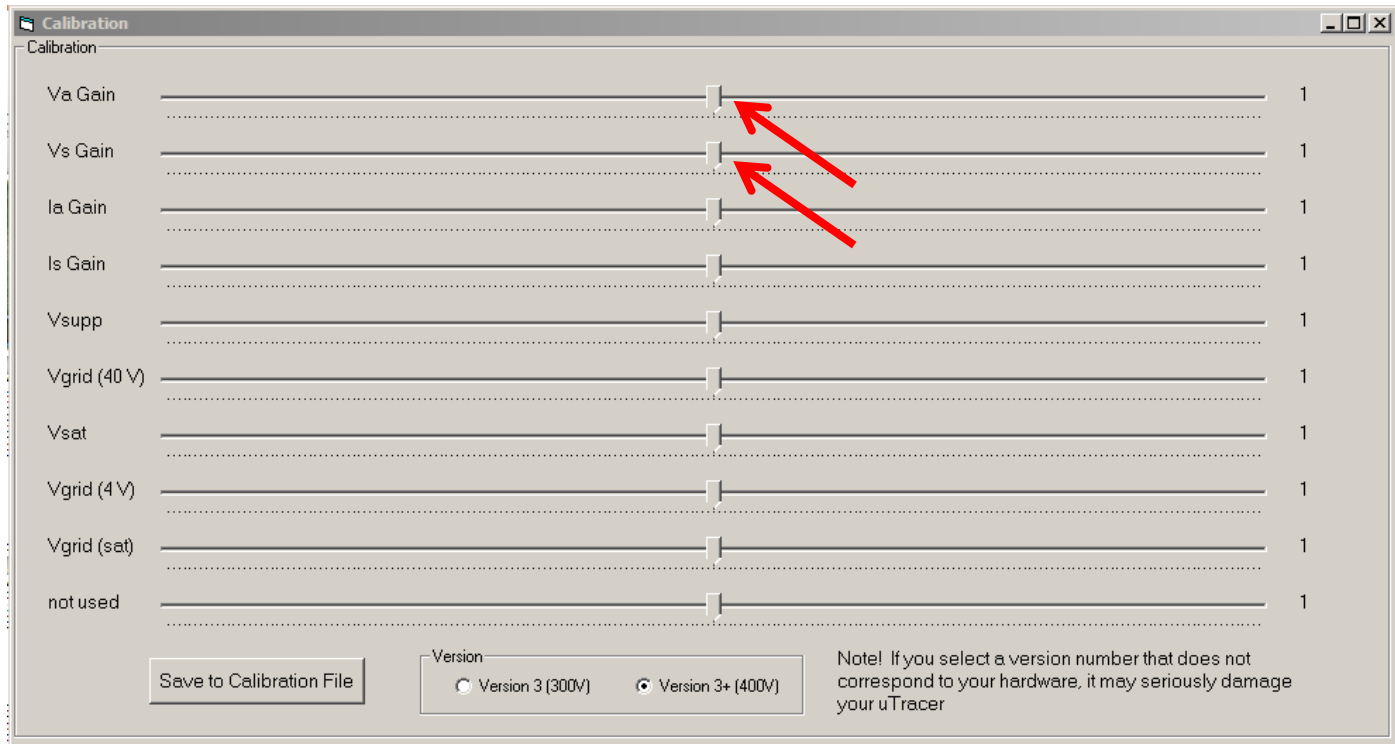
☐ 26. repeat steps 24, 25 until the measured voltage = X

☐ 27. save calibration value to calibration file by pressing "Save to
Calibration file"

☐ 28. wait until the "High Voltage" LED is off

☐ 29. switch off the power supply

☐ 30. connect the DVM to the anode reservoir capacitor C18 again.

☐ 31. repeat steps 20-29 again, but now for the anode voltage adjusting
the VaGain slide bar

☐ 32. wait until the "High Voltage" LED is off

☐ 33. switch off the power supply

☐ 34. disconnect the DVM

Part C. testing of the high voltage switches
and calibration of the current amplifiers

Warning ! During this test lethal voltages present

- ☐ 35. make sure that the power supply is off and that the capacitors are discharged
- ☐ 36. connect a 10 k / 1% resistor between the anode terminal and the cathode terminal
- ☐ 37. connect a 10 k / 1% resistor between the screen terminal and the cathode terminal
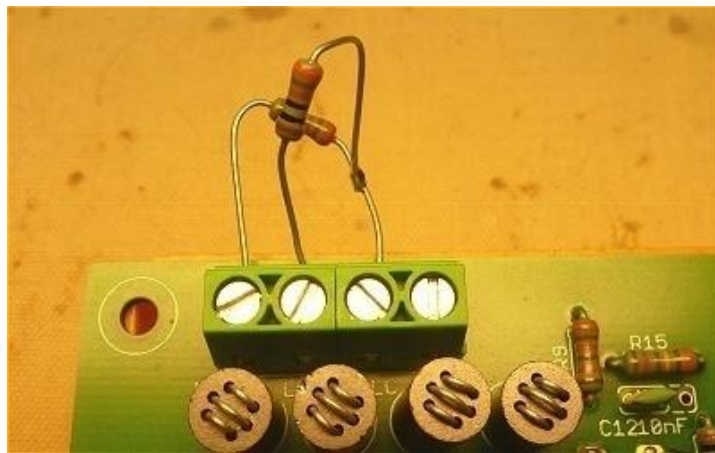
- ☐ 38. set up the measurement sequence shown below (sweep anode and screen voltages from 2 to 200 V in 20 steps with fixed gains)
- ☐ 39. open the calibrate form by clicking "Cal." in the miscellaneous section of the main GUI form
- ☐ 40. start a measurement by clicking "Heater On" followed by clicking on "Measure Curve". Verify that the GUI displays a straight line (a resistor !)
- ☐ 41. click with the mouse cursor on the last measurement point in the graph
- ☐ 42. read from the cursor the actual voltage Vx (198.5 V in example next page) and adjust the IaGain slide bar.
- ☐ 43. perform steps 40 to 42 until the current measured is Vx/ 10 [mA]

u-Tracer (V3.11)

Measurement Set-Up

Select Measurement type:

I (Va=Vs , Vg) with Vh constant

	Start	Stop	Nintervals	
Va = Vs	2	200	20	☐ log

Stepping Variable (e.g: 50 100 150 200):

Vg -1

Constants:

Vh 6.3   pins

Range:

Ia Automatic   Is Automatic

Average: Compliance:

Automatic   200 mA

Delay:

0   Measure Curve   Abort!

☐ Beep   Quick Test

Miscellaneous

Debug   Cal.   Save Data   Save Plot

☐ Ext. heater supply   Open Setup   Save Setup

Curve Output

Ia (mA)   u-Tracer V3.11.6

Vg
-1

Anode and Screen (V)

X= 195.8 Y1= 19.4

	Axis	— Style —	Scale	Min.	Max.	Ticks
X	Va = Vs		Track	0	200 V	10
Y1	Ia	○ Solid	Auto	0	20 mA	4
Y2	none	× Dots	Auto	0	1	10

☐ Keep Plot   Distortion   Clear   Pmax: 0 W   Poly Degree: 3
☑ Color
☑ Grid   Title: Title   ○ top ○ graph ● off   Store

☐ 44. set the Y1 axis to measure the screen current 'Is'

☐ 45. repeat steps 38 to 43 but now for the screen current adjusting the IsGain slide bar

☐ 46. when finished wait until the High Voltage LED is off

☐ 47. save calibration value to calibration file by pressing "Save to Calibration file"

☐ 48. switch off the power supply

☐ 49. remove the two test resistors

Part 11. Extended calibration of the grid bias circuit

In this part the final tuning of the grid bias circuit is performed.
This calibration will improve the accuracy at low grid voltages.
It is assumed that the coarse calibration of the grid circuit has
Been performed as described in section 8

☐ 1. Connect a DVM or multimeter between the cathode and grid terminals

☐ 2. start the GUI

cathode to
- DVM

grid to
+ DVM
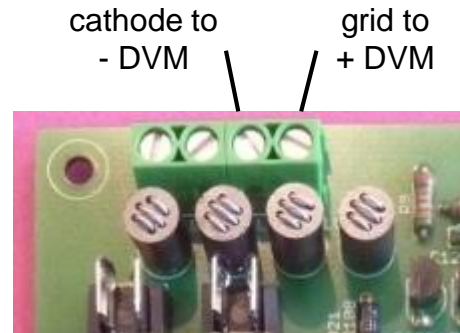
☐ 3. Open the calibration form

☐ 4. Set the "Vgrid (sat)" slider to 0.96 as a starting value

☐ 5. Select the measurement as shown below

Measurement Set-Up

Select Measurement type:

I (Va, Vg) with Vs, Vh constant

	Start	Stop	Nintervals	
Va	2	25	30	☐ log

Stepping Variable (e.g: 50 100 150 200):

Vg -4

Constants:

Vs 25   Vh 6.3   pins

☐ 6. Press "Measure Curve" on the main form.

☐ 7. Read the actual grid voltage on the DVM

☐ 8. Adjust the "Vgrid (4V)" gain slider on the calibration form in the
appropriate direction to close any discrepancy between the DVM
measurement and the -4V set value. (repeat steps 6-8)

☐　9. Check different grid voltages between -0.2 and -1.0 and adjust "Vgrid (sat)" to see if the calibration can be improved any further

☐　10. Press the "Save to Calibration File" button on the calibration form


Congratulations !

This completes the construction.

The uTracer is now ready for use

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 and output voltage in the range of 18 – 22 V. These power supplies combine a high output power with a small volume. It is strongly recommend to include a 330 uH inductor and a 1.5 A fuse in series with the + lead of the power supply.
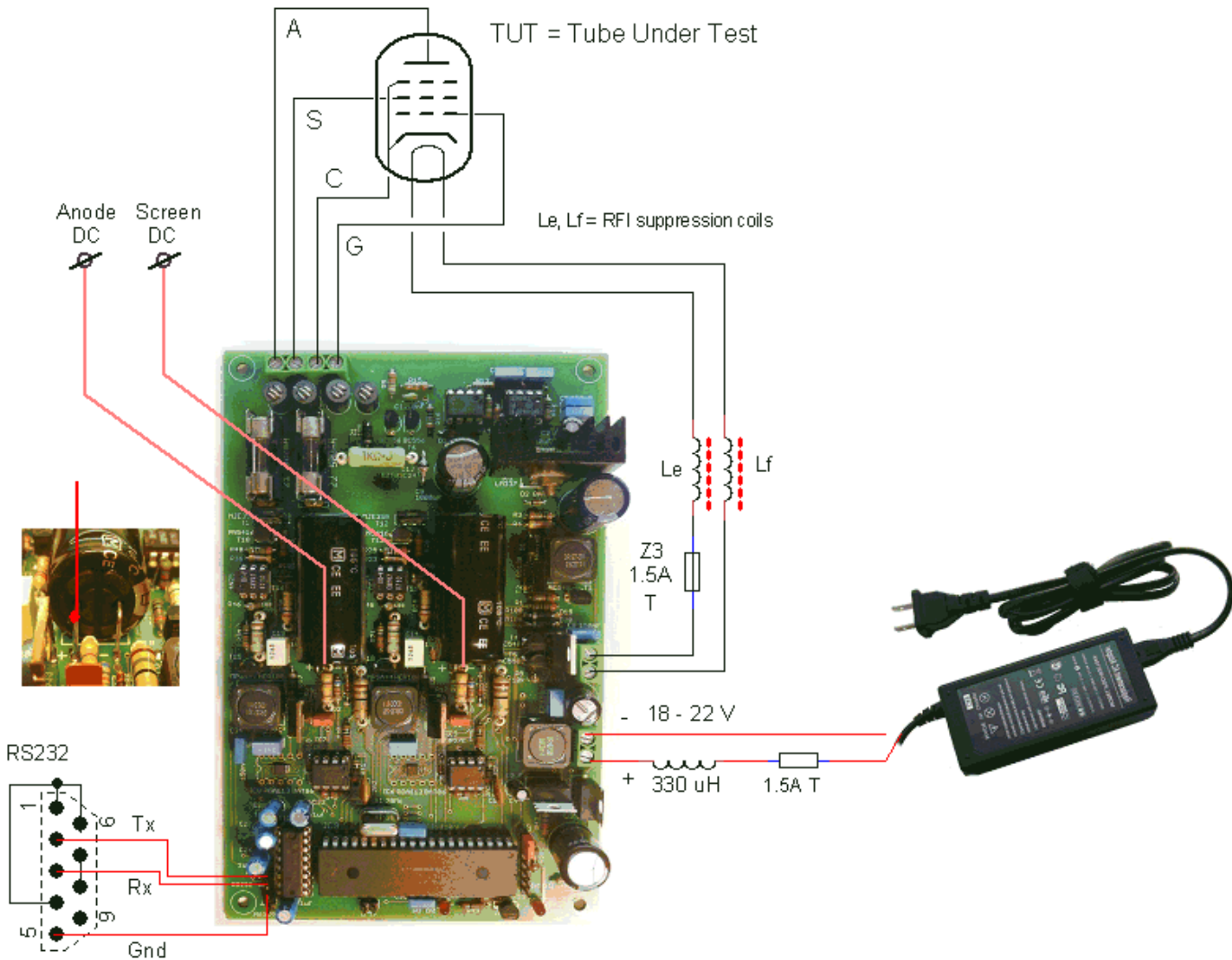
Additionally it is recommended to include a 1.5 A fuse and two RFI suppression beads in series with the heater connections. The connection of the other electrodes to the tubes will be discussed in the next section.

Optionally, the positive terminals of the two reservoir capacitors may be made available for connection to the tube electrodes. The boost converters can supply several milliamps of current so that in this way DC measurements at low current levels are possible. This makes it e.g. possible to test magic eyes.

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.
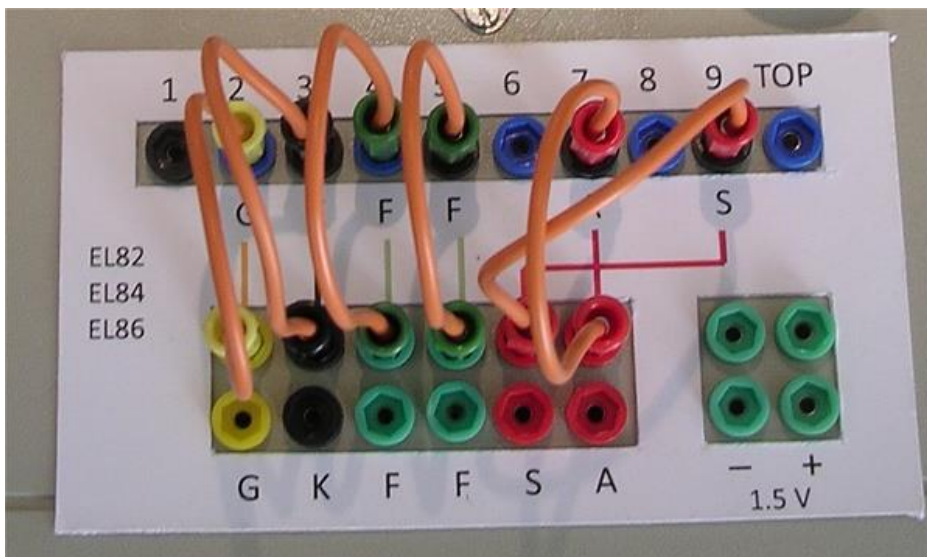
The top row of plugs is connected to the tube sockets. The bottom row is connected to the uTracer terminals. The bottom row contains a double set of plugs so that it is possible to connect two tube pins to one terminal e.g. to connect a pentode as triode (by connecting both the screen grid as well as 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.
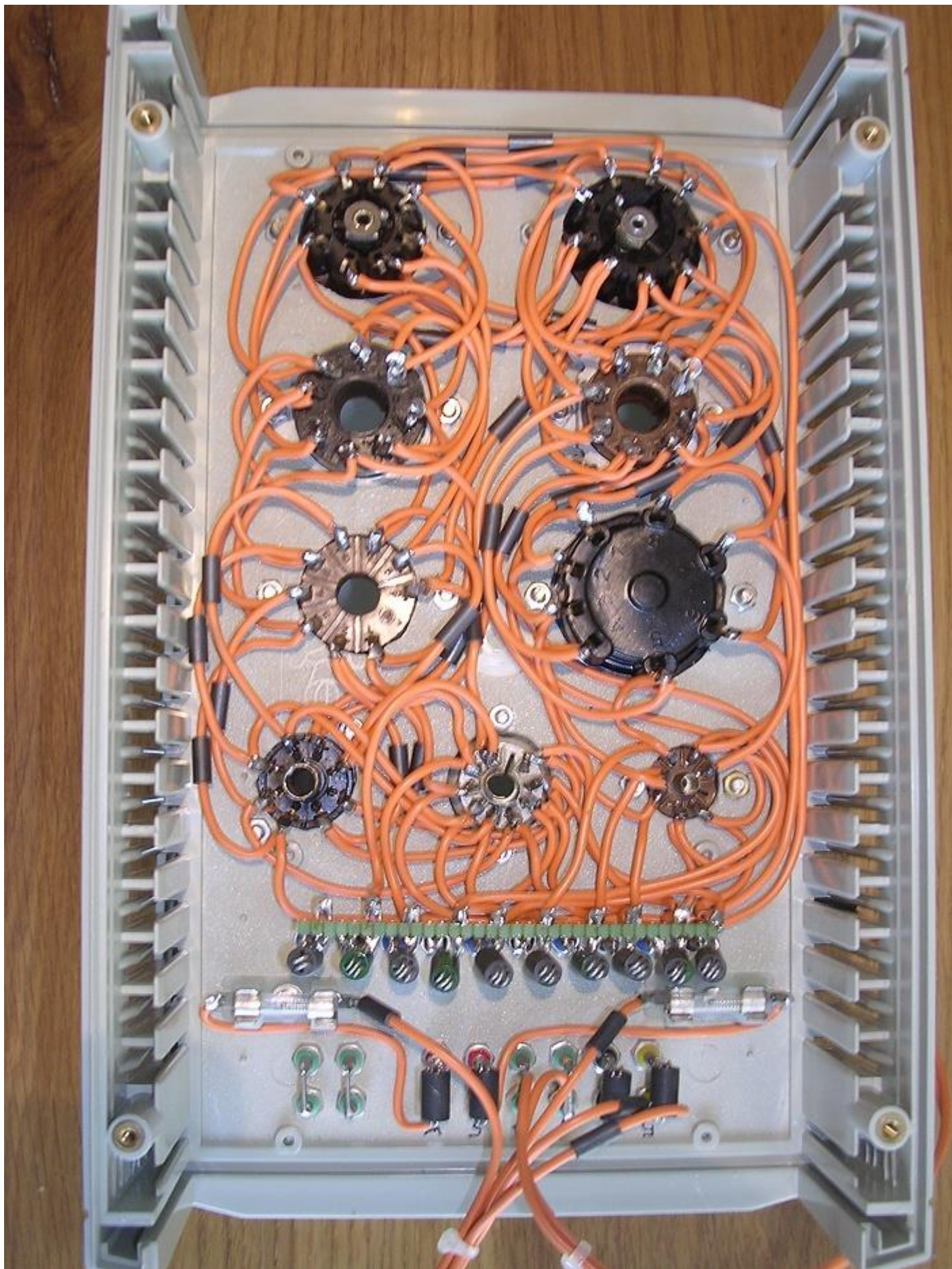
With conventional tubetesters, it can very well happen that a tube is destroyed because a destructive oscillation occurs! A big advantage of the uTracer is that the maximum energy stored in the reservoir capacitors is very small, too small to destroy a tube. The oscillations can however disturb the measurement 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.
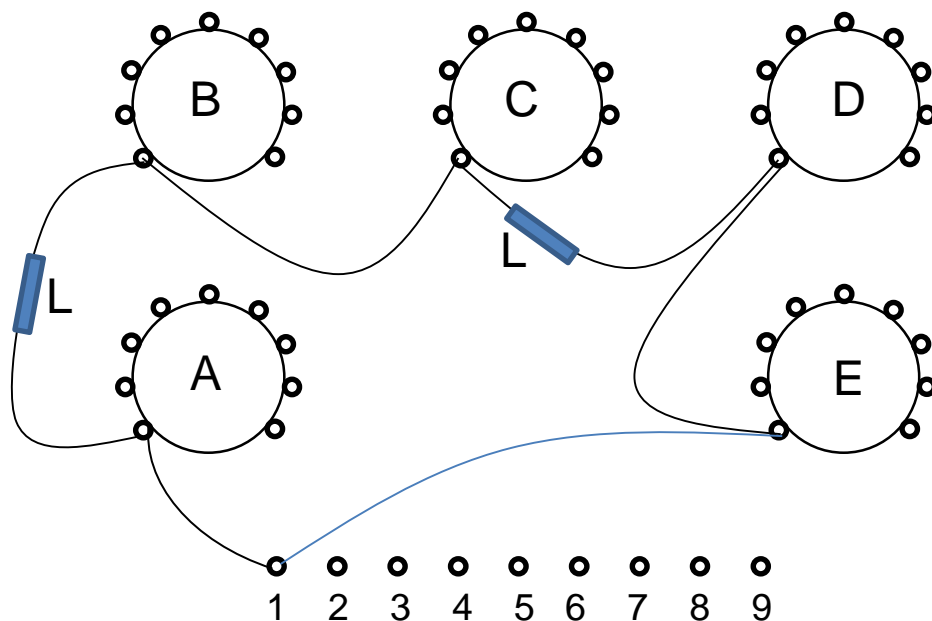
The wire connecting say pin one of each tube socket runs in a loop which also connects to the banana plug or rotary switch. At certain intervals an RF suppression bead (e.g. Wuerth no: 74270015) is shifted over the wires. The physical lengths of all the loops has to be approximately the same. The figure 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,

21 Feb 2016

Diodes marked HER108 can also be: UF1J or UF4007

Diodes D19–D21 HER108

to filament driver

La – Lf RFI
suppression beads
see text

2X BC556

filament

IC4, IC 5 and IC 7 are supplied from +15V / -15V

in-circuit program connector

master clear
Gnd
prgram
prog. data
prog. clock

J3
+5V

+5V
"High Voltage On!"
L1
R49
180
T19
BC548
R47
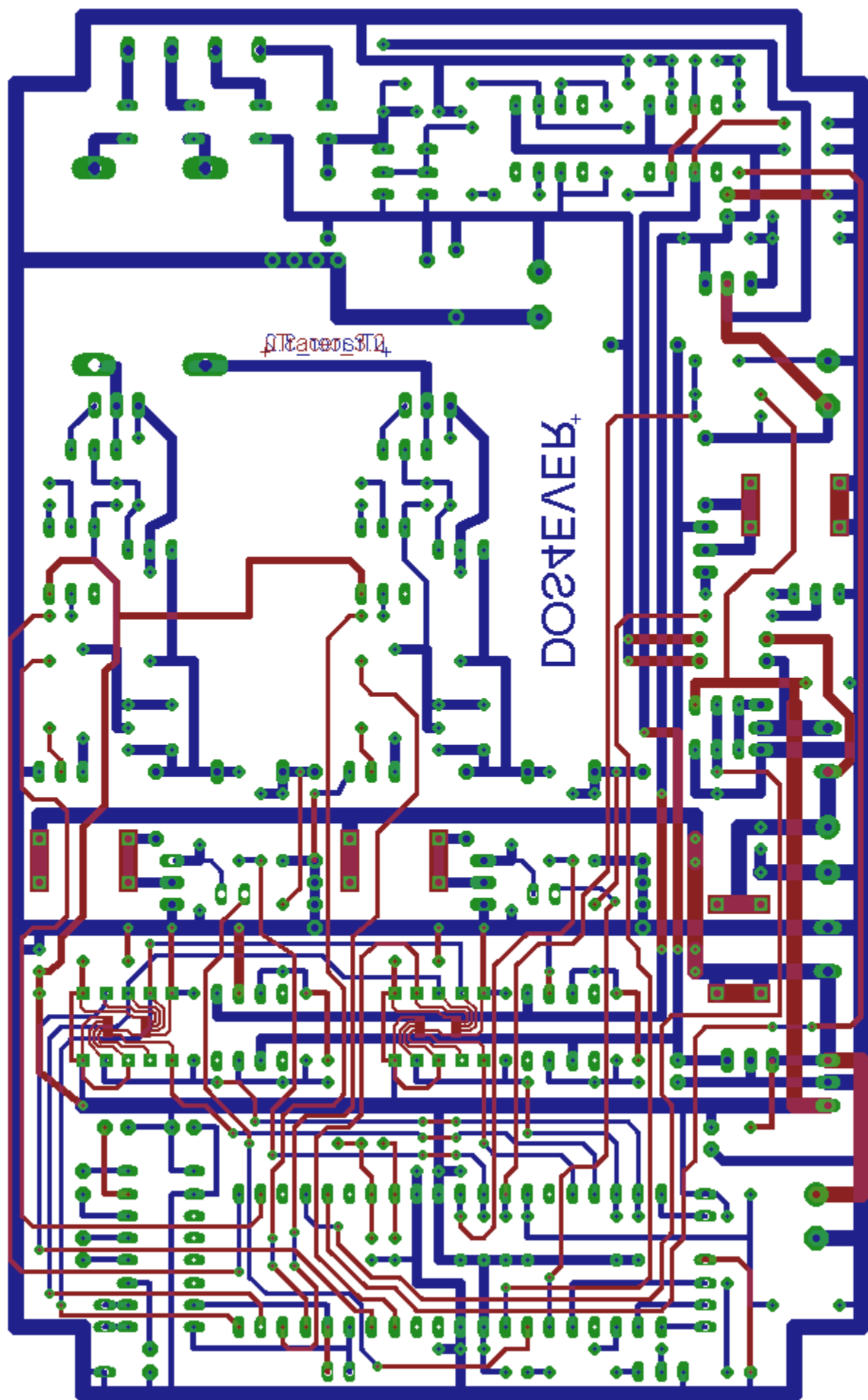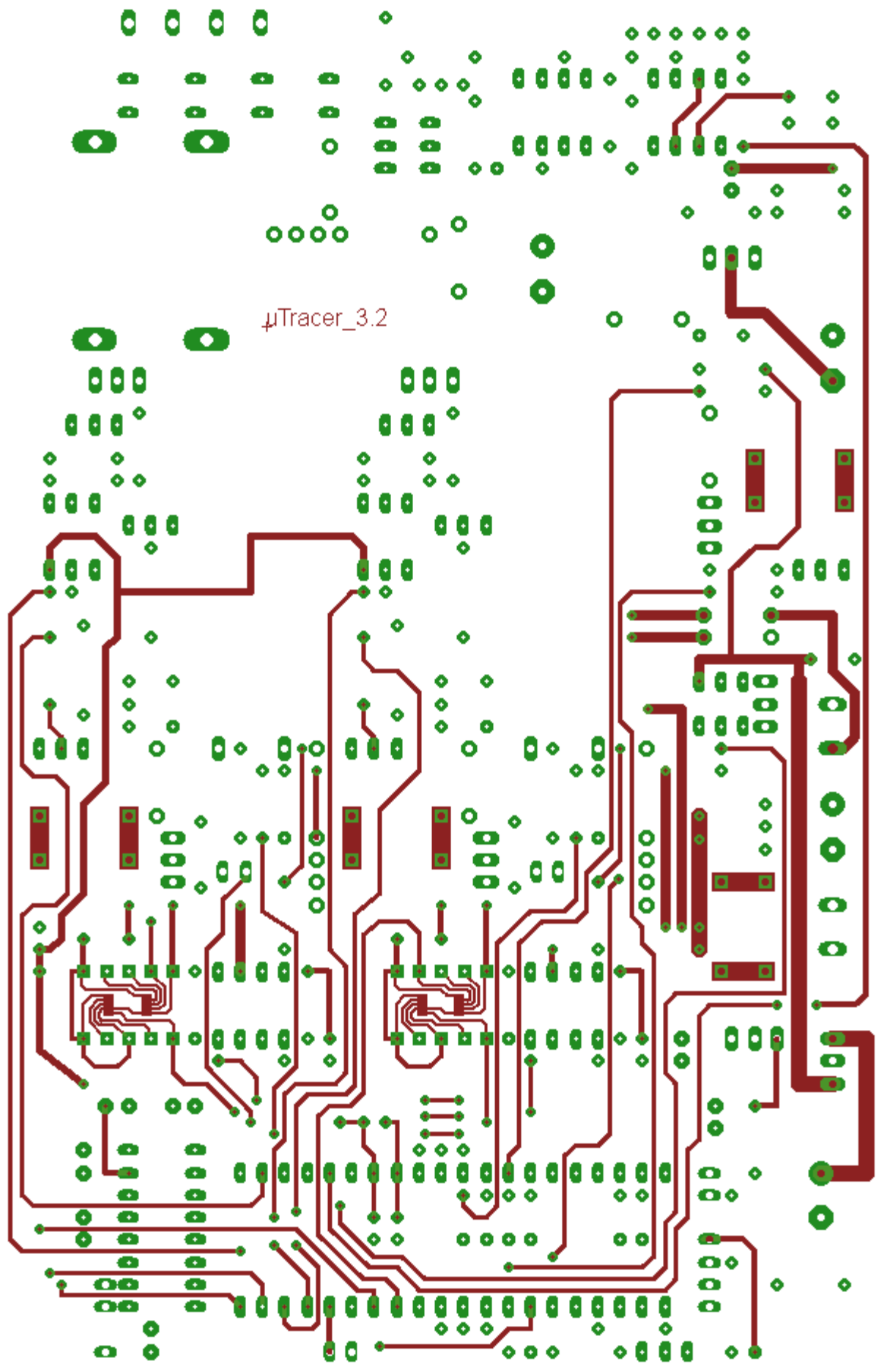6k8

MCLR 1
AN0 2
AN1 3
AN2 4
AN3 5
RA4 6
AN4 7
AN5 8
AN6 9
AN7 10
Vdd 11
Vss 12
CLCI 13
CLCO 14
RC0 15
PWM2 16
PWM1 17
RC3 18
RD0 19
RD1 20

PGD 40
PGC 39
RB5 38
RB4 37
PGM 36
RB2 35
RB1 34
RB0 33
Vdd 32
Vss 31
RD7 30
RD6 29
RD5 28
RD4 27
Rx 26
Tx 25
RC5 24
RC4 23
RD3 22
RD2 21

IC9 16F874

D22 - D25 = BAT86
OC1 and OC2 are the LEDs in the
high voltage switch opto-couplers

D25
D24
D23
D22

+5V
C31
150 nF

33 pF
C30
33 pF
C29
X1 20MHz

R50
180
L2
"Power"
+5V
+5V
C33
150 nF

anode measurement
pulse
OC1
+5V
R46
100
+5V
C32
1 uF

screen measurement
pulse
R48
100
OC2
+5V

+5V
150 nF
C34
10 uF
C24
J4

IC11
C28
1 uF

1 C1+ V+ Vcc 16
3 C1-
14 T1out T1in 11
13 R1in R1out 12
8 R2in R2out 9
7 T2out T2in 10
4 C2+
5 C2- V- Gnd 15
6

C27
1 uF
C26
1 uF
C25 1 uF
MAX232

IC10
7805
in out Gnd
+19.5V
C23
1000 uF
50 V

21 FEB 2016

DOS4EVER

uTracer_3.2

μTracer_3.2