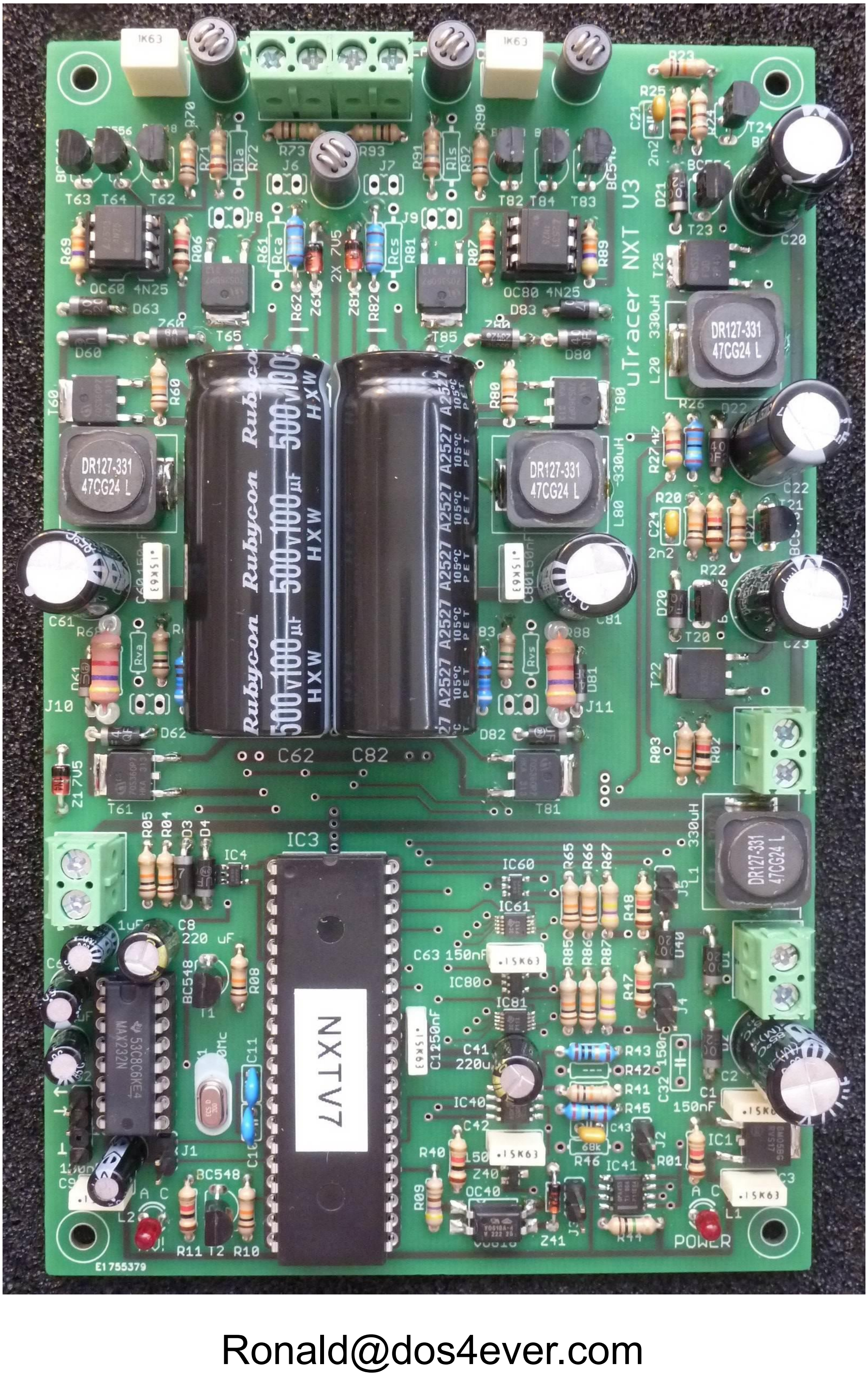
µTracer NXT

Construction Manual V2



Ronald@dos4ever.com

Table of contents:

Warning!

Please read this page carefully!

Although your uTracerNXT 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 100uF / 500V caps are fully discharged
- When you test the circuit on the bench use 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 -120V grid supply is still working !

If you construct and use this kit, it implies that
you have read and understood this warning and accept the risks!

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 am 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 endeavor to help by email whenever 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 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 uTracerNXT circuit.

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

My-µTracer NXT

Configure the µTracerNXT to your specific needs

The uTracerNXT kit as described in this manual is the basic or standard configuration of the device with the following specifications:

- Anode and screen voltage 2 – 500 V
- Anode and screen currents 0 – 350 mA (250 mA with compliance on)
- Grid voltage range 0 to -120 V

However, the circuit, PCB and software have been designed in such a way, that with a few small modifications the uTracerNXT can be adapted to your specific needs.

This will allow you to:

- Reduce the anode and/or screen voltage ranges, e.g. to accurately trace battery tubes.
- Increase the anode and/or screen current ranges up to 1 A per channel for the tracing of high-power tubes
- Decrease the anode/or screen current ranges for improved accuracy for low power and/or battery tubes
- Decrease the grid voltage range for improved resolution at low grid biases.

This manual only covers the construction and calibration of the standard version. A “cookbook” to modify the uTracerNXT to your specific needs can be found online: www.dos4ever.com/My-uTracerNXT.html

Your kit contains a bag with resistors labeled “My-uTracerNXT.” These resistors have been added to get you started changing the ranges of your uTracerNXT to some obvious configurations. Contents of the bag:

value	qty	purpose
68 kΩ	1	0 to -25 V control grid bias range
12.4 Ω	2	Current sense resistor 750 mA anode/screen current range
1.8 Ω	2	Current limit resistor 750 mA anode/screen current range
7.5 Ω	2	Current sense resistor 1 A anode/screen current range
1.0 Ω	2	Current limit resistor 1 A anode/screen current range
240 kΩ	2	2 – 100 V anode/screen voltage range

Before you begin

1. To build the µTracerNXT 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 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 uTracerNXT 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 in case of an accidental short circuit will be limited.
6. The uTracerNXT 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
7. Although I have tried to use as much as possible 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 would be grateful if any errors and / or omissions found were reported to me at: Ronald@dos4ever.com.

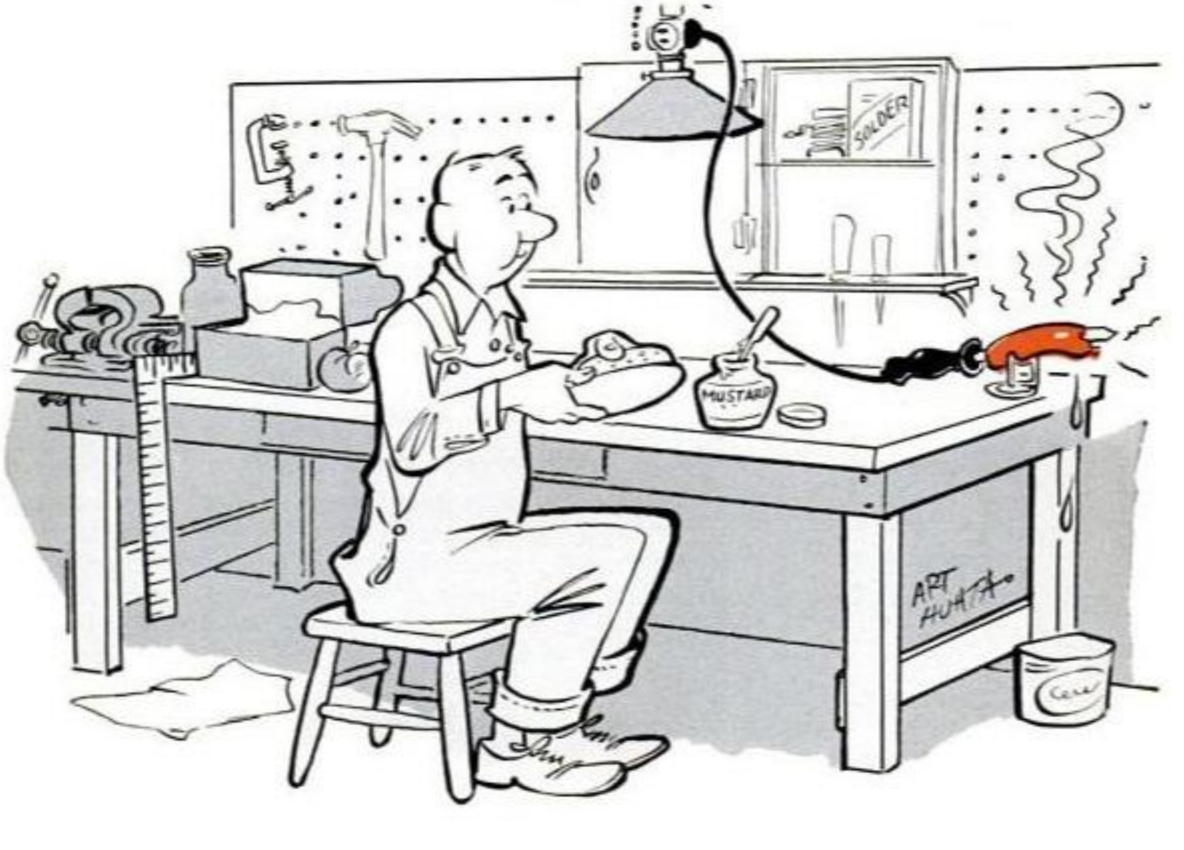
Please note that the manual version number does not have any relation with the PCB version number.

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 screwdriver.

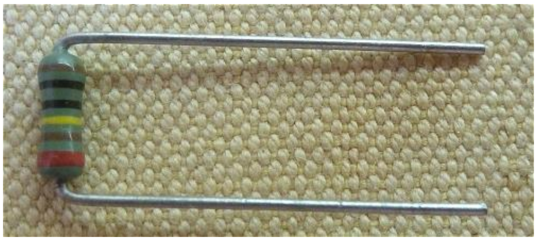
Things that come in handy:

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);
7. A magnifying glass!

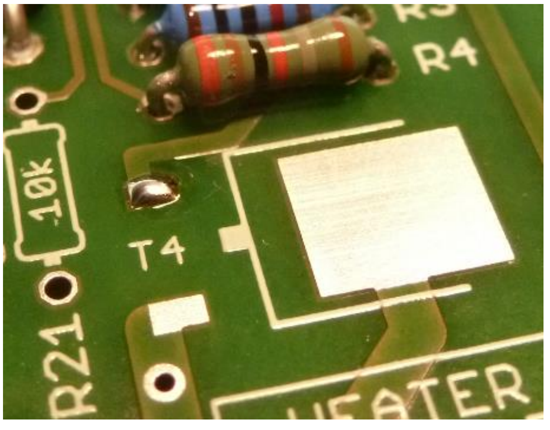


Tips and Tricks:

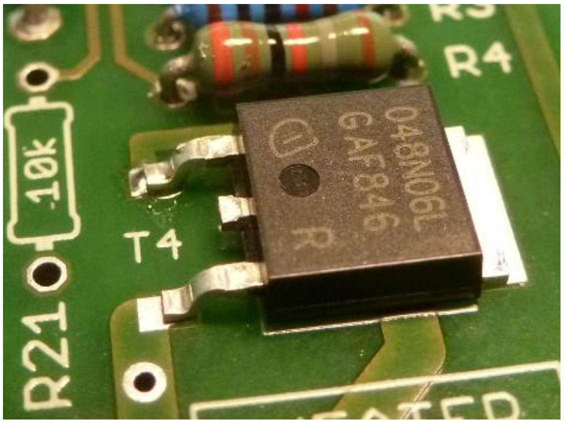
1. Bend the leads of the resistors in a sharp angle close to the body of the resistor.



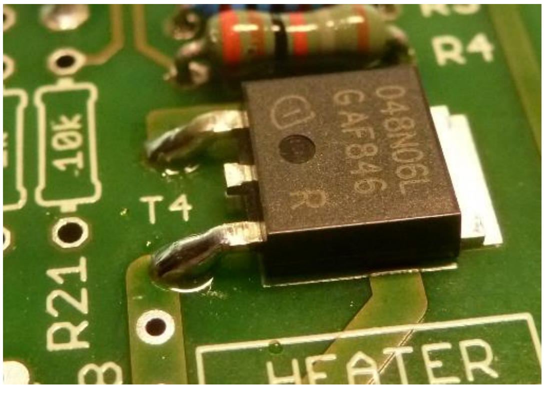
2. This kit contains a number of SMD type transistors. These transistors are easy to mount on the PCB and have the great advantage that they can also be easily replaced when needed. Although not absolutely necessary, some soldering flux either in a bottle or in pen is recommended. This is my preferred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



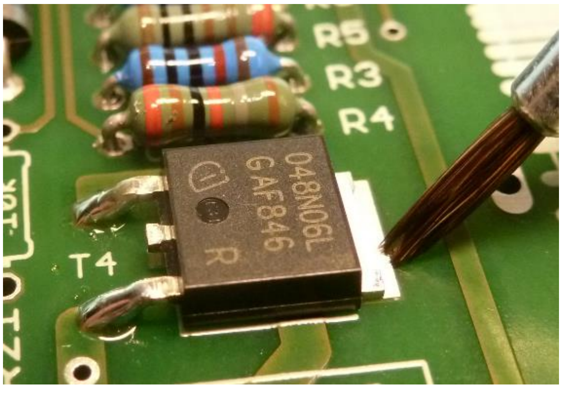
Apply solder to one pad



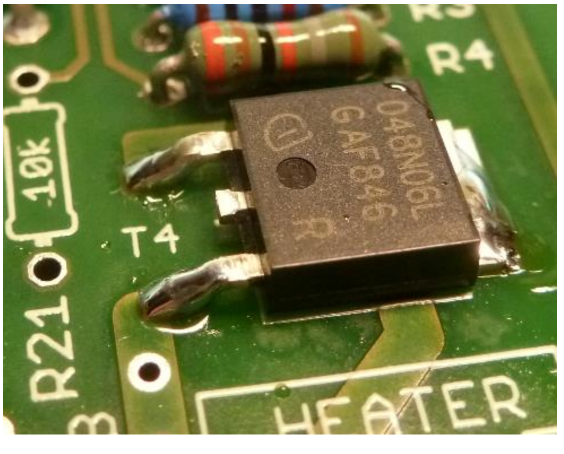
Fix one leg of the transistor and adjust position



Solder the other small terminal

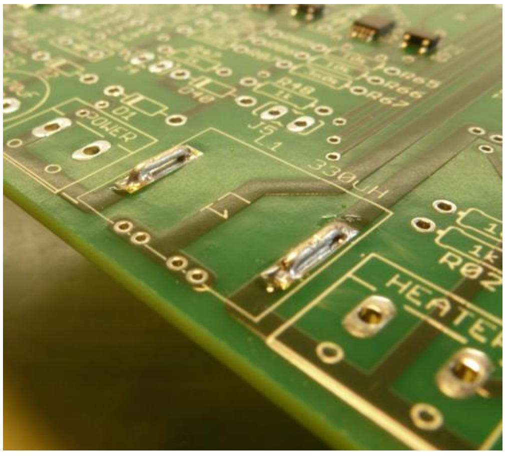


Apply some solder flux to the tab



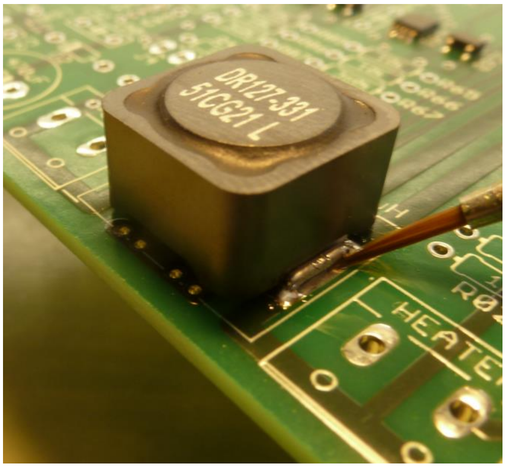
Heat the tab with your soldering iron and add a bit of solder. When hot enough the solder will ready flow between the tab and PCB.

3.

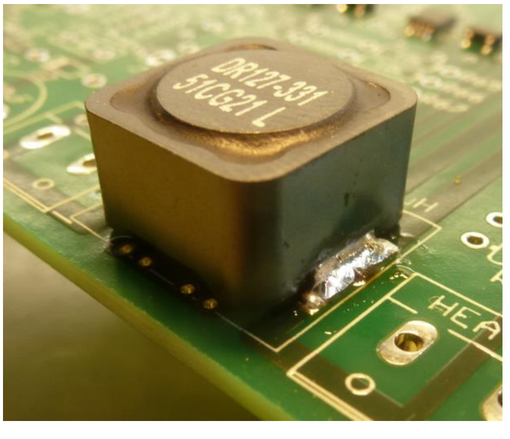


For the mounting of the 330 uH inductors L1, L20, L60, L80 I recommend the procedure depicted on the left.

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



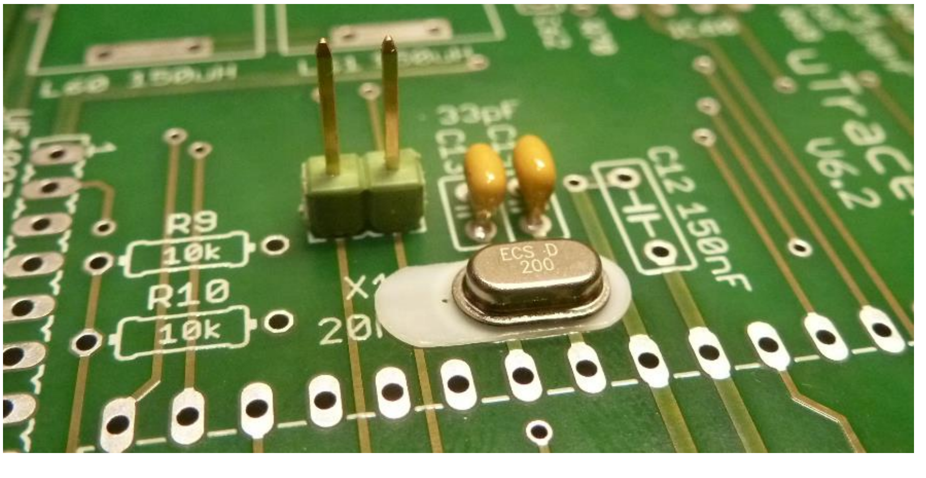
Position the inductors between the wires with the contacts facing the wires. Apply a small amount of soldering flux when available (middle).



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

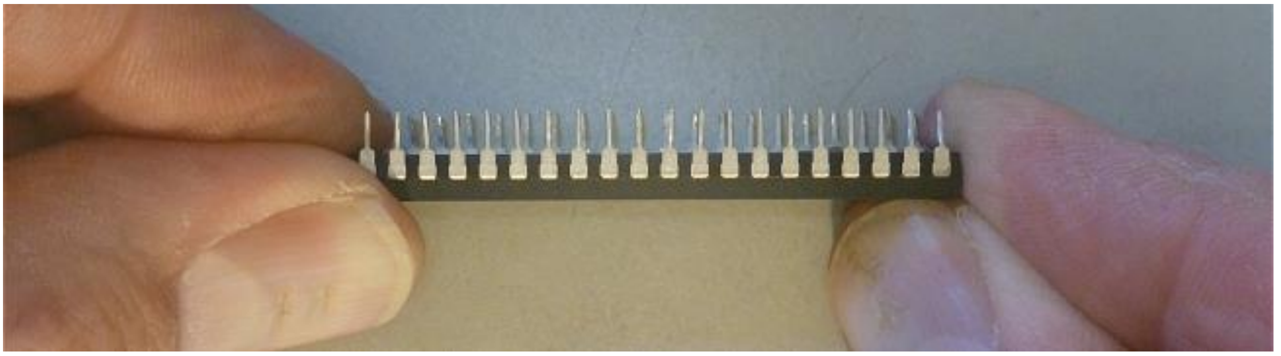
Turn the board around and make sure the solder has flown all the way through the holes of the contact pads.

4.

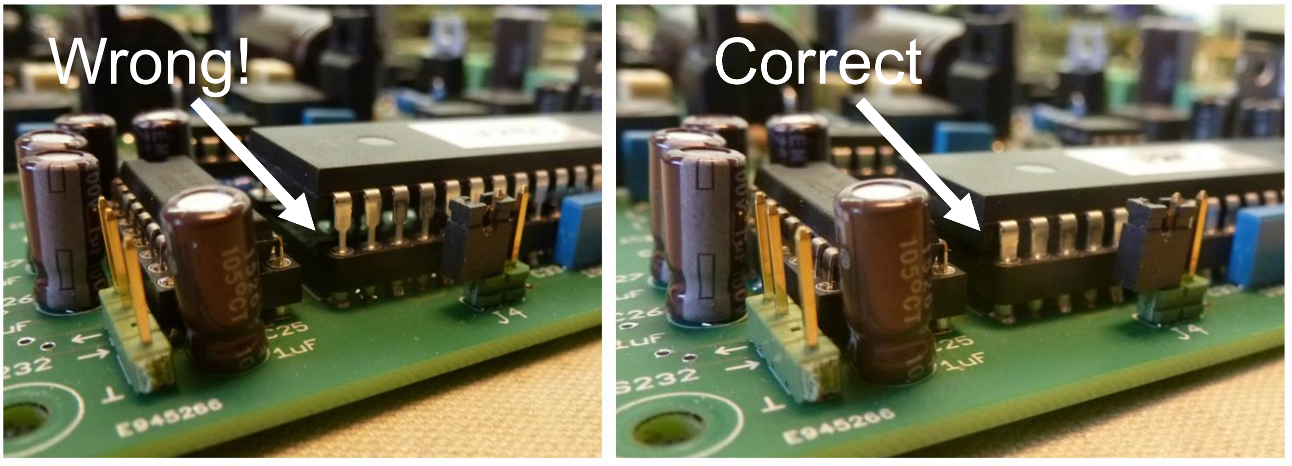


Use the small Teflon spacer provided to isolate the Xtal from the PCB.
(photo taken from uTracer6)

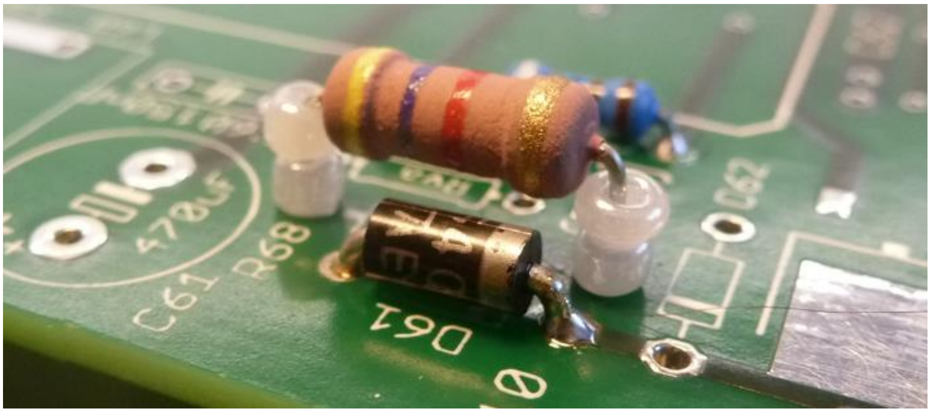
5 As usual, the pins of the PIC processor are at an angle so that the PIC doesn't fit directly in the 40 pin socket. The best way to adjust the angle of the pins is to hold the processor at the sides and correct the angle of 20 pins at once by bending them against a flat surface e.g. the table.



6. 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+)

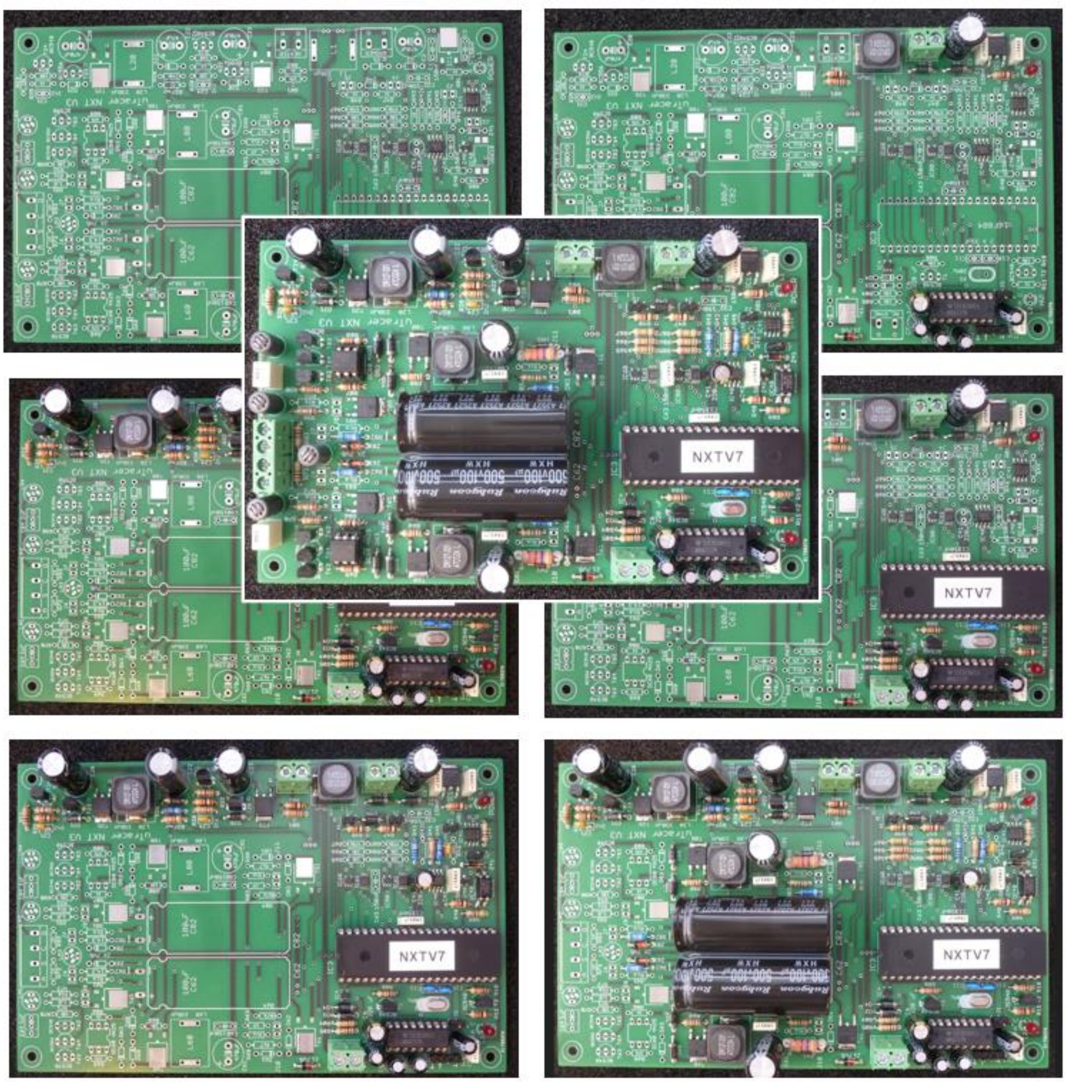


7. Resistors R68 and R88 can become quite warm. It is therefore advised to add some space between the resistor and the PCB by inserting two of the ceramic beads that have been supplied with the kit per side.



8. Trick to bend the leads of the reservoir capacitors into a nice 90-degree angle.

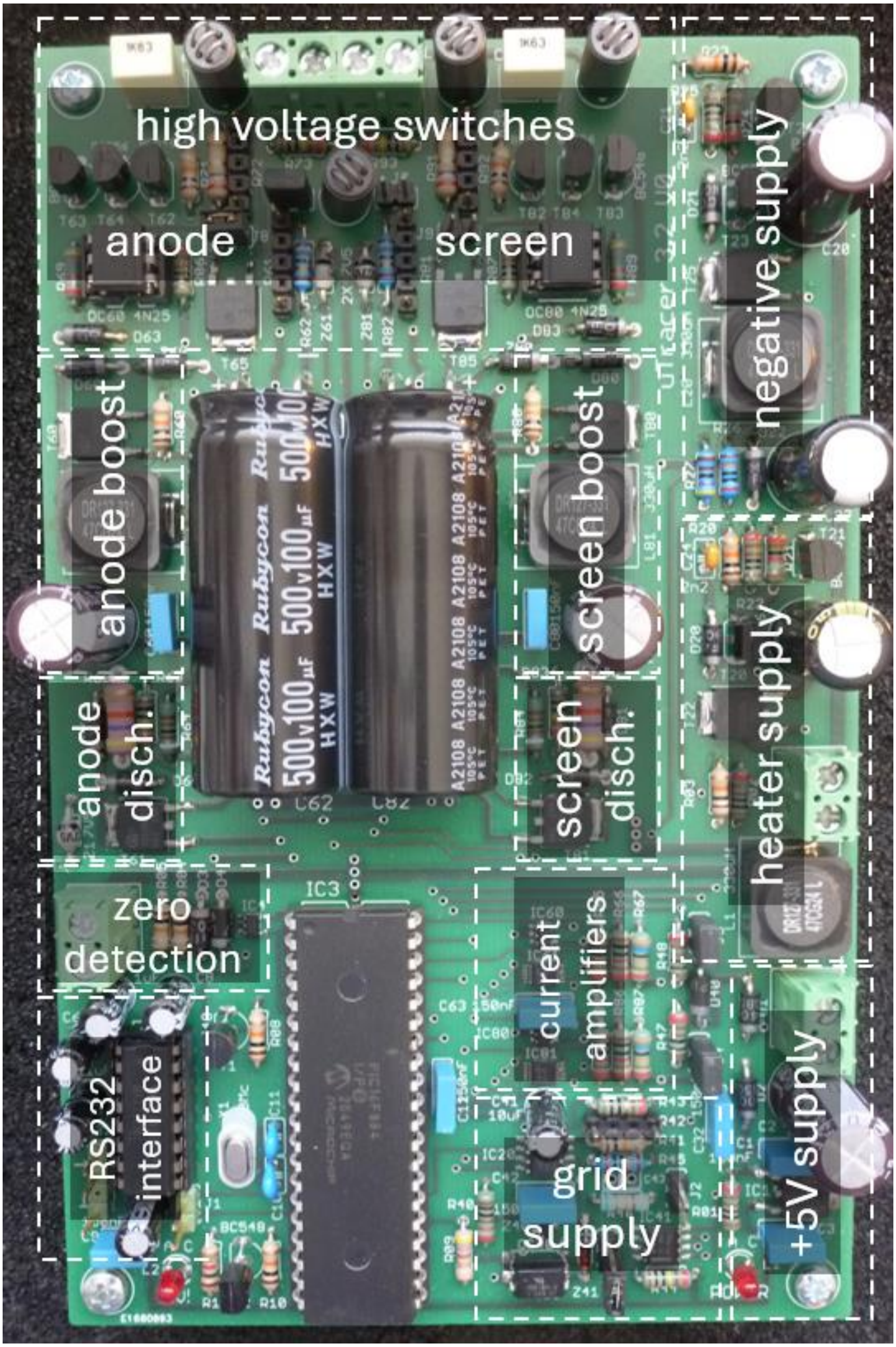
9. For your convenience, a zip file containing a series of high-resolution photographs taken after each construction stage can be downloaded from: https://www.dos4ever.com/uTracerlogNXT/uTracerNXT-pics.zip



Note that the colour and appearance for components may vary depending on availability!

Finding your way around on the board

The picture below shows the approximate location of the different functional parts of the circuit on the PCB.



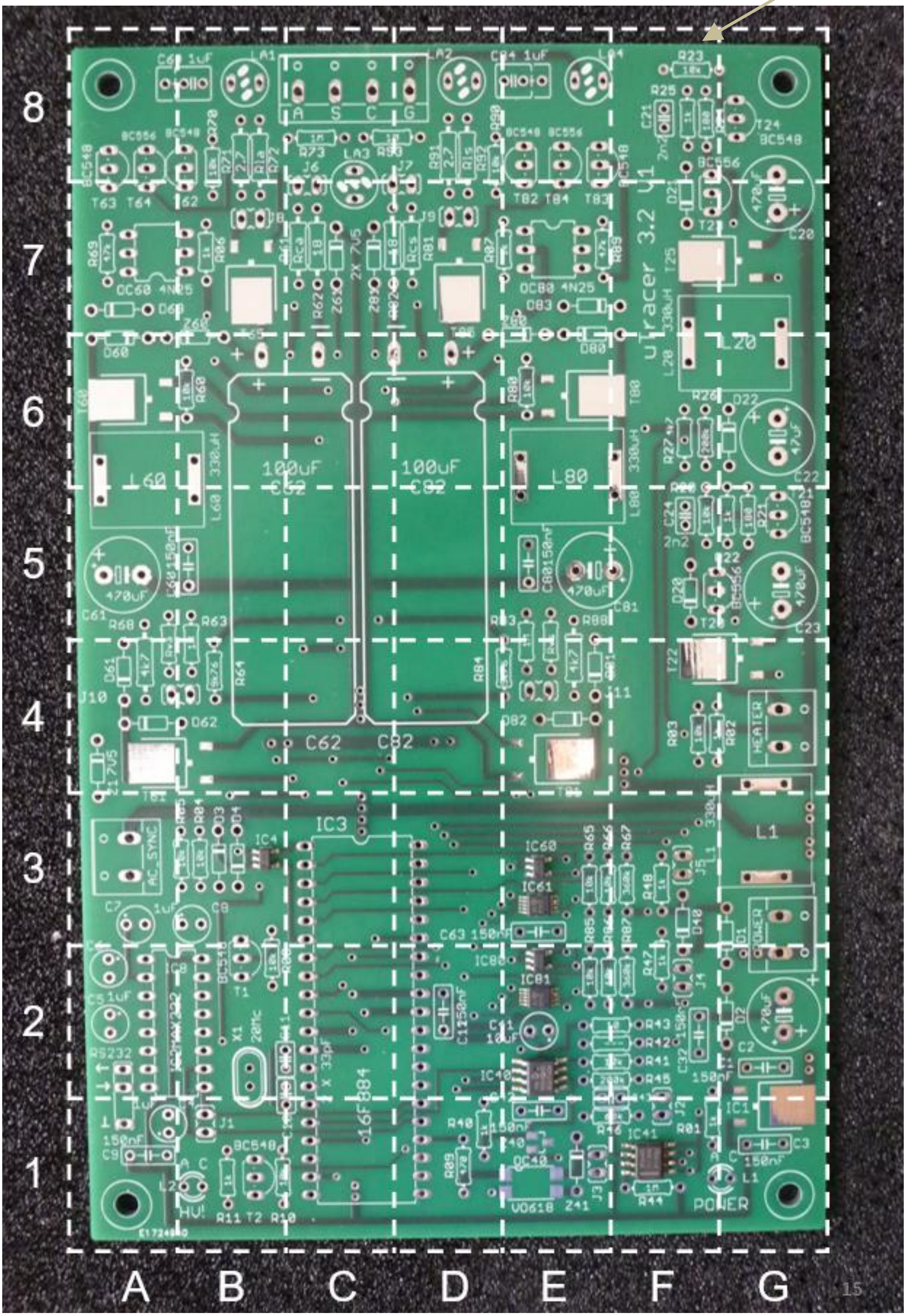
To help you find during the construction the location of a certain component on the PCB, every component to be placed is provided with a grid coordinate where it can be found on the PCB. An example is given on the next page.

Component identifier in schematic

Component type and value

Approximate location on PCB

☐	1	R23	10 kohm	(F,8)



Step-by-step construction of the uTracerNXT

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

The uTracerNXT construction manual consists of the following parts:

1. COM port installation;
2. The Graphical User Interface (GUI);
3. GUI testing;
4. +5V power supply;
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.
12. Calibration

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

In contrast to the uTracer3 the calibration is done after the construction has been completed. 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 following SMD components have already been assembled on the PCB:
.

Grid section:
Z40 LM4040-2.5V
IC40 MCP4921
IC41 OPA455

Anode & Screen section:
IC60,IC80 MCP6V86U
IC61,IC81 PGA113

General:
IC4 MCP6V86U

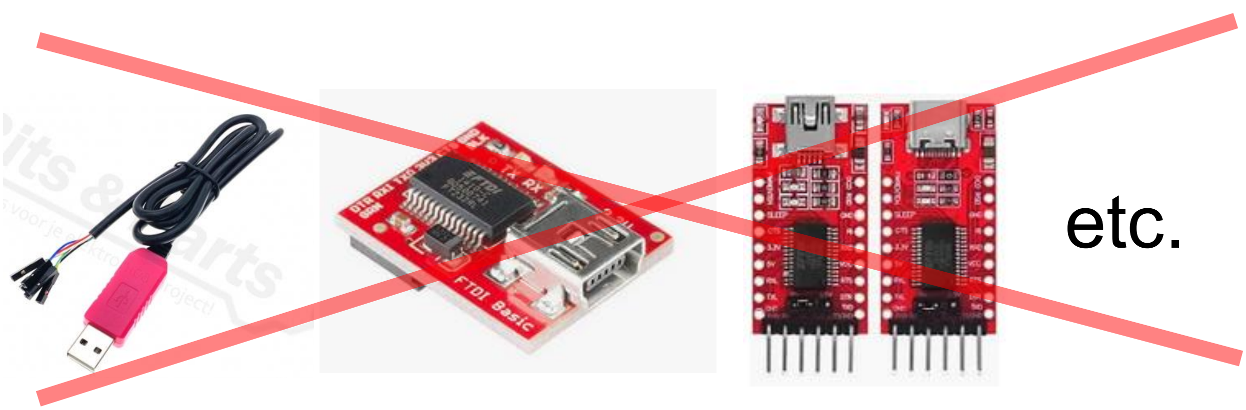
Part 1. COM port installation (if needed)

The GUI communicates to the uTracerNXT 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 number assigned to the adapter



Part 2. The graphical user interface (GUI) installation

In order to install the GUI for the uTracerNXT, 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 uTracerNXT 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:
https://www.dos4ever.com/uTracer3/uTracer3_pag12.html

"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)

To use the uTracerNXT GUI, it is necessary to first install the GUI version 3p11 for the uTracer3. You will not be using it, but the uTracerNXT 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 store the zipped file with the GUI into.
- ☐ 2 Download the zip file with the uTracer3 GUI into the folder from: https://www.dos4ever.com/uTracerlogNXT/uTracer3p11-GUI.zip
- ☐ 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 uTracerNXT GUI – installation

To be able to use the uTracerNXT GUI, it is necessary to first install the GUI version 3p11 for the uTracer3. You will not be using it, but the uTracerNXT GUI will need some software components that are installed during the uTracer3 version 3p11 installation.
If you do not have the uTracer3 GUI version 3p11 installed goto part 2a, otherwise proceed with this part.

Installation of the uTracerNXT GUI does not affect the installation of the uTracer3 GUI.

Installation of the uTracerNXT GUI:

- ☐ 1 Create a folder to download the zipped file with the GUI into.
- ☐ 2 Download the GUI for the uTracerNXT into the folder from: https://www.dos4ever.com/uTracerlogNXT/uTracerNXT-GUI.zip
- ☐ 3 Unzip the downloaded file into that folder.
- ☐ 4 Double click the executable.

Important!

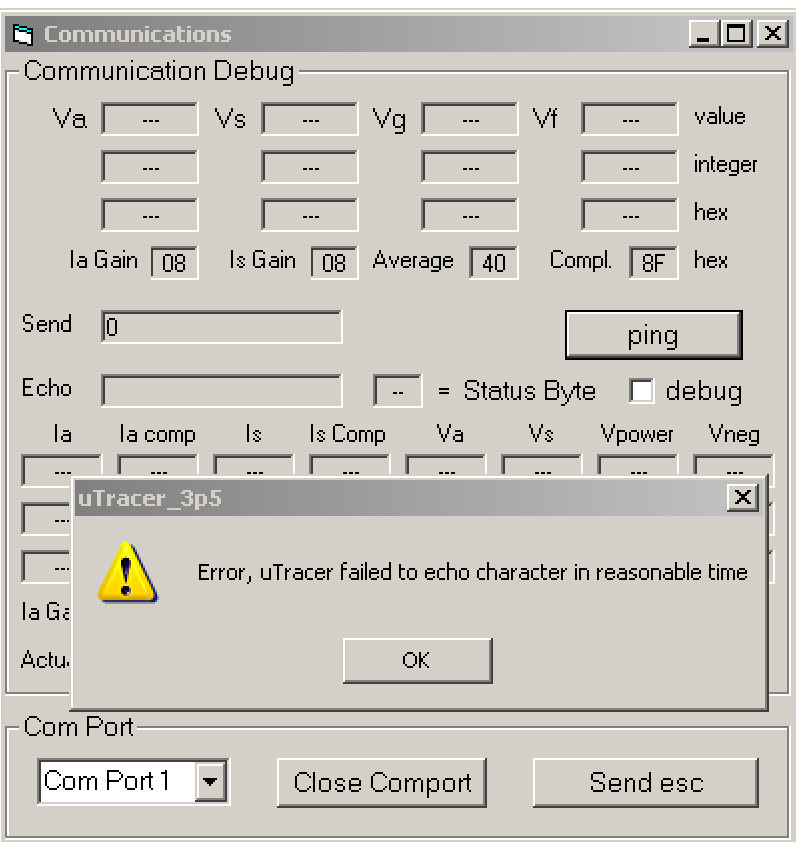
Depending on your computer configuration it is possible that Windows Defender regards the executable as malware and automatically deletes it! If this happens you must add an "exclusion" before it will let you copy the executable without automatically deleting it.

Read more about adding an exception:
https://support.microsoft.com/en-ca/help/4028485/windows-10-add-an-exclusion-to-windows-security

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 "save to calibration file" in the "Calibration" form.
- ☐ 5 Press the "ping" command button.
- ☐ 6 The "Send string" and "Echo string" should now display:

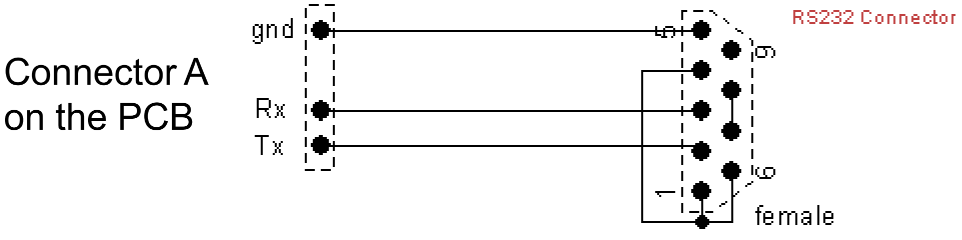


Explanation:
the GUI tries to send a "0" but this zero is not echoed. After a few seconds the GUI detects that no character is returned resulting in the timeout message.

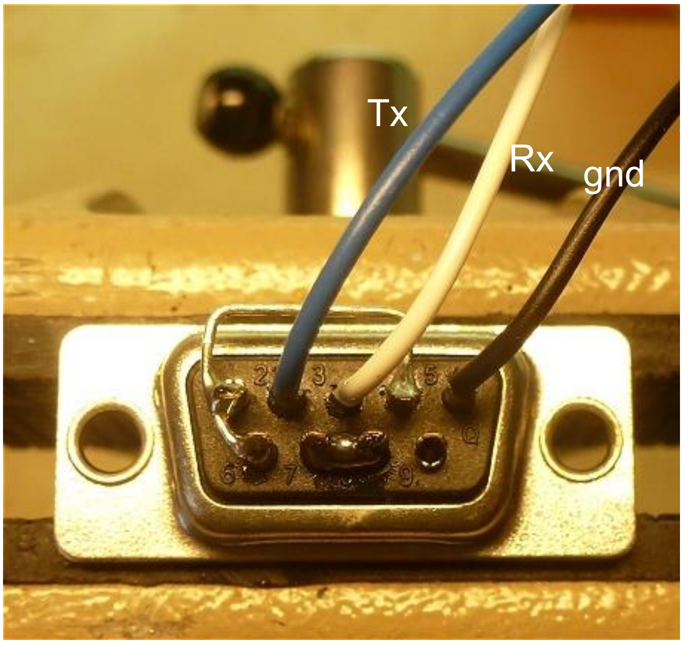
- ☐ 7 Under no circumstance the message "Run time error 8020, Error reading comm device" should occur. When it does, there is a problem with the USB-to-serial adapter.
- ☐ 8 For a full check, connect the RS232 cable (next section) and perform the test described in that section.

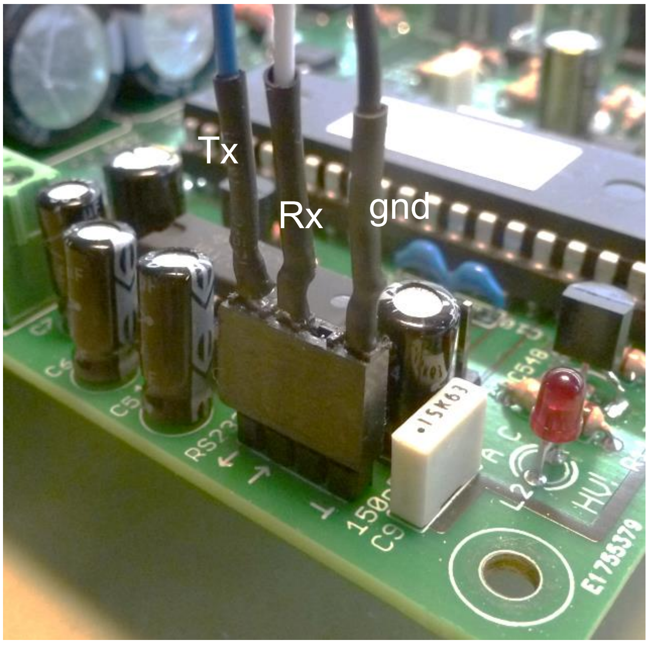
Part 3. The RS232 connector cable - construction

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.



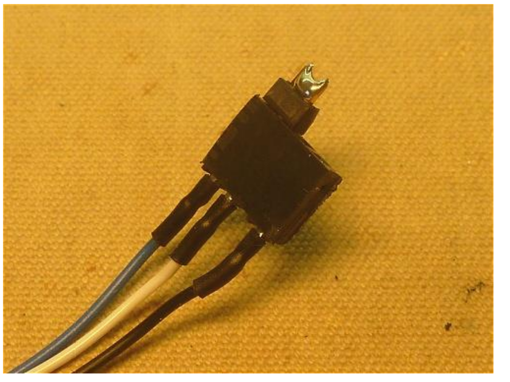
On the D-SUB connector pins 1,4,6 and pins 7,8 have been connected to "fool" the handshake signals. Strictly speaking this should not be necessary because the GUI does not check for these signals, but I am never sure what USB to serial adapters do?



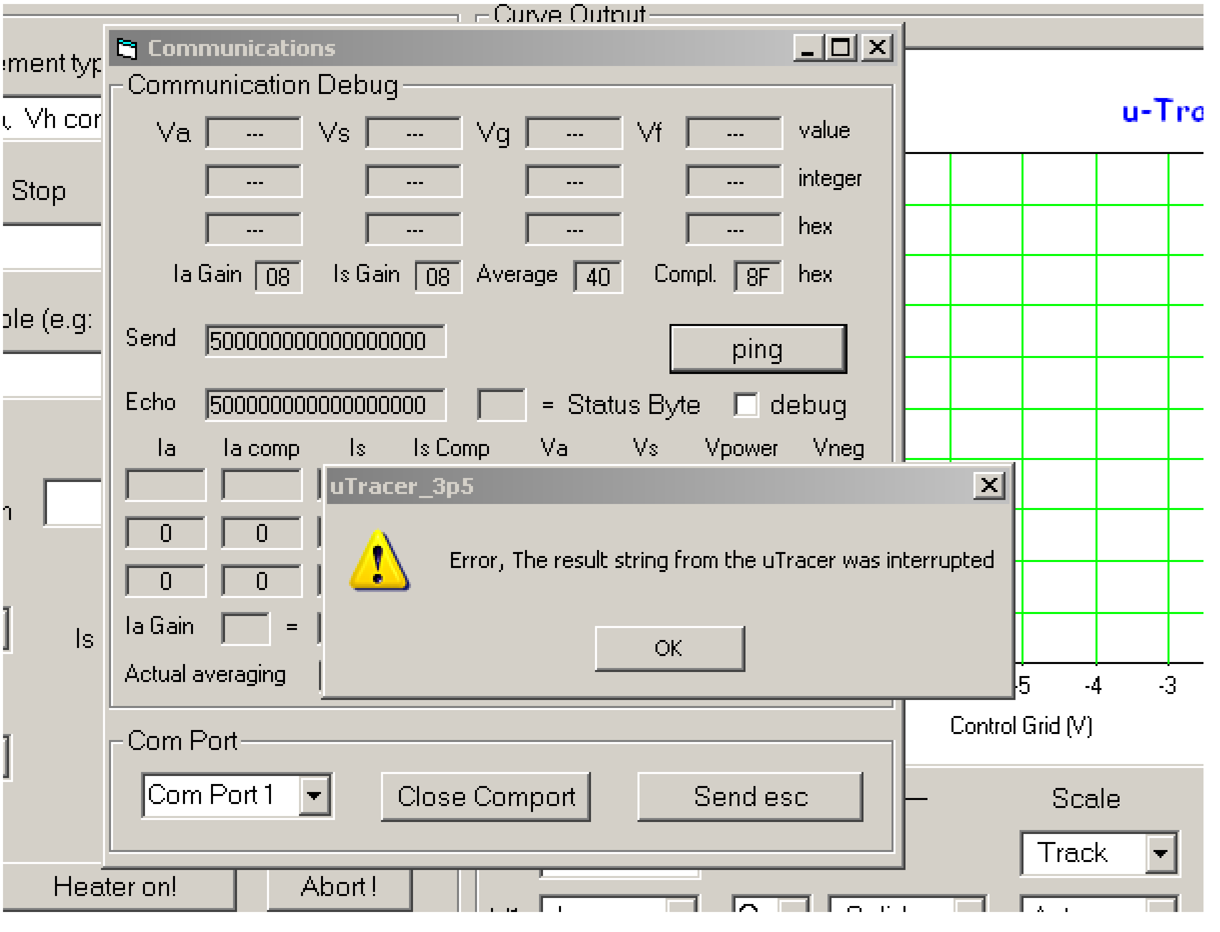


Part 3. The RS232 connector cable – testing

- ☐ 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.



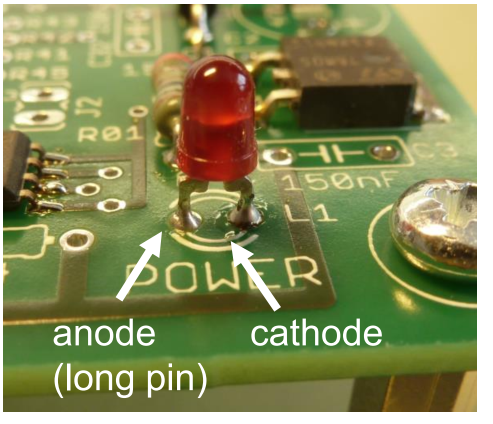
- ☐ 4 Start the GUI.
- ☐ 5 Open the "Debug/Communications" window by pressing "Debug" in the "Miscellaneous" section of the GUI
- ☐ 6 Press the "ping" command button.
- ☐ 7 The "Send string" and "Echo string" should now display:



- ☐ 8 After approximately 10sec the error message: "The result string from the uTracerNXT was interrupted" should appear . This is the proper response at this point of the construction.

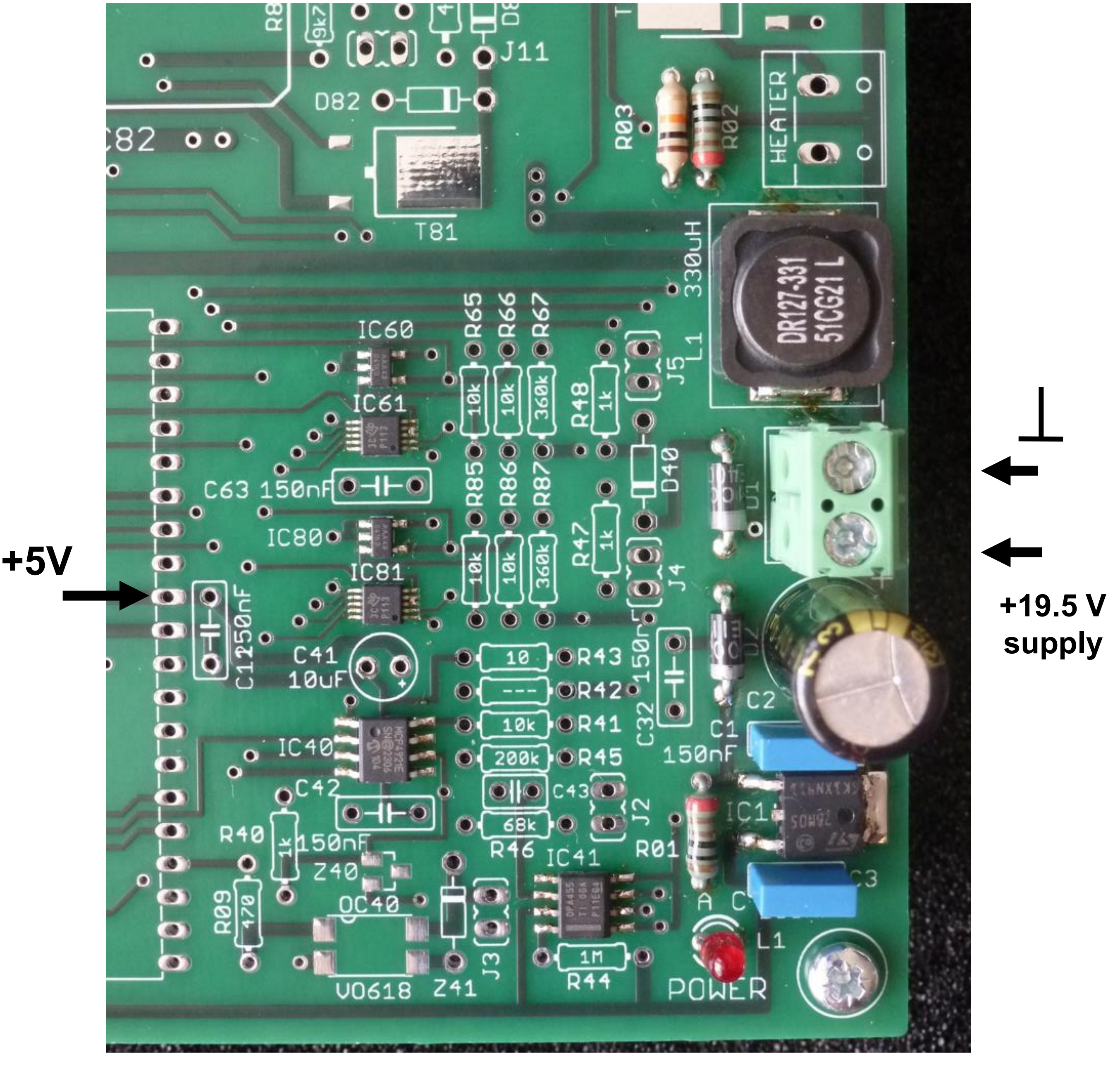
Part 4. +5V power supply - construction

☐	1	L1	330uH inductor, **see "Tips and Tricks no 3"**	(G,3)
☐	2	D1	UF4007, note proper polarity!	(G,3)
☐	3	D2	UF4007, note proper polarity	(G,2)
☐	4		"Power" screw connector	(G,3)
☐	5	IC1	78M05 +5V voltage regulator, see "Tips and Tricks no 2"	(G,1)
☐	6	R01	1 kohm	(F,1)
☐	7	R02	1 kohm	(G,4)
☐	8	R03	10 kohm	(F,4)
☐	9	L1	LED (power on)	(G,1)
☐	10	C1	150 nF	(G,2)
☐	11	C3	150 nF	(G,1)
☐	12	C2	470 uF, 35 V, note polarity!	(G,2)
☐	13	Z1	7.5 V Zener diode (1N4737), note polarity!	(A,4)



Part 4. +5V power supply - testing

- ☐ 1 Connect the uTracerNXT to a power supply ca. 19.5V, if possible limit the current to ca. 100 mA.
- ☐ 2 Switch the power supply on (supply current ≈ 25 mA @ 19 V).
- ☐ 3 Check LED L1 is on
- ☐ 4 Check + 5 V on pin 32 of the PIC (see picture below)
  Make sure not to slip with the measurement probe!

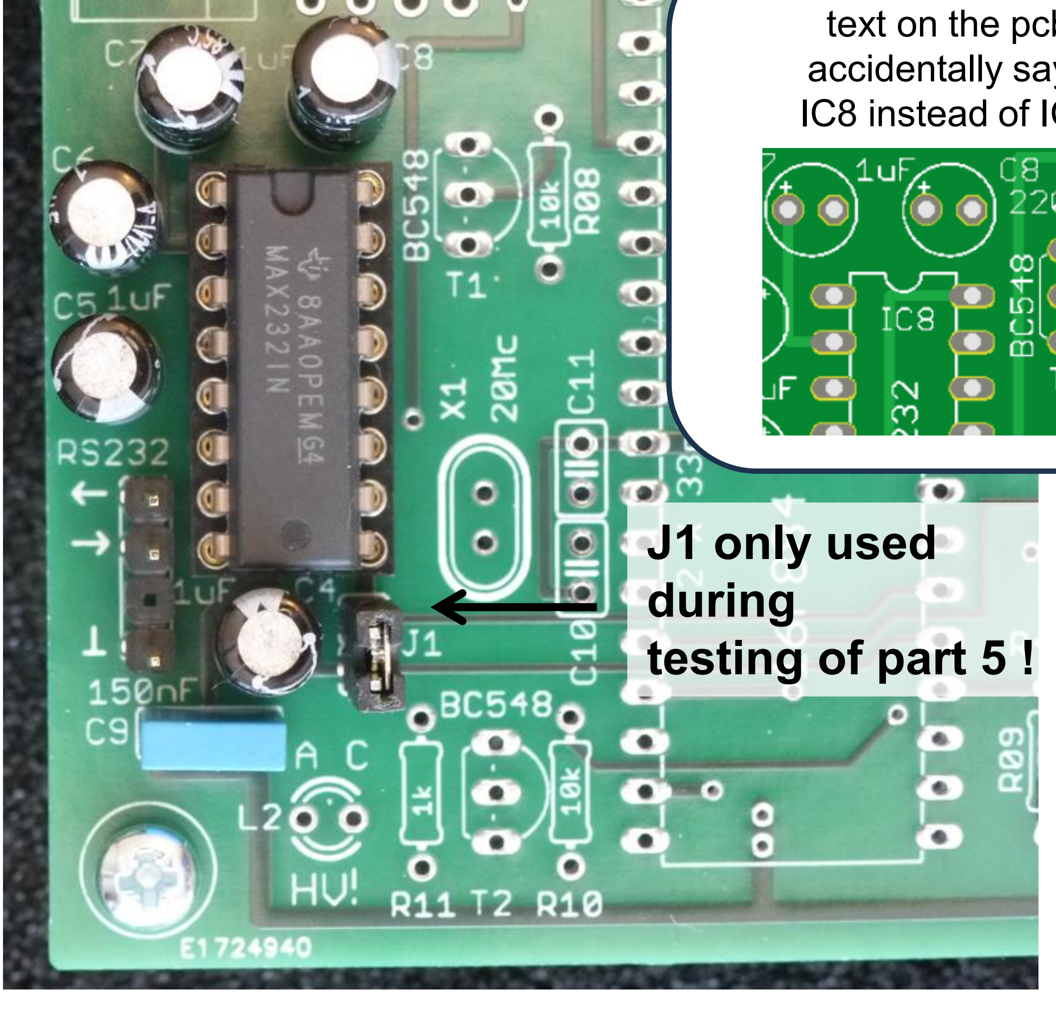


Overview of the PCB after completion of Part 4 of the construction, and the location of the test points. Note that Zener diode Z1 is not visible on this photograph.

Part 5. RS232 Interface - construction

☐	1		16 pin IC socket for IC2, note orientation ! (see note)	(A,2)
☐	2	IC2	Insert IC2, MAX232, note orientation ! (see note)	(A,2)
☐	3		3 pin serial interface male pin header	(A,2)
☐	4	C9	150 nF	(A,1)
☐	5	J1	2 pin male jumper pin header	(B,1)
☐	6	C4	1uF / 50V, note polarity !	(A,1)
☐	7	C5	1uF / 50V, note polarity !	(A,2)
☐	8	C6	1uF / 50V, note polarity !	(A,2)
☐	9	C7	1uF / 50V, note polarity !	(A,3)
☐	10	C8	220uF / 10V, note polarity !	(B,3)

Note, the 1uF capacitors look very similar to the 220uF capacitors, do not confuse them!

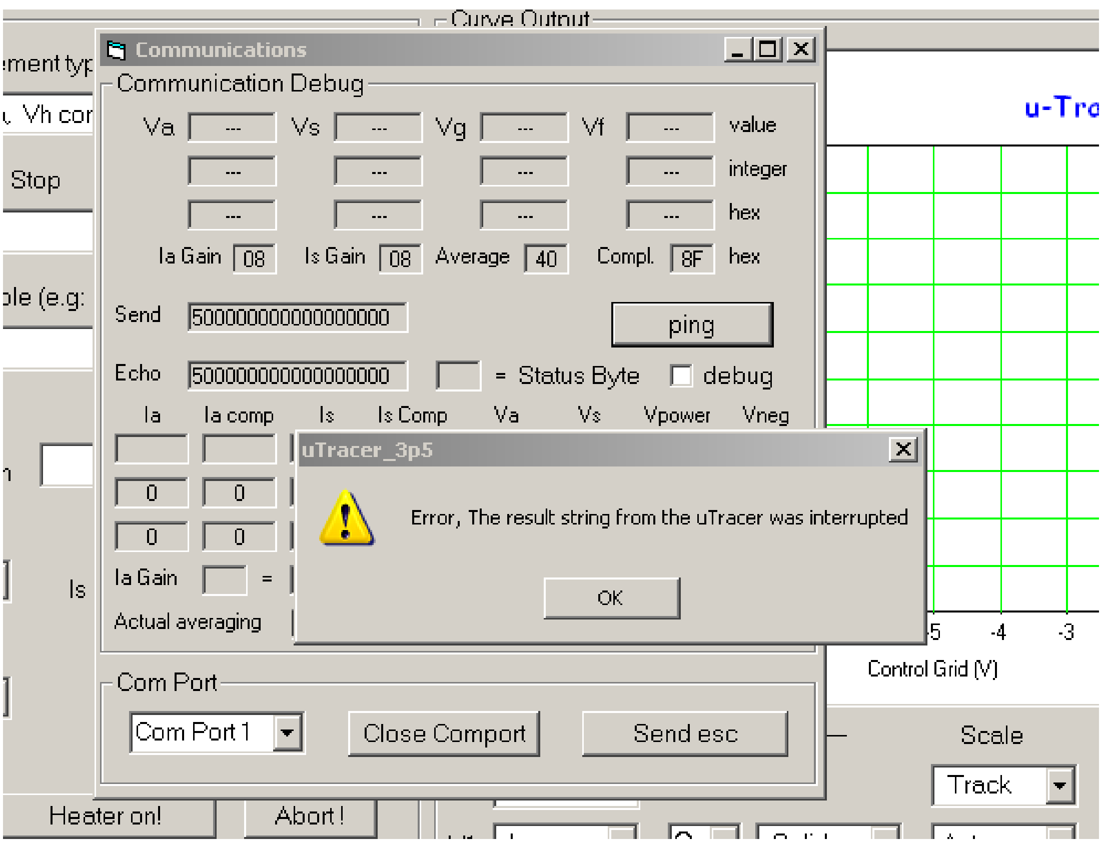


please note that the text on the pcb accidentally says IC8 instead of IC2

The RS232 interface after completion of Part 5.

Part 5. RS232 Interface - testing

- ☐ 1 Make sure that the GUI is properly installed and tested.
- ☐ 2 Connect the uTracerNXT to the PC
- ☐ 3 Connect the uTracerNXT to the power supply, if possible, set current limit to 100 mA .
- ☐ 4 Make sure jumper J1 is in place.
- ☐ 5 Start the GUI
- ☐ 6 Switch on the power supply (supply current ≈ 30 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:



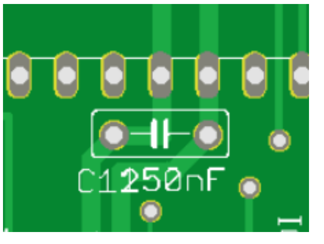
- ☐ 10 After approximately 10 sec the error message: “The result string from the uTracerNXT was interrupted” should appear . This is the proper response.
- ☐ 11 **<u>Be sure to remove jumper J1 !!</u>** ⬅ !

Part 6. Micro-controller - construction

☐	1	X1	20 MHz Xtal (see note below)	(B,2)
☐	2	C10	33 pF	(B,1)
☐	3	C11	33 pF	(B,2)
☐	4		40 pin IC socket for IC3, note orientation !	(C,3)
☐	5	C12	150 nF	(D,2)
☐	6	R04	10 kohm	(B,3)
☐	7	R05	10 kohm	(B,3)
☐	8	R08	10 kohm	(B,2)
☐	9	R10	10 kohm	(B,1)
☐	10	R11	1 kohm	(B,1)
☐	11	D3	UF4007, note orientation !	(B,3)
☐	12	D4	UF4007, note orientation !	(B,3)
☐	13		"AC_sync" screw connector	(A,3)
☐	14	L2	LED "High voltage on" Note orientation !	(B,1)
☐	15	T1	BC548, note orientation !	(B,2)
☐	16	T2	BC548, note orientation !	(B,1)
☐	17	IC3	PIC16F884 note orientation ! see "Tips and Tricks" no. 5 and 6	(C,3)

C10, C11: Make sure the central leads of the capacitors do not short circuit

C12: please note that the texts "C12" and "150 nF" accidentally overlap

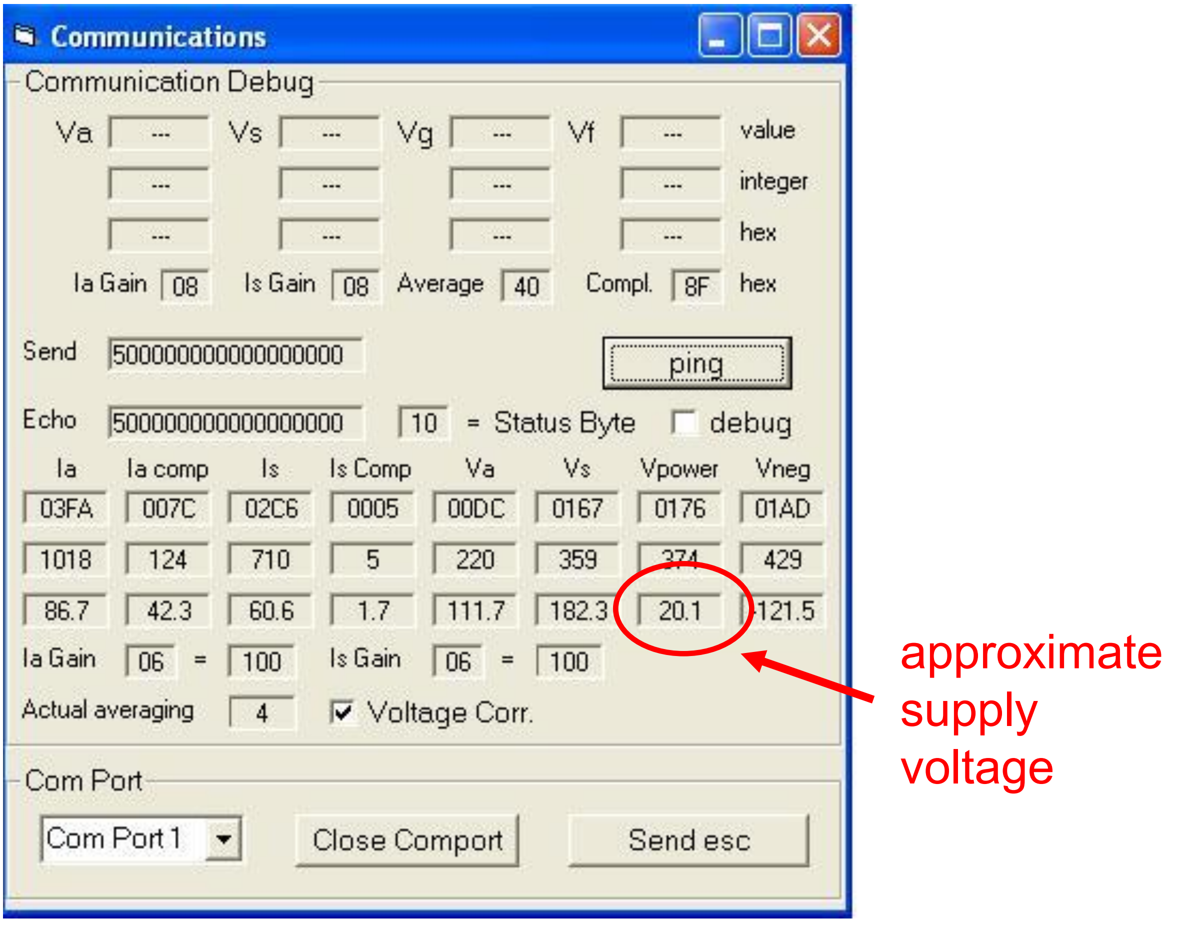


Note: The Xtal is located on the conductive foam.
Insert the plastic isolator between Xtal and PCB, see "Tips and Tricks" no 4 !

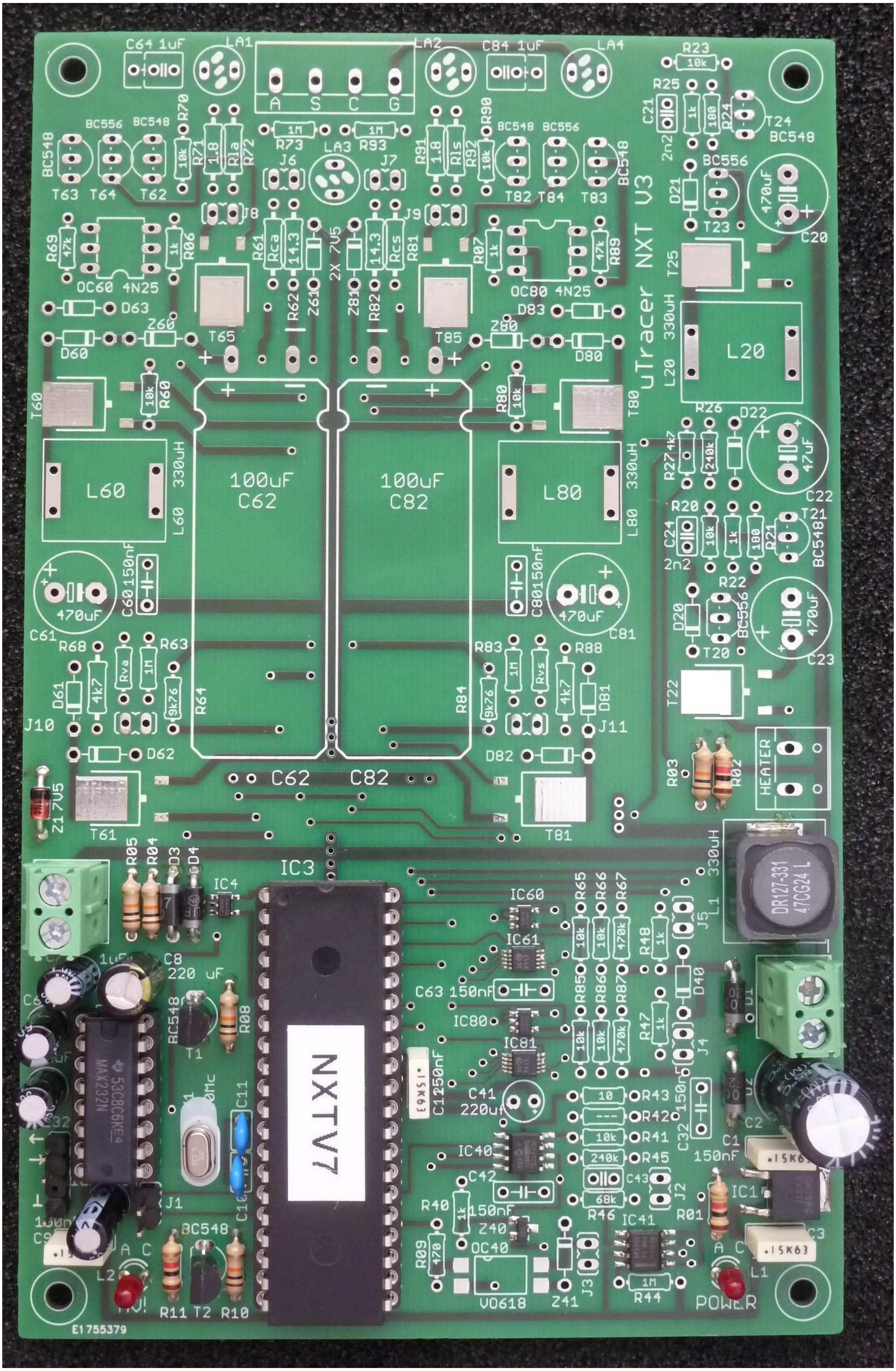
Part 6. Micro-controller - testing

Note: Both LEDs will be on during this test.

- ☐ 1 Make sure that the GUI is properly installed and tested.
- ☐ 2 Connect the uTracerNXT to the PC
- ☐ 3 Connect the uTracerNXT to the power supply, if possible, set current limit to 100 mA .
- ☐ 4 Make sure jumper J1 is removed !
- ☐ 5 Start the GUI
- ☐ 6 Switch on the power supply (supply current ≈ 55 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:



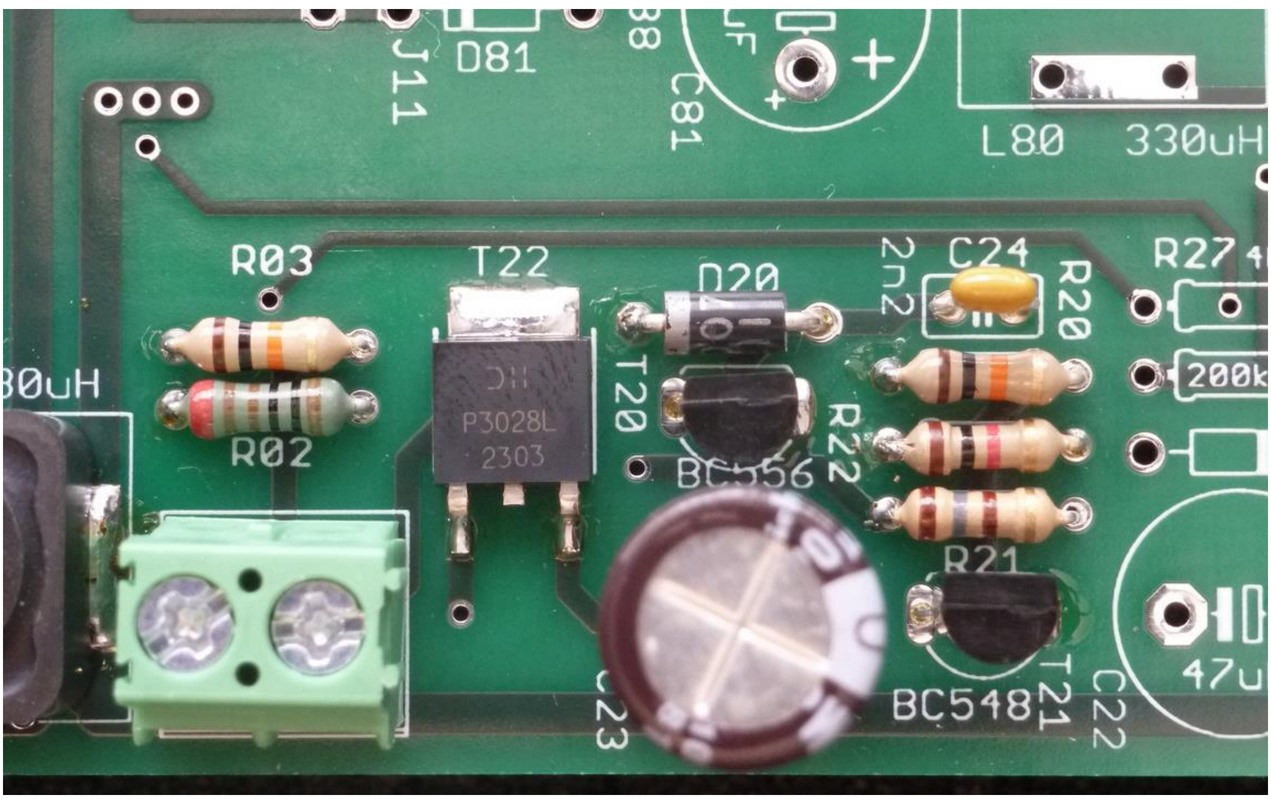
- ☐ 10 No error message should appear !
- ☐ 11 The value in the circle is the approximate supply voltage; all other fields contain random values at this stage.



Overview of the PCB after completion of Part 6.

Part 7. Heater supply - construction

☐	1	R20	10 kohm	(F,5)
☐	2	R21	180 ohm	(G,5)
☐	3	R22	1 k	(G,5)
☐	4	D20	UF4007, note orientation !	(F,5)
☐	5	C24	2.2 nF	(F,5)
☐	6	T22	PMOS transistor DMP3028 **see Tips and Tricks no 2**	(F,4)
☐	7	T20	BC556, note orientation !	(F,5)
☐	8	T21	BC548, note orientation !	(G,5)
☐	9		"HEATER" screw connector	(G,4)
☐	10	C23	470 uF / 35 V	(G,5)



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 uTracerNXT to the PC.

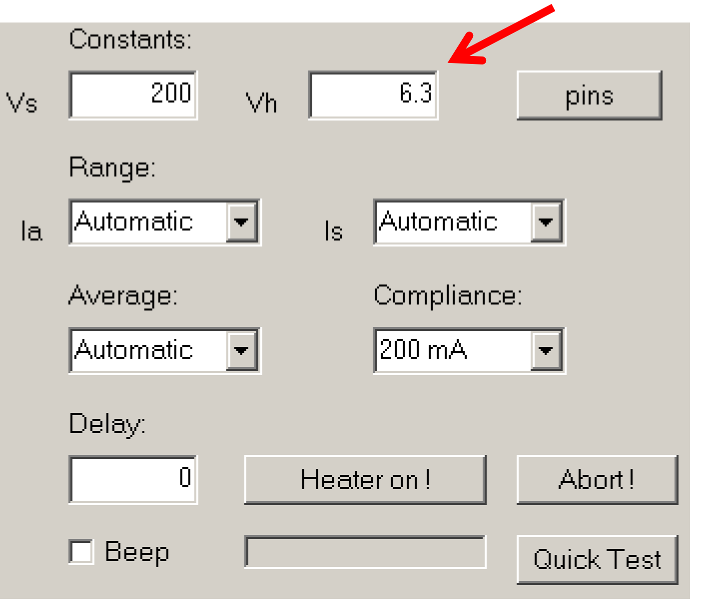
☐ 3 Connect the uTracerNXT to the power supply, if possible, set current limit to 500 mA.

☐ 4 Connect a lamp 6 or 12 volts / ≈ 100 mA to the heater terminals.

☐ 5 Start the GUI.

☐ 6 Switch-on the power supply of the uTracerNXT. The supply current without load is ≈ 60 mA.

☐ 7 Enter the voltage of the lamp in the designated field (arrow).



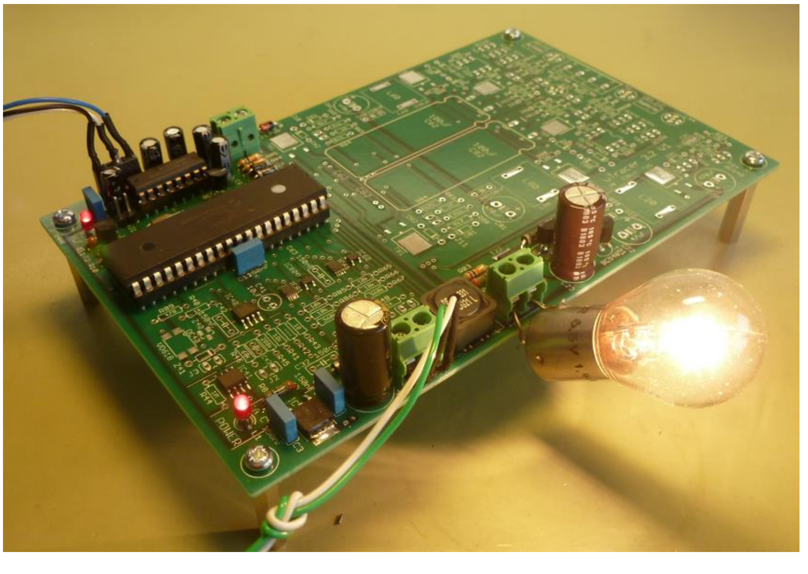
☐ 8 Press "Heater on !"

☐ 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 1 to 1.5 A. (Increase the current limit of your supply to 1A or use the final laptop adaptor.)

☐ 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 8. Negative power supply - construction

☐	1	R23	10 kohm	(F,8)
☐	2	R24	180 ohm	(F,8)
☐	3	R25	1 kohm	(F,8)
☐	4	R26	240 kohm	(F,6)
☐	5	R27	4.7 kohm (use the small 4.7 k resistor, not the large 2W)	(F,6)
☐	6	D21	UF4007, note orientation !	(F,7)
☐	7	D22	UF4007, note orientation !	(G,6)
☐	8	C21	2.2 nF	(F,8)
☐	9	T25	PMOS transistor FQD2P40TM **see Tips and Tricks no 2**	(F,7)
☐	10	L20	330 uH inductor **see Tips and Tricks no 3**	(G,6)
☐	11	T23	BC556	(F,7)
☐	12	T24	BC548	(G,8)
☐	13	C20	470 uF / 35 V	(G,7)
☐	14	C22	47 uF / 200 V	(G,6)

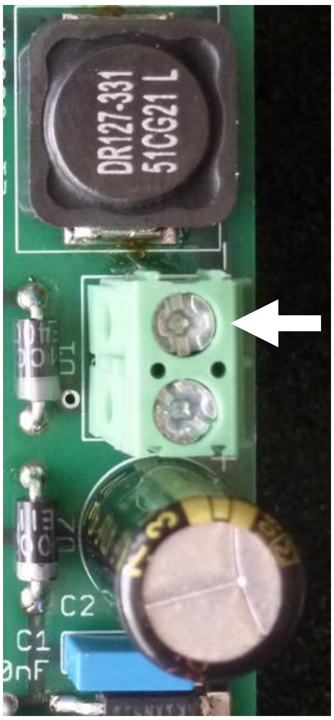
(C20 and C22:) Do not confuse the two capacitors, it will give a big bang!

Please note that the negative power supply produces a voltage of -125 V! Although this (probably) will not kill you, it will give you a nasty shock! Be careful handling the PCB when the circuit is powered

"That's got it! Hold it . . . right there!"

Part 8. Negative power supply - testing

- ☐ 1 Connect the uTracerNXT to the power supply, if possible, set current limit to 200 mA. No need to connect the uTracerNXT to the PC for this test. (supply current ≈ 85 mA @ 19.5 V)
- ☐ 2 Measure the ≈ - 125 V negative supply voltage (with respect to ground) on the point indicated



ground

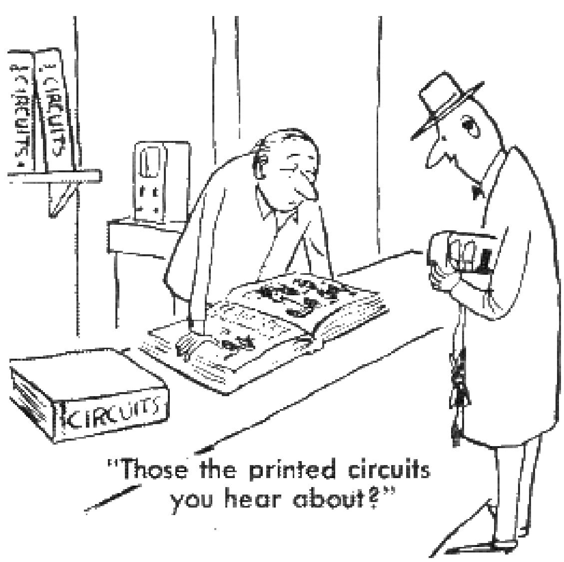
Measure approxiately -125 V on this tiny via hole (with respect to ground)

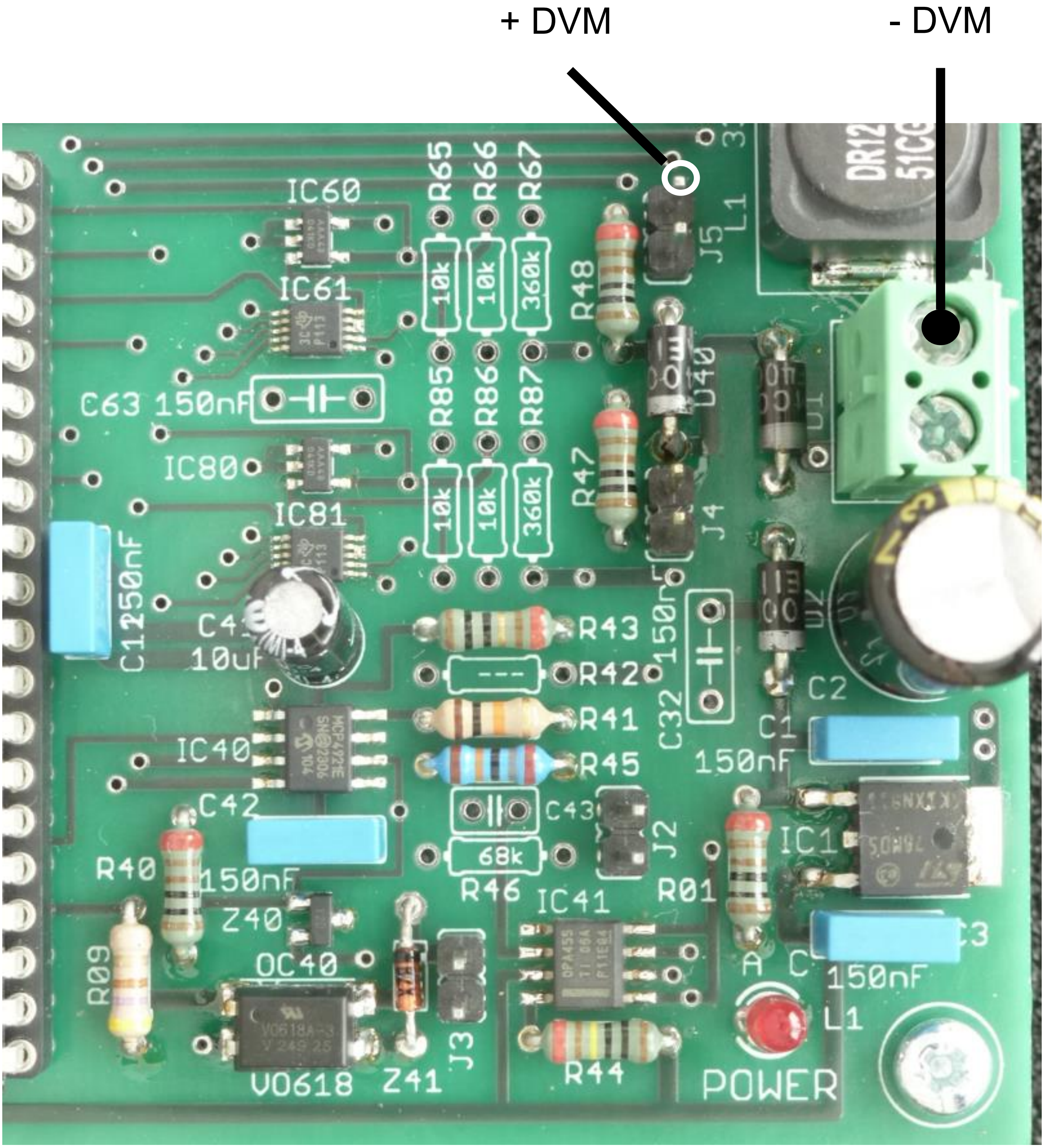
Be careful not to slip with the test pins!

Part 9. Grid bias circuit - construction

☐	1	R09	470 ohm	(D,1)
☐	2	R44	1 Mohm	(F,1)
☐	3	R41	10 kohm	(F,2)
☐	4	R43	10 ohm	(F,2)
☐	5	R45	240 kohm	(F,2)
☐	6	C43	220 pF	(F,2)
☐	7	R47	1 kohm	(F,2)
☐	8	R48	1 kohm	(F,3)
☐	9	R40	1 kohm	(D,1)
☐	10	Z41	Zener diode 5V1, note orientation !	(E,1)
☐	11	D40	UF4007, note orientation !	(F,3)
☐	12	OC40	VO618 optocoupler	(E,1)
☐	13	C41	220 uF, 10 V, note polarity !	(E,2)
☐	14	C42	150 nF	(E,1)
☐	15	J2	2 pin header, do **not** place jumper	(F,1)
☐	16	J3	2 pin header, do **not** place jumper	(E,1)
☐	17	J4	2 pin header, do **not** place jumper	(F,2)
☐	18	J5	2 pin header, do **not** place jumper	(F,3)

Note, that at this stage R42, R46, C32 as well as the jumpers have not been placed!



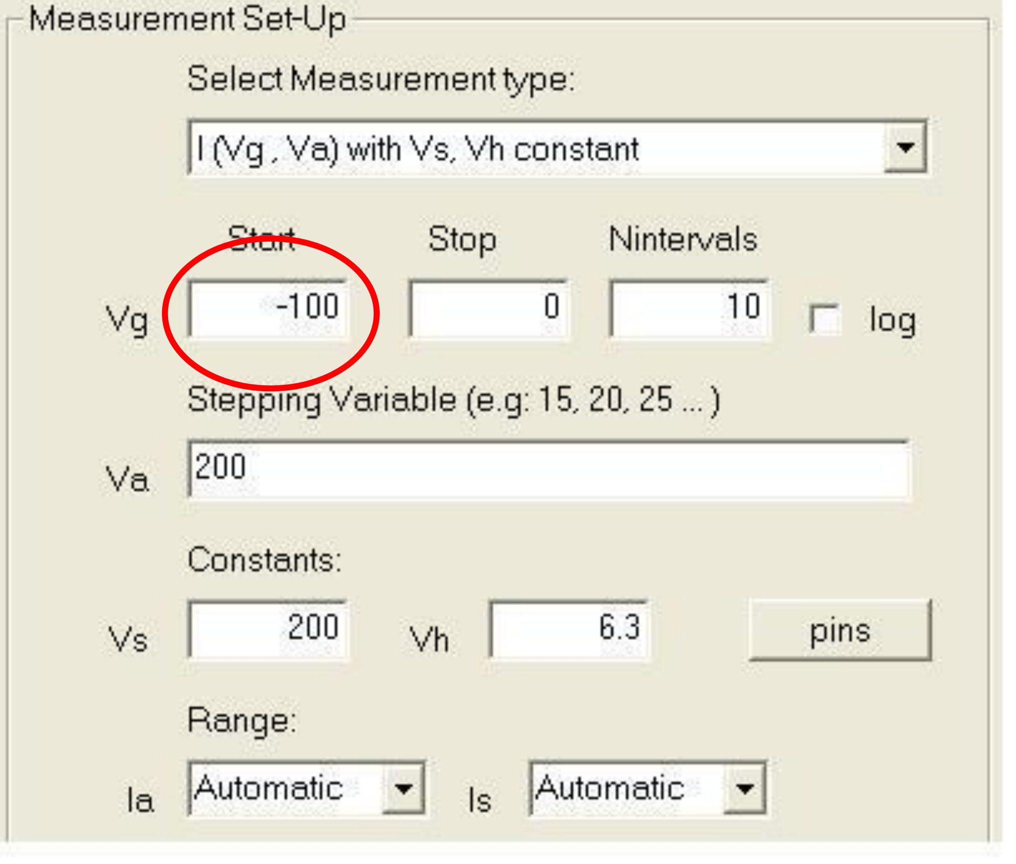


Detail of the board after construction of part 9

Please note that the spot for C32 remains open!

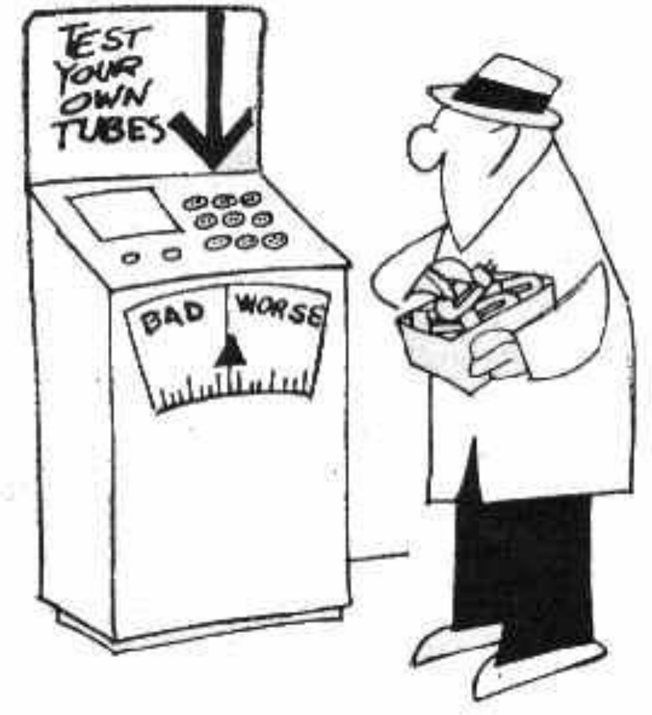
Part 9. Grid bias circuit – testing

- ☐ 1 Make sure jumpers J2, J3, J4 and J5 are not installed
- ☐ 2 Make sure that the GUI is properly installed and tested.
- ☐ 3 Connect the uTracerNXT to the PC.
- ☐ 4 Connect a DVM or multimeter between the pin indicated on the picture (previous page) and ground.
- ☐ 5 Connect the uTracerNXT to the power supply, if possible, set current limit to 200 mA.
- ☐ 6 Switch on the power supply (supply current ≈ 110 mA @ 19.5 V).
- ☐ 7 The grid voltage should be approximately 0 V +/- 100 mV. During the calibration in the next section, it will be adjusted to exactly 0 mV.
- ☐ 8 In the main form of the GUI select the measurement type indicated below and fill in -100 V as start value for the grid bias:



- ☐ 9 Press “Heater On!” in the main form.
- ☐ 10 To skip the “slow heating” feature, press again on “Heating …”
- ☐ 11 Start a measurement by pressing “Measure Curve.”
- ☐ 12 Read the actual grid voltage on the DVM or multimeter. The grid voltage should be around -100 V +/- a few volts. The exact calibration will be performed after the complete construction is finished.
- ☐ 13 Switch off the power

Note: The measurement will end with an error message.
Explanation: after the processor has set the grid voltage, it wants to charge capacitors in the high-voltage section. But since that section has not been implemented at this stage, it fails to do so, and automatically after 10 seconds a time out message is generated. You can ignore this error message.

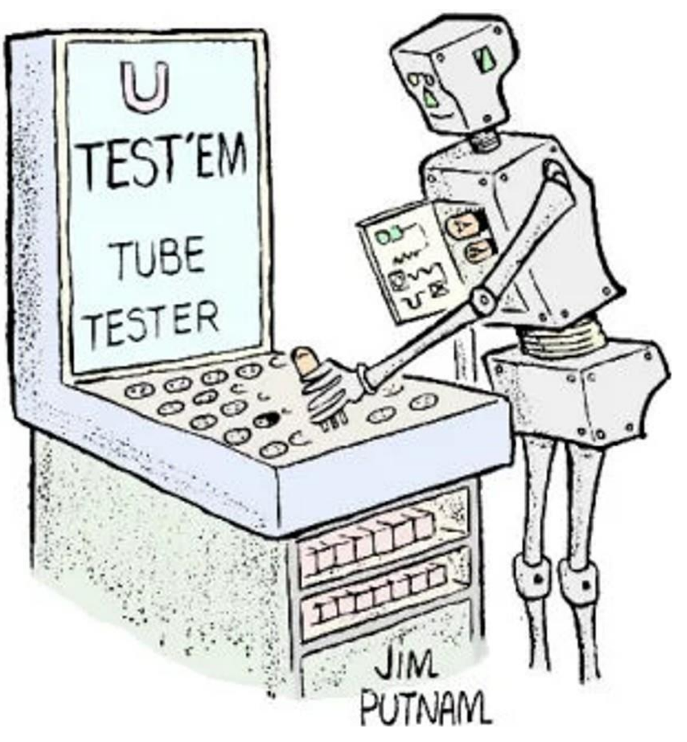


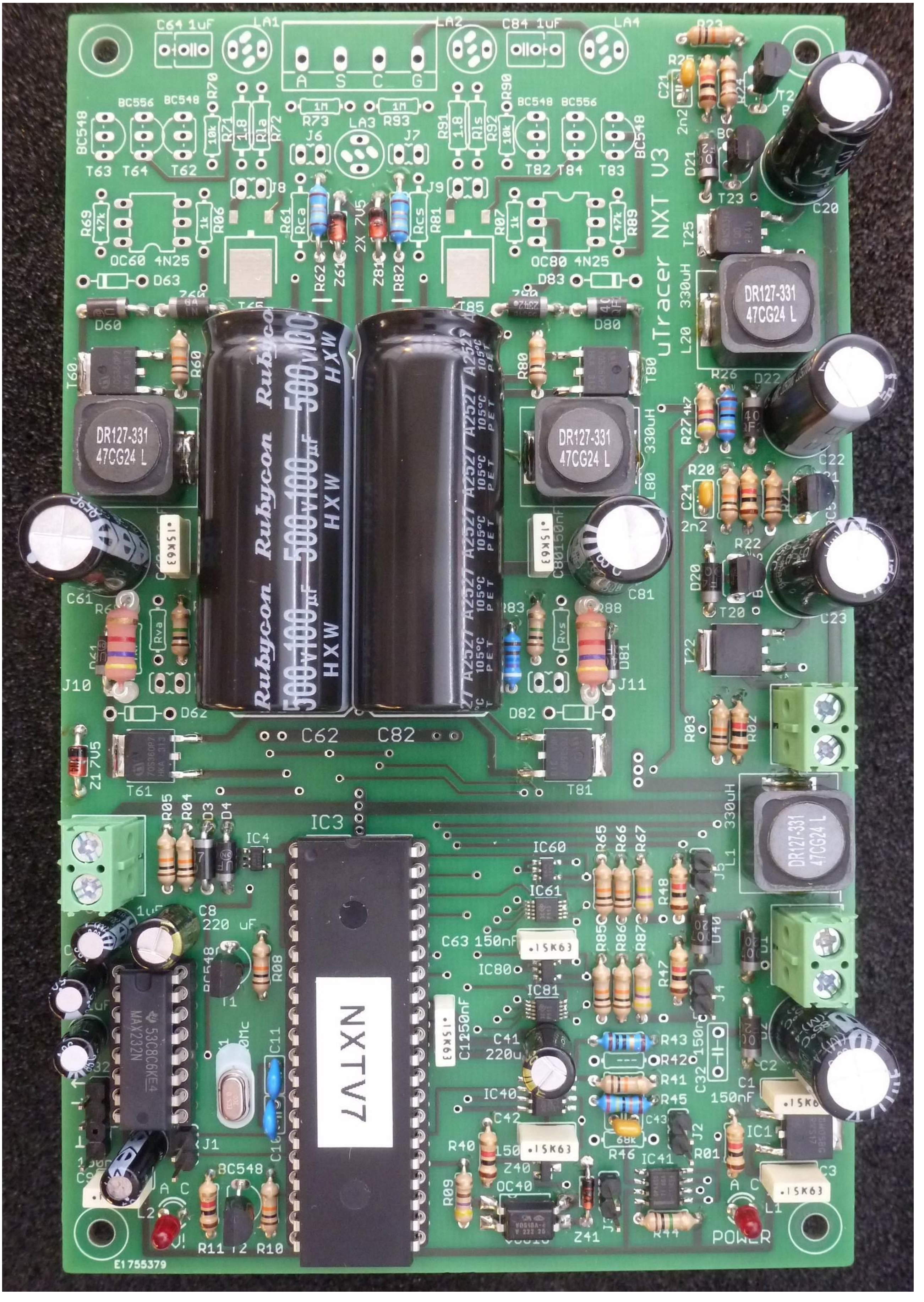
Part 10. Boost converters - construction

Please note, there are a lot of components in both this part as well as the next. A mistake is easily made, and most of the time very difficult to detect! Please work calmly and check and double check what you are doing. Especially, double check resistor values (with your DVM) and note the correct orientation of (Zener) diodes and transistors!

☐	1	R65	10 kohm	(E,3)
☐	2	R66	10 kohm	(E,3)
☐	3	R85	10 kohm	(E,2)
☐	4	R86	10 kohm	(E,2)
☐	5	R60	10 kohm	(B,6)
☐	6	R80	10 kohm	(E,6)
☐	7	R67	470 kohm	(F,3)
☐	8	R87	470 kohm	(F,2)
☐	9	R64	9.76 kohm	(B,4)
☐	10	R84	9.76 kohm	(D,4)
☐	11	R63	1 Mohm	(B,5)
☐	12	R83	1 Mohm	(E,5)
☐	13	R62	14.3 ohm	(C,7)
☐	14	R82	14.3 ohm	(C,7)
☐	15	D60	UF4007, note orientation !	(A,6)
☐	16	D80	UF4007, note orientation !	(E,6)
☐	17	D61	UF4007, note orientation !	(A,4)
☐	18	D81	UF4007, note orientation !	(E,4)
☐	19	Z60	24 V Zener diode, note orientation !	(B,7)
☐	20	Z80	24 V Zener diode, note orientation !	(E,7)
☐	21	Z61	7.5 V Zener diode, note orientation !	(C,7)
☐	22	Z81	7.5 V Zener diode, note orientation !	(C,7)

☐	23	R68	4.7 kohm, 2 Watt	**Insert caramic beads see Tip & Tricks 7**	(A,4)
☐	24	R88	4.7 kohm, 2 Watt		(E,4)
☐	25	T60	IPD70R360	**See Tip & Tricks 2**	(A,6)
☐	26	T61	IPD70R360		(A,4)
☐	27	T80	IPD70R360		(E,6)
☐	28	T81	IPD70R360		(E,4)
☐	29	L60	330 uH inductor	**See Tip & Tricks 3**	(A,6)
☐	30	L80	330 uH inductor		(E,6)
☐	31	C60	150 nF		(B,5)
☐	32	C80	150 nF		(E,5)
☐	33	C63	150 nF		(E,3)
☐	34	C61	470 uF / 35 V, note polarity!		(A,5)
☐	35	C81	470 uF / 35 V, note polarity!		(E,5)
☐	36	C62	100 uF / 500 V	**See Tip & Tricks 8 Do not clip the wires!**	(C,6)
☐	37	C82	100 uF / 500 V		(D,6)





The board after construction of part 10

Part 10. Boost converters – testing

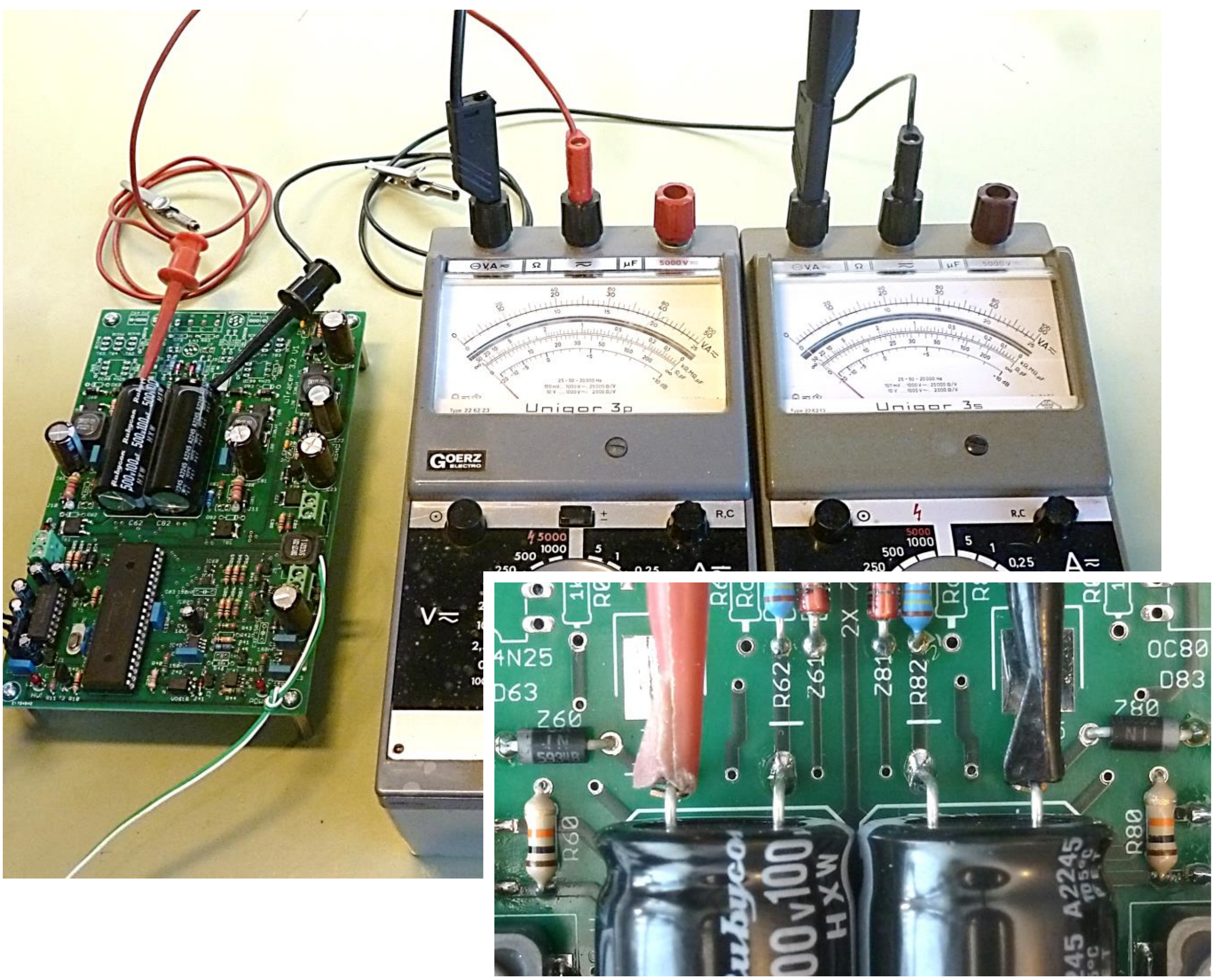
Please note that, from this point onward there are potentially lethal voltages present on the board !!
After a test / measurement, before working with the board:

1. Wait until the HV LED is off
2. Disconnect the power supply
3. Check that both 100 uF / 500 V caps are discharged

The testing consists of two separate tests !

Test 1 – testing of the boost converters:

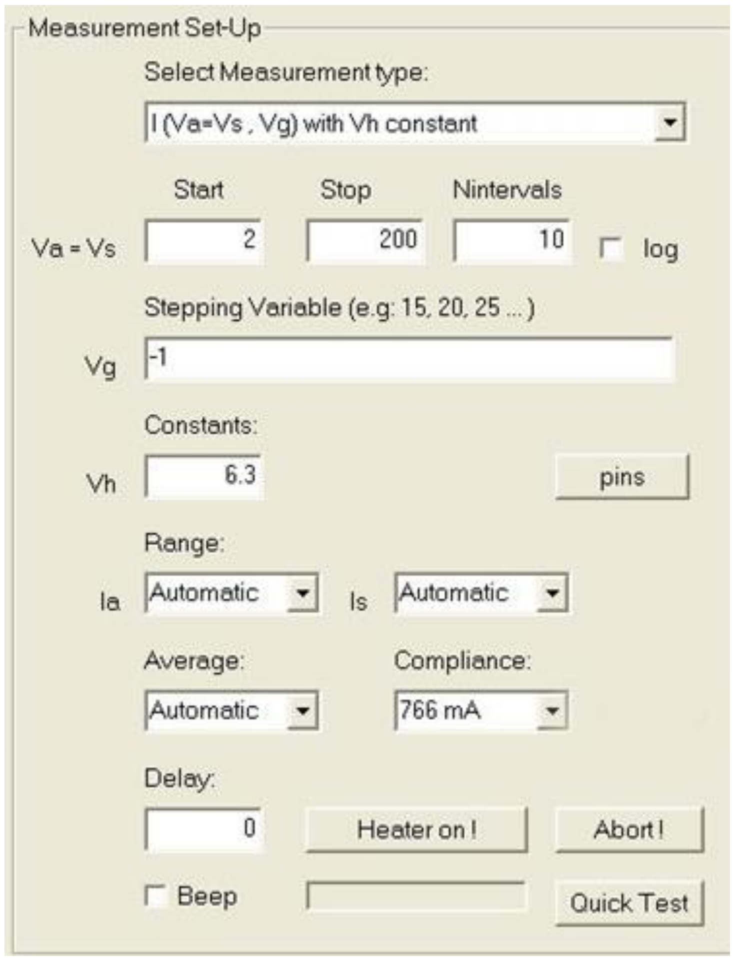
- ☐ 1 Make sure that the GUI is properly installed and tested.
- ☐ 2 Connect the uTracerNXT to the PC.
- ☐ 3 Connect a DVM or multimeter between the (+) of C62 and ground, and a second DVM or multimeter between the (+) of C82 and ground. If you only have a single DVM or meter do the measurements after each other.



☐ 4 Connect the uTracerNXT 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 uTracerNXT directly to the laptop supply you plan to use in the final setup

☐ 5 Switch on the power supply (supply current ≈ 100 mA @ 19.5 V).

☐ 6 Select the following measurement:



☐ 7 Press “Heater on!”

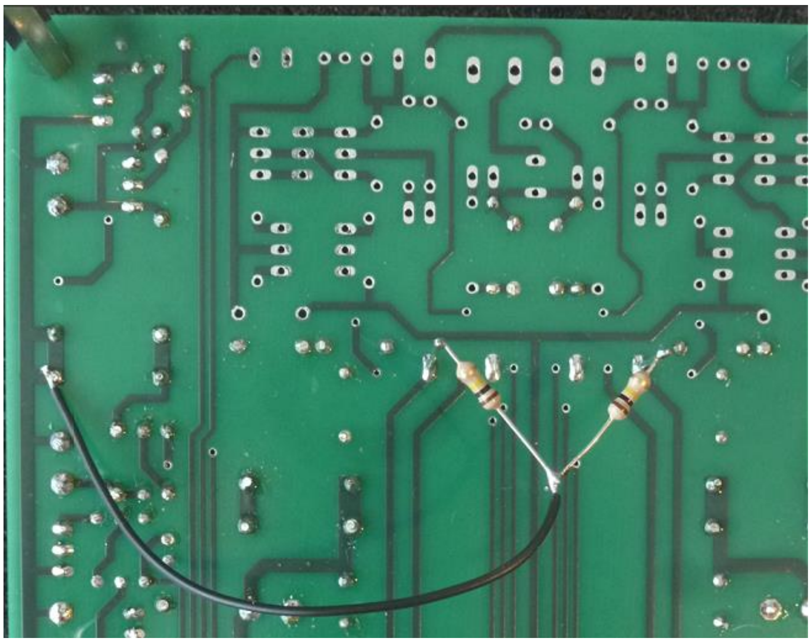
☐ 8 The text on the button now changes to “Heating.” Press the button again to skip the slow heating procedure and finally press the button again to start a measurement.

☐ 9 The voltage on the caps should now increase in ten steps from approximately 2 V (the supply voltage) to 200 V. If this does not happen stop here and first sort out what the problem is!
The graph shows no sensible data during this test.

☐ 10 Wait till the HV LED is off after the test. In any doubt check that the 100 uF capacitors are fully discharges, before handling the PCB!

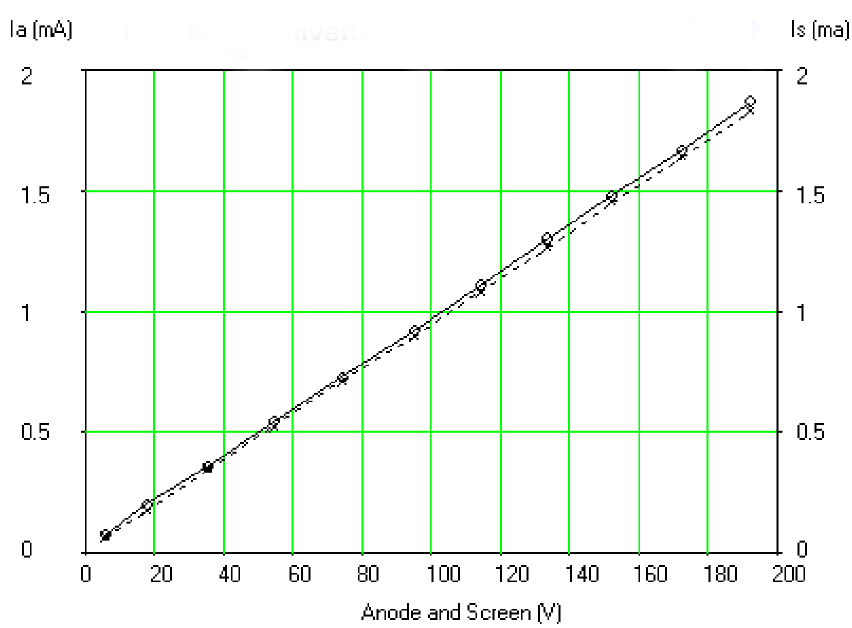
Test 2 – testing of the current amplifiers:

☐ 1 Connect a 100 kohm resistor between the (+) of C62 (the anode 100 uF cap) and ground. Connect a second 100 kohm resistor between the (+) of C82 (the screen 100 uF cap) and ground. If you did not clip the leads of these caps (see construction), you can conveniently do this on the backside of the PCB.



☐ 2 Repeat exactly the same steps (1-10) as in the previous test.

☐ 3 Select the screen current to be displayed on the right y-axis. Verify that the graph shows two straight lines as shown below. During the test, the power supply current can peak to 400 mA.



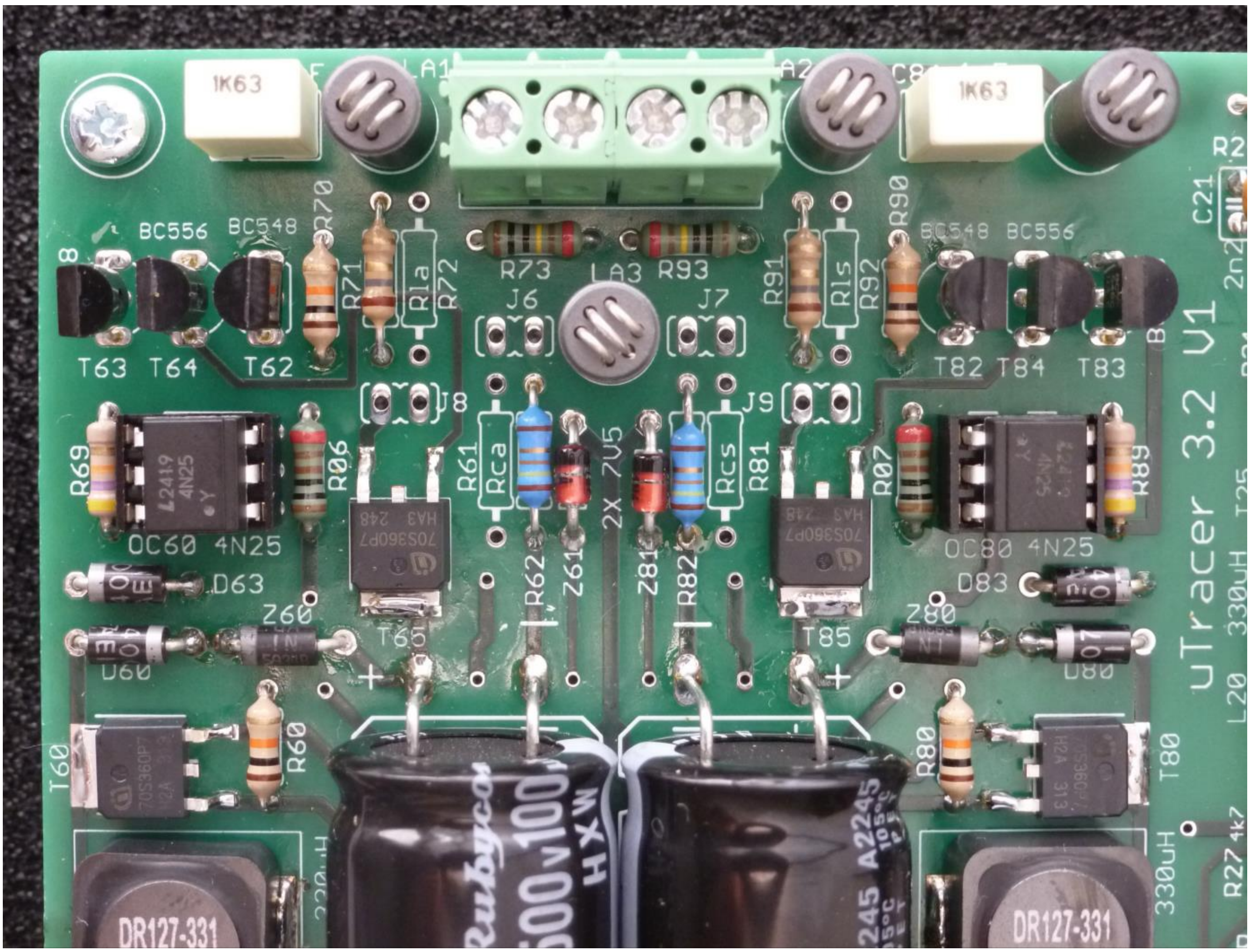
☐ 4 Remove the resistors and clip the leads of the 100 uF capacitors

Part 11. HV switches - construction

Please note that, although the layout of the HV switches
Is largely symmetrical, some components (e.g. OC60/OC80 and the discrete transistors have a different orientation!

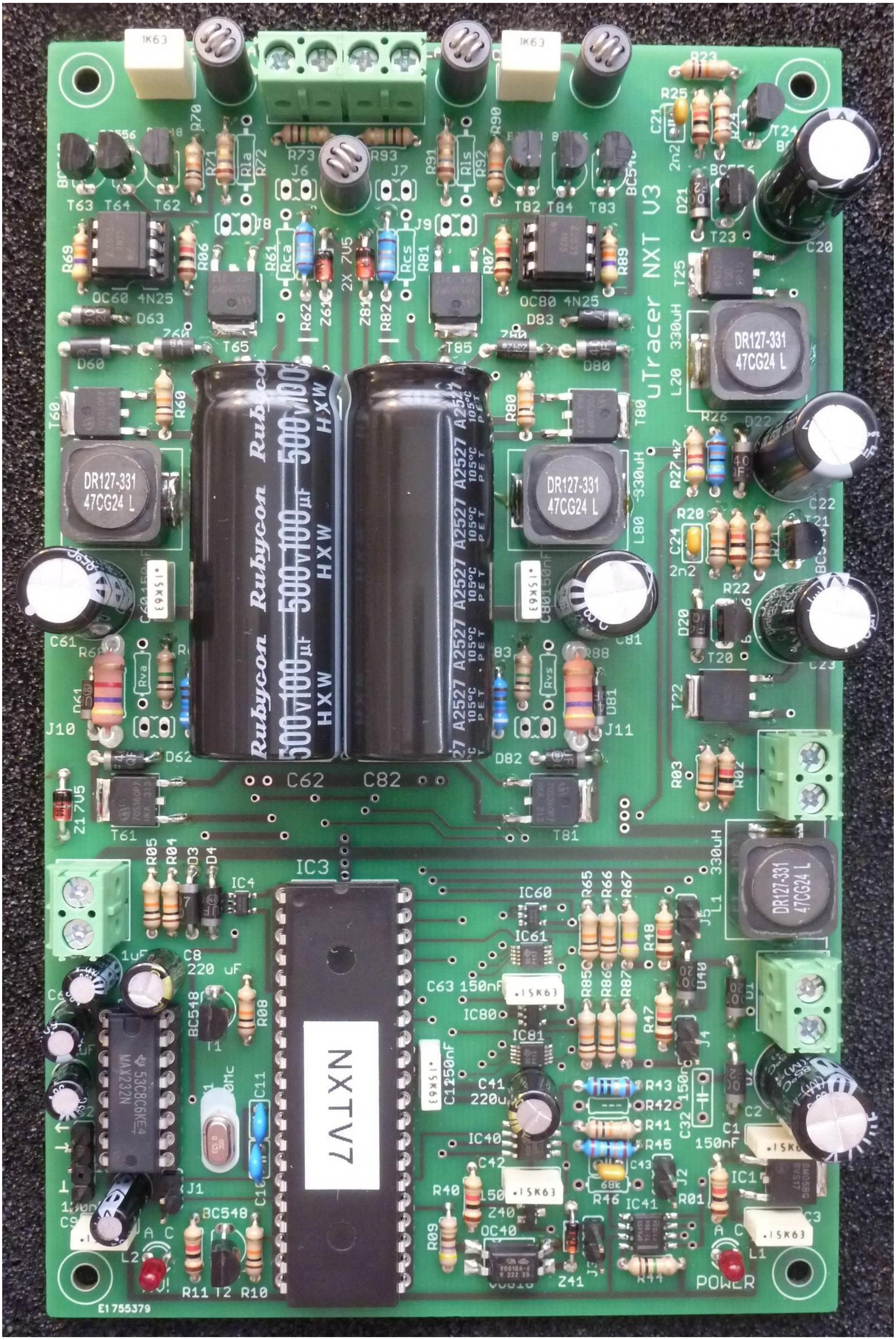
☐	1	R06	1 kohm		(B,7)
☐	2	R07	1 kohm		(D,7)
☐	3	R69	47 kohm		(A,7)
☐	4	R89	47 kohm		(E,7)
☐	5	R70	10 kohm		(B,8)
☐	6	R90	10 kohm		(D,8)
☐	7	R73	1 Mohm		(C,8)
☐	8	R93	1 Mohm		(C,8)
☐	9	R71	1.8 ohm		(B,8)
☐	10	R91	1.8 ohm		(D,8)
☐	11	D62	UF4007, note orientation!		(A,4)
☐	12	D82	UF4007, note orientation!		(E,4)
☐	13	D63	UF4007, note orientation!		(A,7)
☐	14	D83	UF4007, note orientation!		(E,7)
☐	15	T65	IPD70R360	**see Tip & Tricks 2** (15–16)	(B,7)
☐	16	T85	IPD70R360		(D,7)
☐	17	OC60	6 pin IC socket, note orientation!		(A,7)
☐	18	OC80	6 pin IC socket, note orientation!		(E,7)
☐	19	OC60	Insert 4N25 opto coupler, note orientation!		(A,7)
☐	20	OC80	Insert 4N25 opto coupler, note orientation!		(E,7)
☐	21	T64	BC556	**Do not confuse BC548 with BC558** (21–26)	(A,8)
☐	22	T84	BC556		(E,8)
☐	23	T62	BC548		(B,8)
☐	24	T63	BC548		(A,8)
☐	25	T82	BC548		(E,8)
☐	26	T83	BC548		(E,8)

☐	27	C64	1 uF foil capacitor	(B,8)
☐	28	C84	1 uF foil capacitor	(E,8)
☐	29	LA1	Ferrite RFI suppression bead	(B,8)
☐	30	LA2	Ferrite RFI suppression bead	(D,8)
☐	31	LA3	Ferrite RFI suppression bead	(C,8)
☐	32	LA4	Ferrite RFI suppression bead	(E,8)
☐	33		4 terminal screw block	(C,8)



Detail of the board after construction of part 11

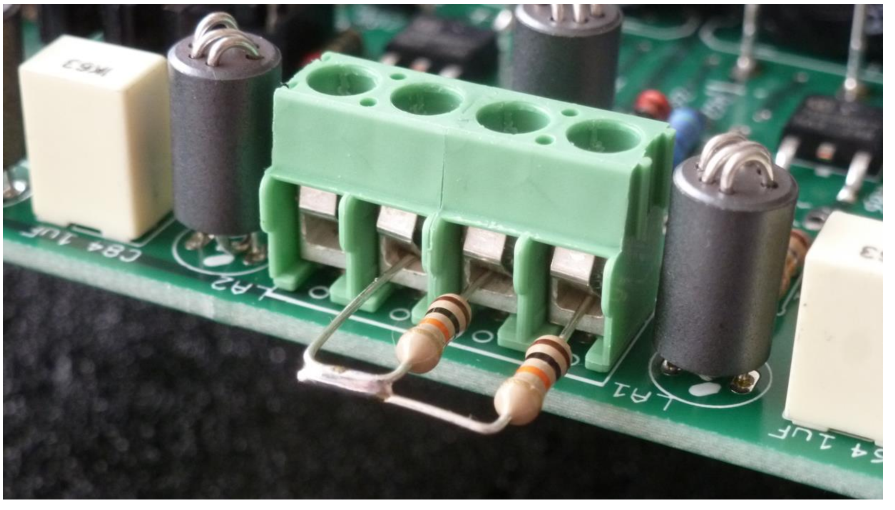
Note that at this stage jumpers J6…11 and resistors Rva, Rla, Rca, and Rvs, Rls, Rcs have not been placed. They are only used when the extended/modifies current and voltage ranges are implemented. For more information see: www.dos4ever.com/My-uTracerNXT.html



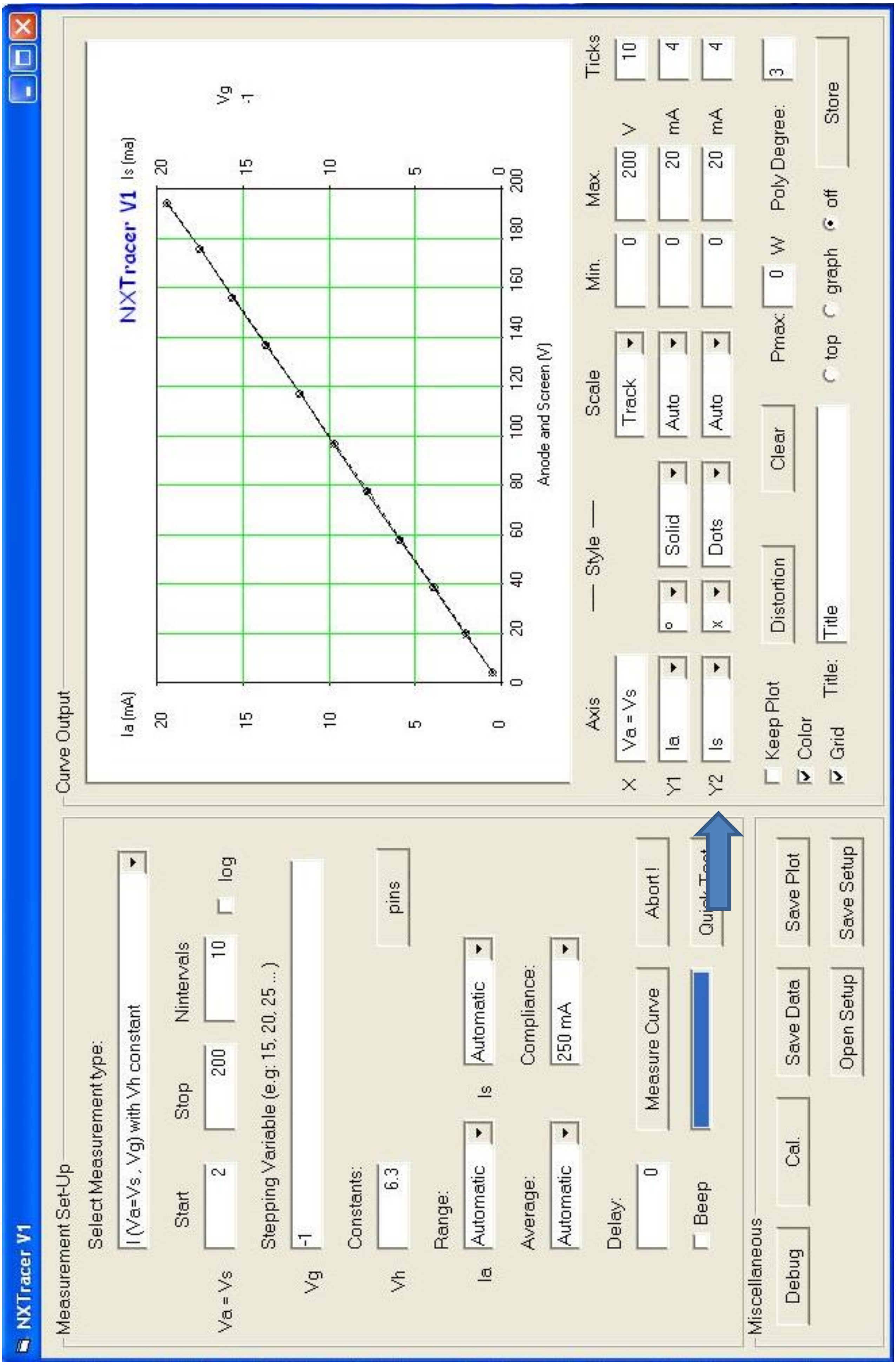
The full board after construction of part 11

Part 11. HV switches - testing

- ☐ 1 Make sure that the GUI is properly installed and tested
- ☐ 2 Connect the uTracerNXT 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 uTracerNXT 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 uTracerNXT directly to the laptop supply.
- ☐ 5 Switch on the power supply (supply current ≈ 100 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!”
- ☐ 9 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
- ☐ 10 Verify that the uTracerNXT measures two linear curves between 0 and approximately 20 mA (see picture next page).
- ☐ 11 Switch off the uTracerNXT and save the two 10 kohm resistors for the calibration phase.


NXTracer V1
Measurement Set-Up
Select Measurement type:
I (Va=Vs , Vg) with Vh constant
Start
Stop
Nintervals
Va = Vs
2
200
10
log
Stepping Variable (e.g: 15, 20, 25 ...)
Vg
-1
Constants:
Vh
6.3
pins
Range:
Ia
Automatic
Is
Automatic
Average:
Automatic
Compliance:
250 mA
Delay:
0
Measure Curve
Abort !
Beep
Quick Test
Miscellaneous
Debug
Cal.
Save Data
Save Plot
Open Setup
Save Setup
Curve Output
NXTracer V1
Ia (mA)
Is (mA)
Vg
-1
Anode and Screen (V)
Axis
Style
Scale
Min.
Max.
Ticks
X
Va = Vs
Track
0
200
V
10
Y1
Ia
o
Solid
Auto
0
20
mA
4
Y2
Is
x
Dots
Auto
0
20
mA
4
Keep Plot
Color
Grid
Distortion
Clear
Pmax:
0
W
Poly Degree:
3
Title:
Title
top
graph
off
Store

Part 12 Calibration - introduction

The calibration of the uTracerNXT is done by opening the calibration form (see screen capture next page). There are two ways to calibrate the uTracerNXT:

- By using the sliders in the upper part of the calibration form, **or**
- By directly entering calibration values into the lower part of the form.

In this manual only the calibration of the standard or basic configuration of the uTracerNXT is described. Just as a reminder, the basic version's specifications:

- Anode and screen voltage 2 – 500 V
- Anode and screen currents 0 – 350 mA (250 mA with compliance on)
- Grid voltage range 0 to -120 V

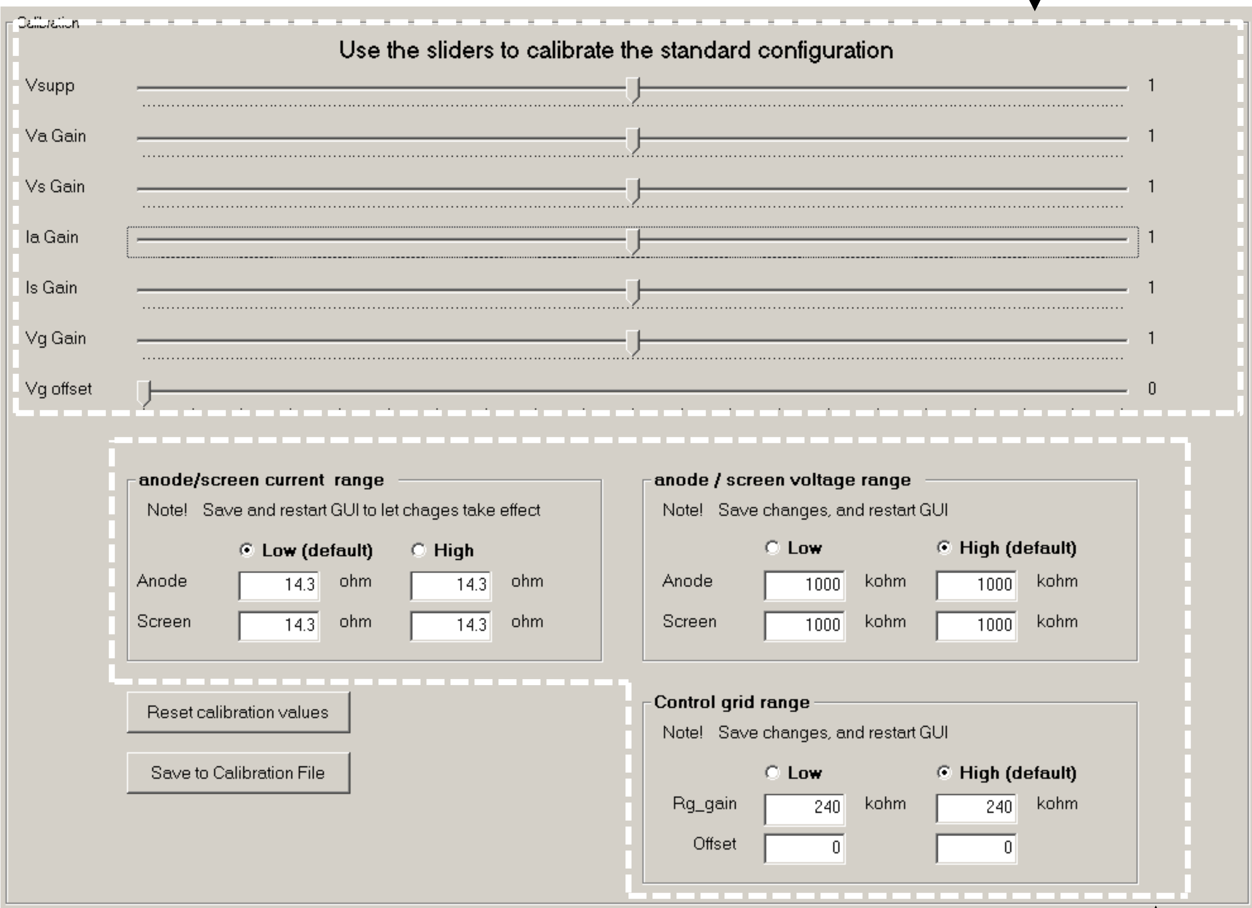
For this basic calibration **only the sliders in the upper part** of the form are used in a way very similar to the calibration of the uTracer3+. Do not modify the values in the lower part of the form!

If you decide to configure the uTracerNXT to your own needs, e.g. by changing the voltage or current ranges, a more elaborate calibration is needed. The extended calibration procedure can be found online: www.dos4ever.com/My-uTracerNXT.html

Before you configure the uTracerNXT to your own needs, it is recommended to first build and calibrate the standard version, to facilitate debugging of any potential problems that arise during the construction.

"Now, madam, are you certain there are no other parts that need explaining?"

Use the sliders to calibrate the standard configuration



Do not change these fields when using the standard configuration

In all calibration steps it is assumed that you have your uTracerNXT connected to your laptop/PC, switched on, and that the GUI is running.

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

Press “Save to Calibration File” after each calibration step to save your calibration values!

Although the calibration values are normally stored in the calibration file, it is a good idea to write them down in the form at the end of this manual so that you can retrieve them in case your computer crashes or in case of other calamities.

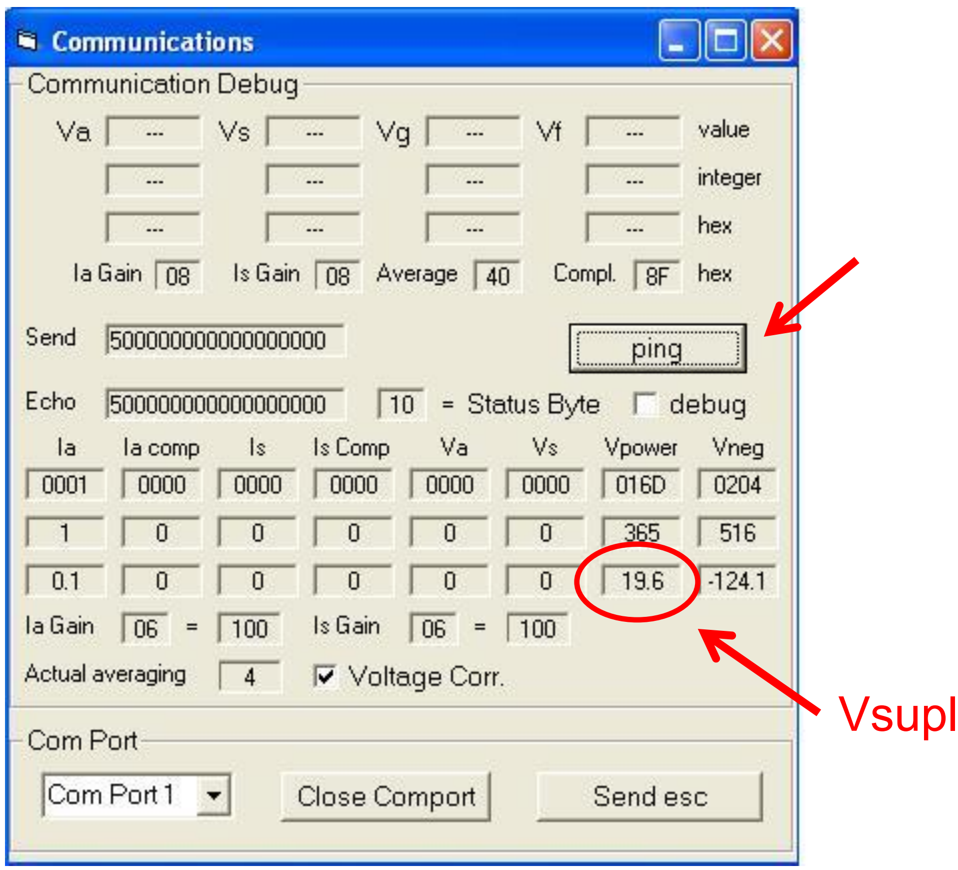
Part 12.1 Heater supply - calibration

☐ 1 Switch on your uTracerNXT

☐ 2 Open the “Debug/Communications” window by pressing “Debug” in the “Miscellaneous” section of the GUI.

☐ 3 Press the “ping” command button.

☐ 4 The debug form should now display something like:
(Ignore values underneath Ia, Iacomp, Is, Iscomp, Va, Vs)



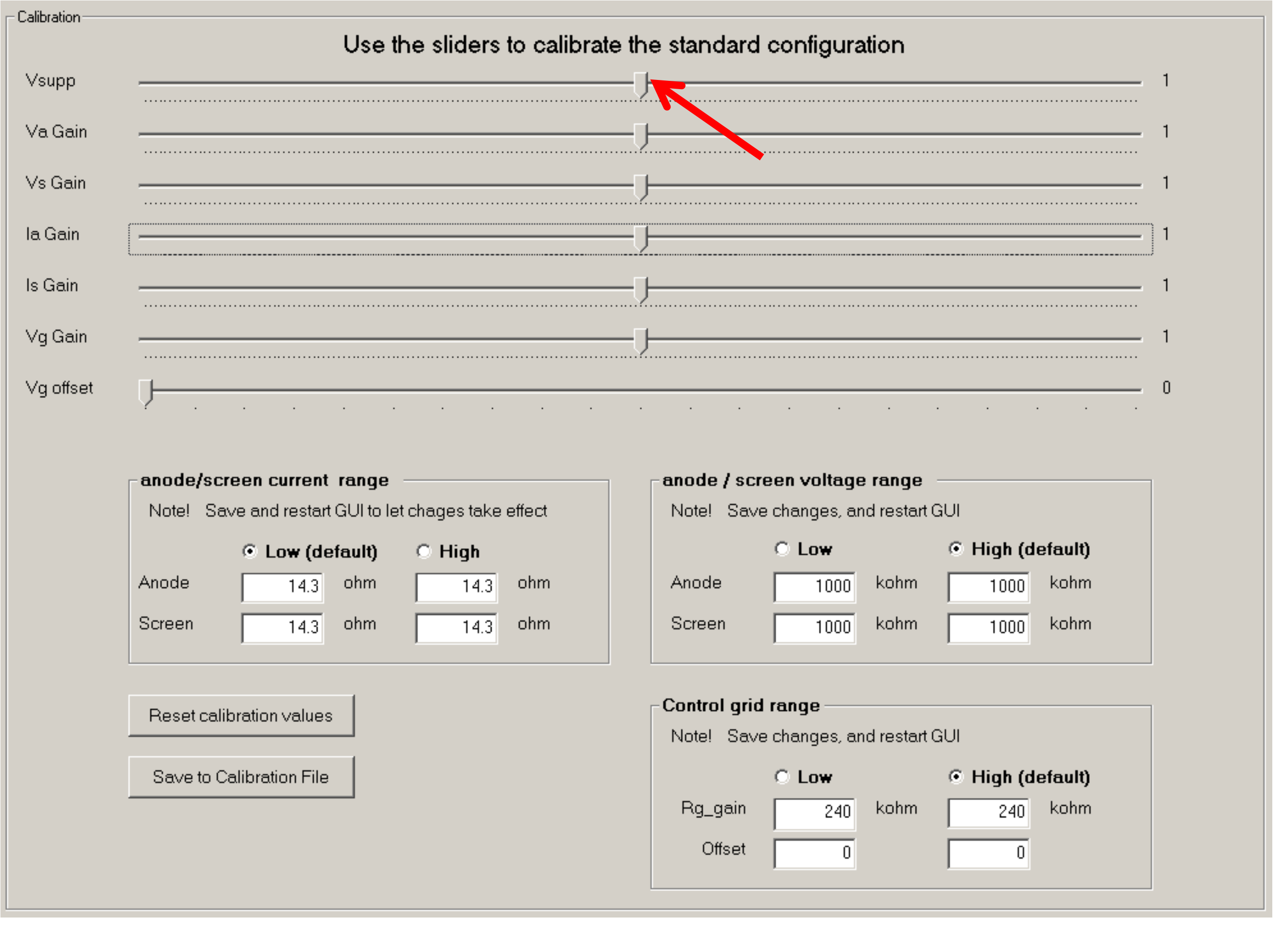
☐ 5 No error message should appear !

☐ 6 The value in the circle is the supply voltage as measured by the PIC Microcontroller.

☐ 7 Measure the real supply voltage using a multi-meter or DVM.

☐ 8 Open the Calibration form by pressing “Cal.” in the“Miscellaneous” section of the main GUI form.

☐ 9 Adjust the Vsupl slide bar (see picture next page). Move it to the left to decrease the value measured by the uTracerNXT, to the right to increase it.

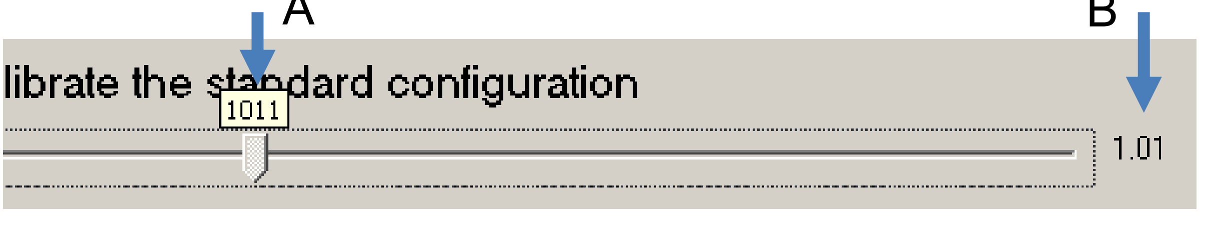


☐ 10 Press “ping” on the “Debug/Communications” form.

☐ 11 Check if the supply voltage measured by the uTracerNXT is now equal to the real supply voltage you measured yourself.

☐ 12 If not, repeat steps 9, 10 , 11

☐ 13 When equal, press “Save to Calibration File” to save the calibration values to the calibration file



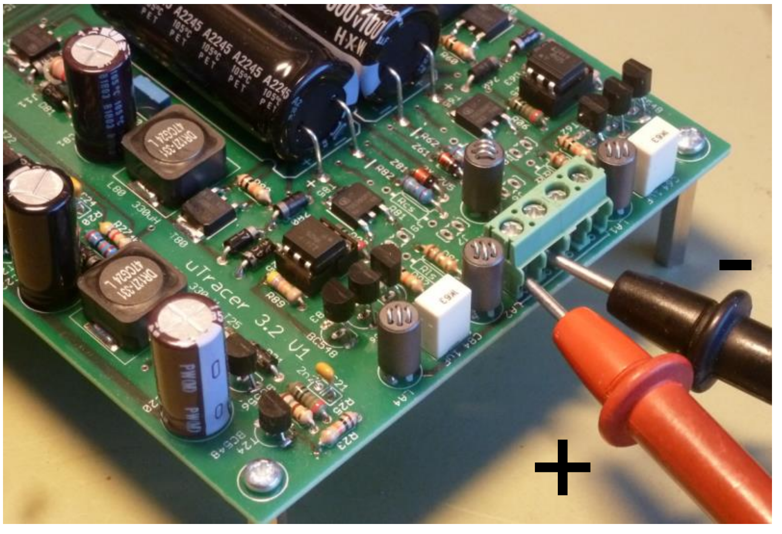
I have noticed that in some configurations of Windows, the value at the position of the slider (A), is not always copied to the recorded value at the right (B). Keep an eye on this and if this does not happen, click on the slider again!

Part 12.2 Grid bias circuit – calibration

Step I Compensating for a negative offset

The grid offset voltage can only be corrected in software when it is positive. In this step we check if the offset voltage is positive. If it is not, then depending on the value of the negative offset, resistor R42 (location F2) needs to be placed.

Connect the grid connections of the uTracerNXT to your (digital) voltmeter as shown below. Please note the correct polarity!



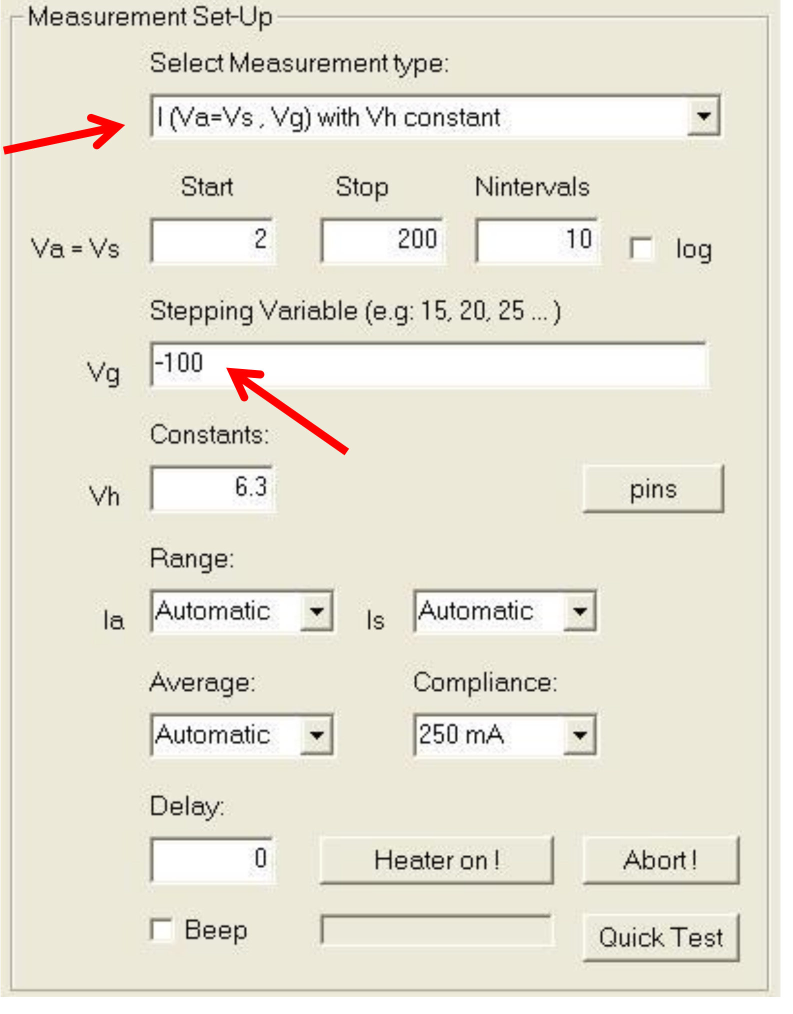
- ☐ 1 **Make sure jumpers J2, J3, J4 and J5 are removed**
- ☐ 2 Make sure that in the calibration form, the “Vg Gain” and the “Vg offset” sliders are in the positions show in the picture on the previous page.
- ☐ 3 Switch on the uTracerNXT
- ☐ 4 Wait 5 min and then read the voltage on your digital voltmeter.
  At power up, the processor resets the grid voltage to 0V.
  What you are measuring now is the offset voltage, which should be something between -100 mV and +100 mV.
- ☐ 5 If the offset voltage is positive, continue with Step II, if not continue with the next step
- ☐ 6 Switch off the uTracerNXT
- ☐ 7 If the offset voltage was
  Between -15 mV and zero → insert R42 = 47 kohm
  Between -50 mV and zero → insert R42 = 10 kohm
  Between -100 mV and zero → insert R42 = 4k7
- ☐ 8 Switch the uTracerNXT on again and verify that the offset voltage is now some positive value.

Step II Calibrating the grid bias gain

For the grid bias circuit calibration, it is recommended to have the uTracerNXT **switched on for at least 15 min** so that all components have reached their final operation temperature.

☐ 1 Make sure jumpers J2, J3, J4 and J5 are removed, and that you have your DVM connected as in Step I

☐ 2 In the main form, select the measurement type as indicated below and fill in -100 V as value for the grid bias:



☐ 3 Open the calibration form by pressing “Cal.” in the “miscellaneous” section of the main GUI form.

☐ 4 Make sure that slider Vg Gain = 1 and Vg offset = 0

☐ 5 Switch on the uTracerNXT

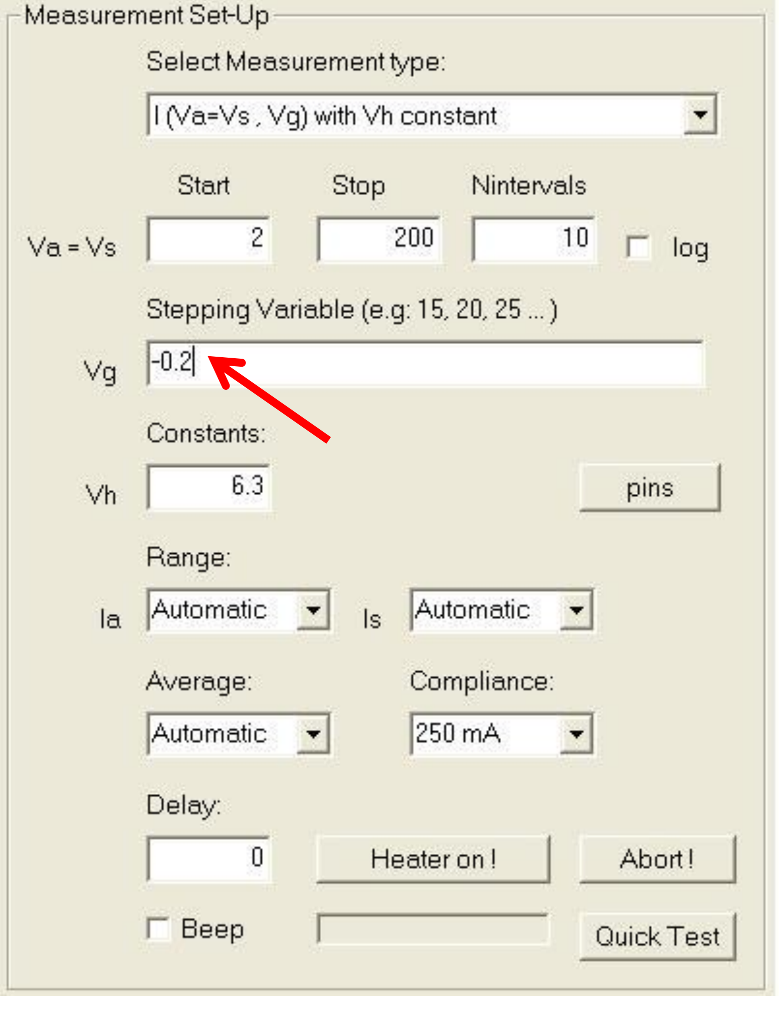
☐ 6 Press “Heater On!” on the main form

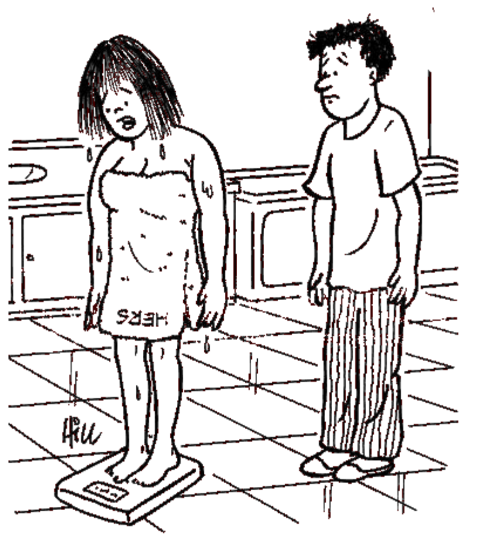
☐ 7 To skip the “slow heating” procedure press again on “Heating…. “

- ☐ 8 Start a measurement by pressing “Measure Curve”
- ☐ 9 Read the actual grid voltage on your DVM
- ☐ 10 Wait till the measurement is finished or press “Abort!”
  Wait till the High Voltage Led is off.
- ☐ 11 Adjust the “Vg Gain” slider on the “Calibration” form is the measured voltage is not equal to -100 V. (To the left if it was more negative (< -100V) than -100 V, to the right if it was less negative (> -100V) than -100 V).
- ☐ 12 Repeat steps 6 till 11 until the measured grid voltage is -100 V
- ☐ 13 Press the “Save to Calibration File” button to save the calibration value to the calibration file.

Step III Final offset correction

- ☐ 1 Make sure jumpers J2, J3, J4 and J5 are removed,
  and that you have your DVM connected as in Step I
- ☐ 2 In the main form, change the value of Vg to -0.2 (see below)





But how do we know **For Sure** that the scale is properly calibrated?

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

☐ 4 Make sure that slider “Vg Gain” indicates the value you found in the previous step and that “Vg offset” is set to zero.

☐ 5 Switch on the uTracerNXT if it is not on already.

☐ 6 Press “Heater On!” on the main form.

☐ 7 To skip the “slow heating” procedure press again on “Heating…. “

☐ 8 Start a measurement by pressing “Measure Curve”

☐ 9 Read the actual grid voltage on your DVM.

☐ 10 Wait till the measurement is finished or press “Abort!”

☐ 11 Advance the “Vg offset” slider if the grid voltage is higher (less negative) than -200 mV.

☐ 12 Repeat steps 6 till 11 until the grid voltage is as close as possible to -200 mV.
Note:
In the standard configuration the resolution of the grid voltage is 140/4096 = 35 mV. Do not expect the offset in the standard configuration to be better than this.

☐ 13 Press the “Save to Calibration File” button to save the calibration value to the calibration file.

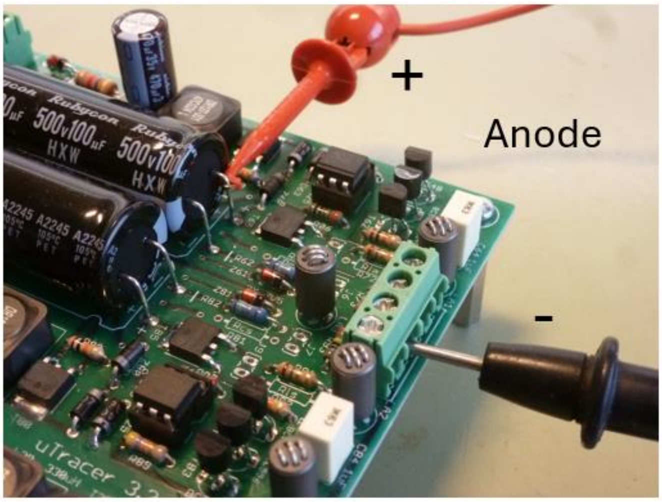
☐ 14 **Insert jumper J3.**

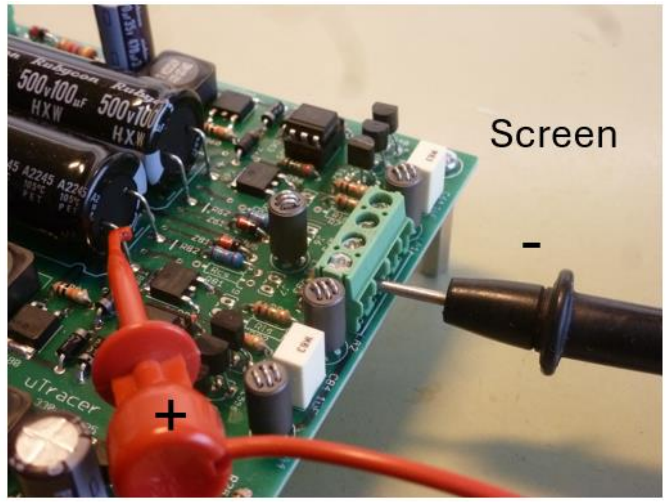
Note:
With jumper J3 placed, the grid is operated in the standard pulse mode. In this mode it is not possible to check/,measure the grid voltage with a simple DVM.

Part 12.3 Boost converters – calibration

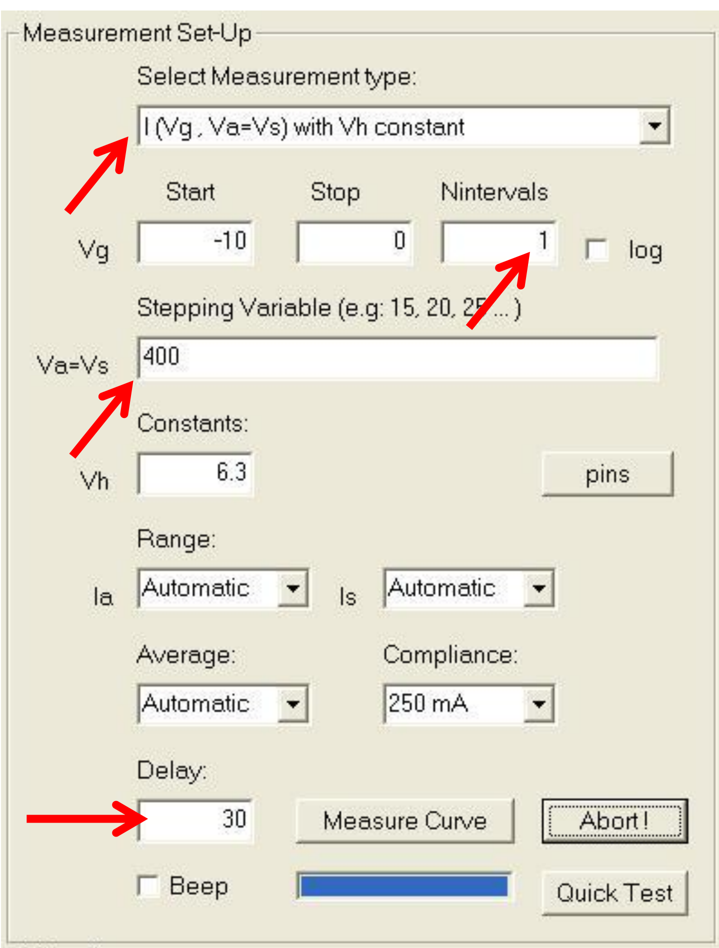
In the procedure below first the anode boost converter will be calibrated, followed by the screen boost converter.

☐ 1 Connect your DVM between the cathode terminal and the anode of C62 (100 uF anode capacitor), see left picture below





☐ 2 Configure the measurement setup as below. Note the 30 sec second delay! If your DVM cannot handle 400 V then select a lower voltage that your DVM can handle



!

Explanation:
The 30 second delay introduces a 30 seconds stabilization pause for the voltages to stabilize before a measurement. We use this pause to read out the voltages.

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

☐ 4 Switch on your uTracerNXT

☐ 5 Press “Heater on!”

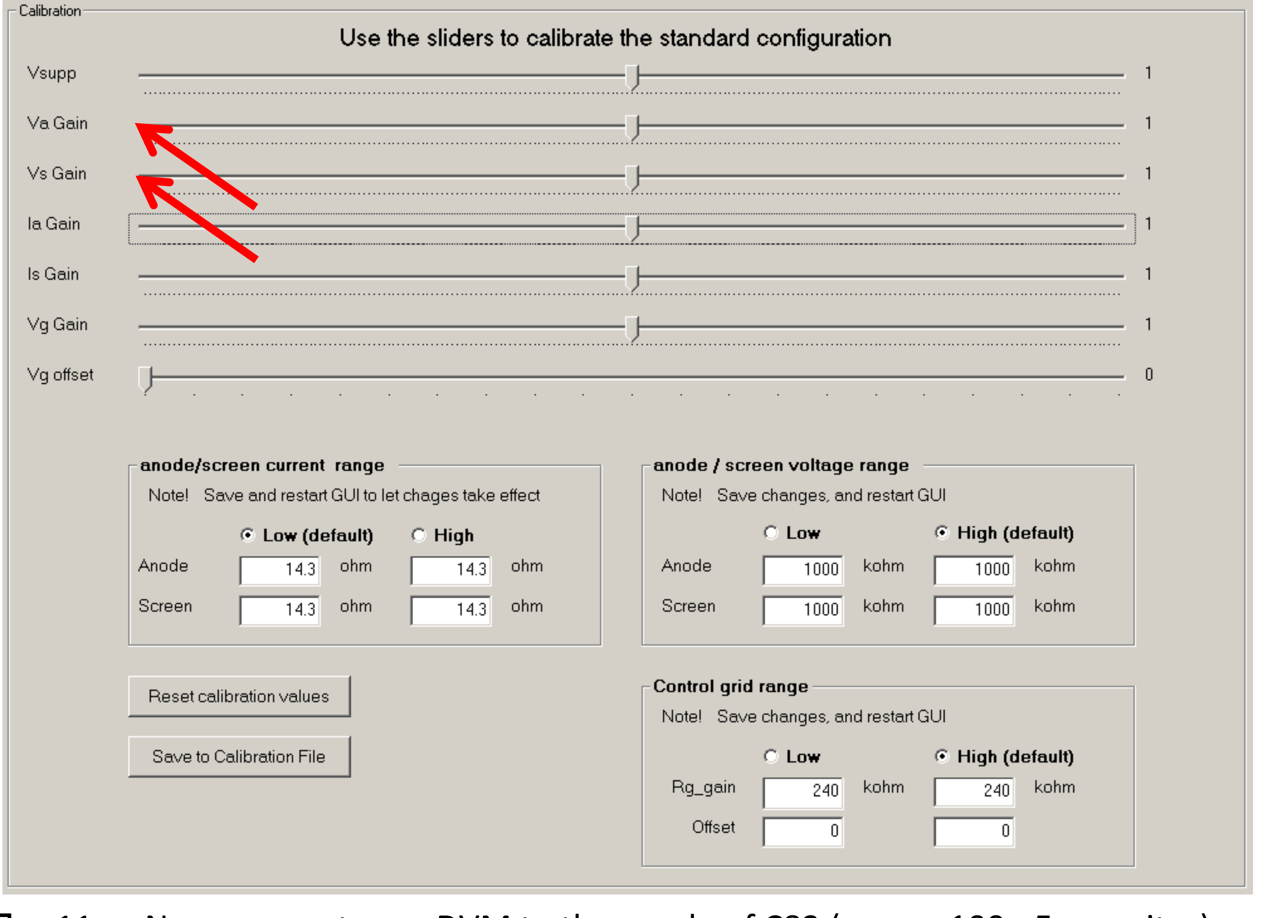
☐ 6 The text on the button now changes to “Heating” Press the button again to skip the slow heating procedure and finally press the button again to start a measurement.

☐ 7 Read the voltage on your DVM or multimeter.

☐ 8 Press “Abort” to stop the measurement.
Wait until the High Voltage Led is off!

☐ 9 If the measured voltage does not equal 400 V (or the voltage you selected), then adjust the “Va Gain” slider in the calibration form. (to the right of the voltage was too low, to the left if it was too high).

☐ 10 Repeat steps 5 to 7 until the measured voltage matches the selected voltage (e.g. 400 V)



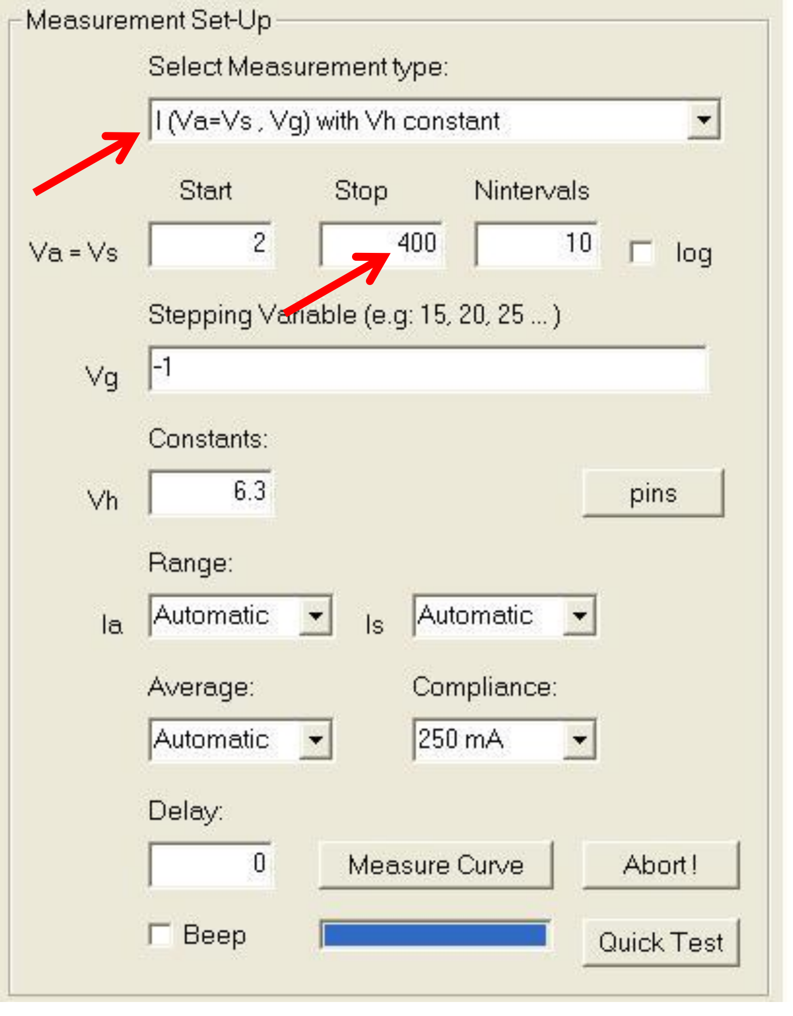
☐ 11 Now, connect your DVM to the anode of C82 (screen 100 uF capacitor). See right photograph on the previous page. Repeat the calibration procedure but now adjust the “Vs Gain” slider.

☐ 12 Press “Save to Calibration File” and switch off your uTracerNXT

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.



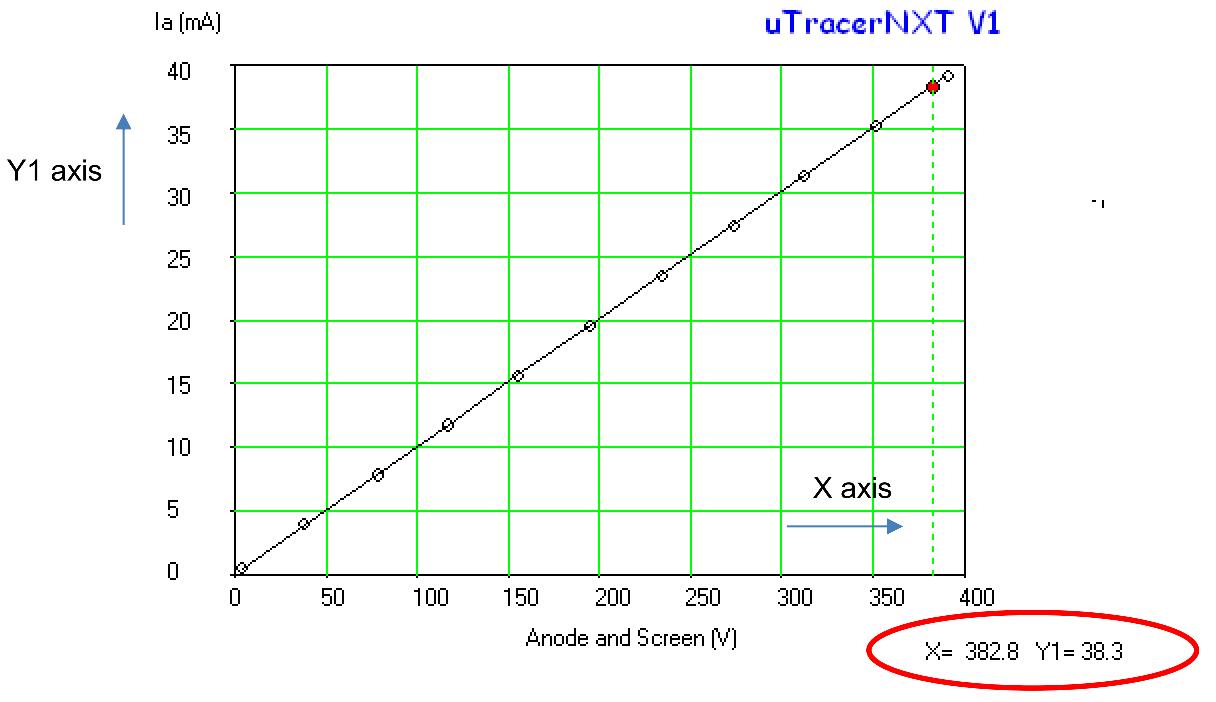
☐ 3 Open the calibration form by clicking on "Cal." in the miscellaneous section of the main GUI form.

☐ 4 Switch on your uTracerNXT

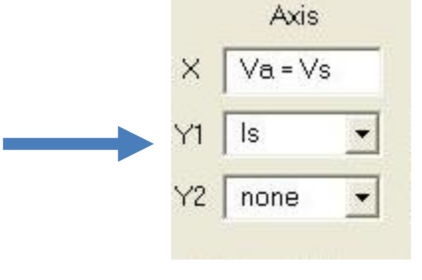
☐ 5 Press "Heater on!"

☐ 6 The text on the button now changes to "Heating." Press the button again to skip the slow heating procedure, and then press the button again to start a measurement.

☐ 7 Since a resistor is connected to the anode output, the measurement will produce a straight line (see picture next page).



- ☐ 8 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 kohm resistor, Y1 is the current flowing through the resistor.
  Note that the current should be Y1 = X / 10 kohm.
  So when X = 382.8, ideally Y1 = 382.8 / 10.000 = 38.3 mA.
- ☐ 9 If Y1 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.
- ☐ 10 Repeat steps 6 to 9 until the values match
- ☐ 11 Press “Save to Calibration File”
- ☐ 12 Now select the screen current Is to be plotted on the Y1 axis:



- ☐ 13 Repeat steps 6 – 11, but now adjust the “Is Gain” slider
- ☐ 14 Save calibration values by pressing “Save to Calibration File”

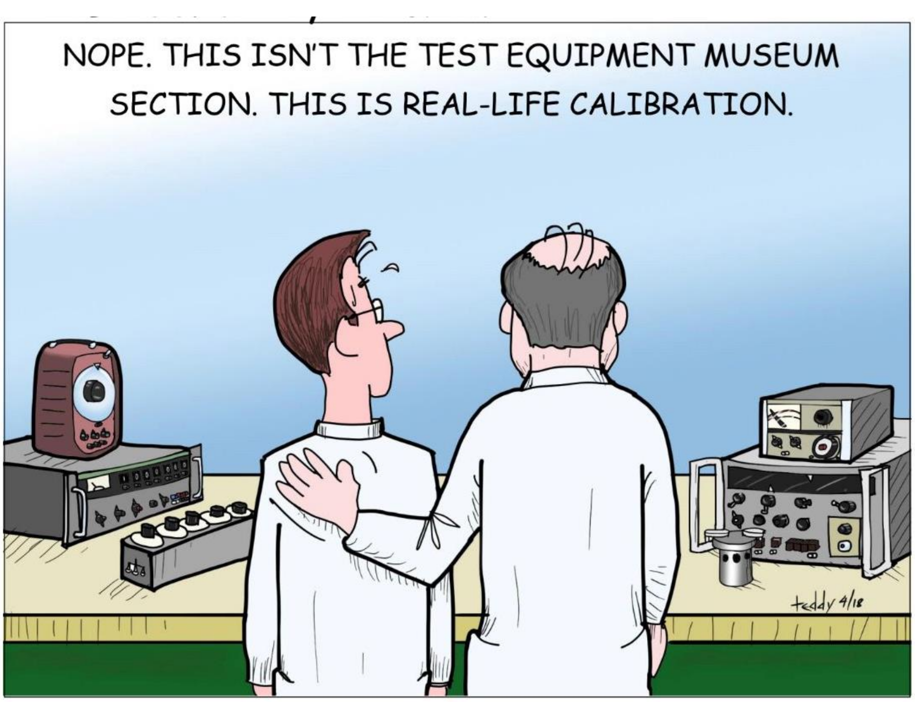
This completes the calibration of the standard configuration of your µTracer NXT!

Part 12.4 Backup calibration values

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

Vsupp	
Va Gain	
Vs Gain	
Ia Gain	
Is Gain	
Vg Gain	
Vg offset	

Again, make sure jumper J3 is installed for normal use of the uTracerNXT !



	Item(s)	Description	Loc.
anode adjust	J6 & Rca	Jumper J6 and resistor Rca, activate and define the increased current sensing range for the anode channel	B7 & C7
	J8 & Rla	Jumper J8 and resistor Rla, activate and define the increased current limit range for the anode channel	C7 & B8
	J10 & Rva	Jumper J1 and resistor Rva, activate and define the reduced voltage range for the anode channel	A4 & B4
screen adjust	J7 & Rcs	Jumper J7 and resistor Rcs, activate and define the increased current sensing range for the screen channel	D7
	J9 & Rls	Jumper J9 and resistor Rls, activate and define the increased current limit range for the screen channel	D7 & D8
	J11 & Rvs	Jumper J11 and resistor Rvs, activate and define the reduced voltage range for the screen channel	E4
control grid section	J2 & R46	Jumper J2 and resistor R46, activate and define the reduced voltage range for the grid	F1
	J3	When placed the control grid is pulsed, just as the anode and screen channels. **It is normally placed.** J3 is only removed to check the grid voltages with a DC voltmeter	E1
	J4	Placing jumper J4 short circuits resistor R47 which is in series with the control grid and which protects the circuit against short circuits. It is normally not placed	F2
	J5	Placing jumper J5 short circuits resistor R48 which is in series with the control grid and which protects the circuit against short circuits and flash-overs. It is normally not placed	F3
misc.	J1	When placed directly connects the Tx and Rx channels of the MAX232. **It is used only to test the serial link and MAX232 when the PIC is not placed!**	B1

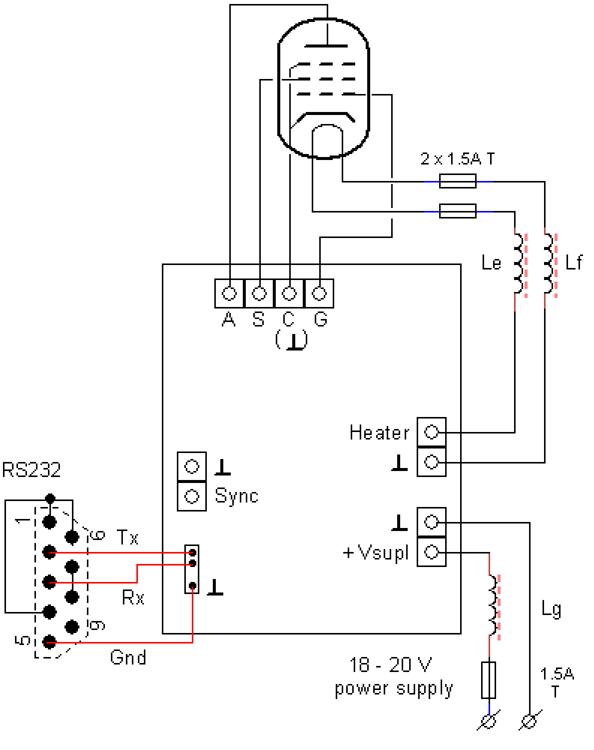
Wiring of the uTracer NXT

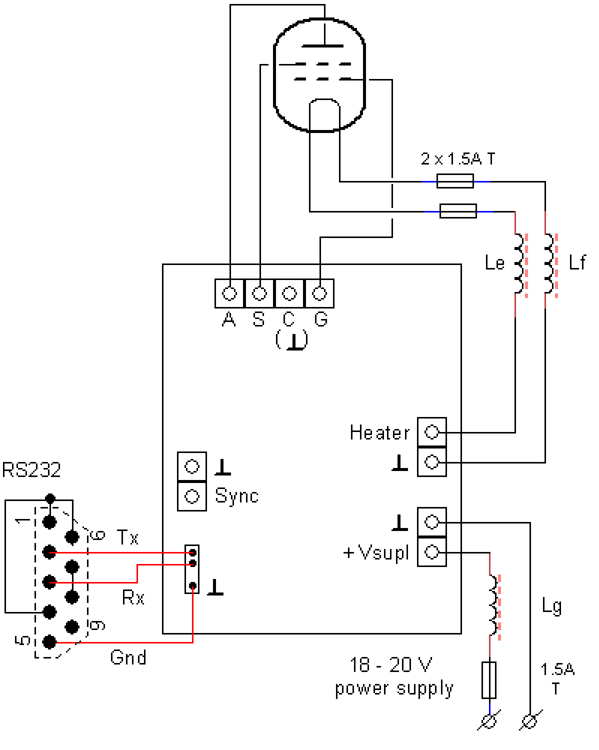
The uTracerNXT was designed to operate from an old laptop power supply with an output voltage in the range of 18 – 21 V. These power supplies combine a high output power with a small volume, and not in the least, low cost!

The figures below shows the principal way the uTracerNXT board is connected to the power supply, the tube, and the RS232 connector, in case the **internal heater supply** is used.

The picture on the left shows the situation in case an indirectly heated tube is used, in this example a pentode. The picture on the right shows the situation in case a directly heated tube is used, in this example a tetrode. Of course, the schematic is the same in case a triode is connected, apart from the fact that in that case the screen terminal of the uTracerNXT is not used.

It is recommended to include a (surplus) 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.





An important difference of the uTracerNXT compared to the uTracer3+ and uTracer6 is that both the cathode as well as the heater are referenced to ground instead of to the + of the power supply!

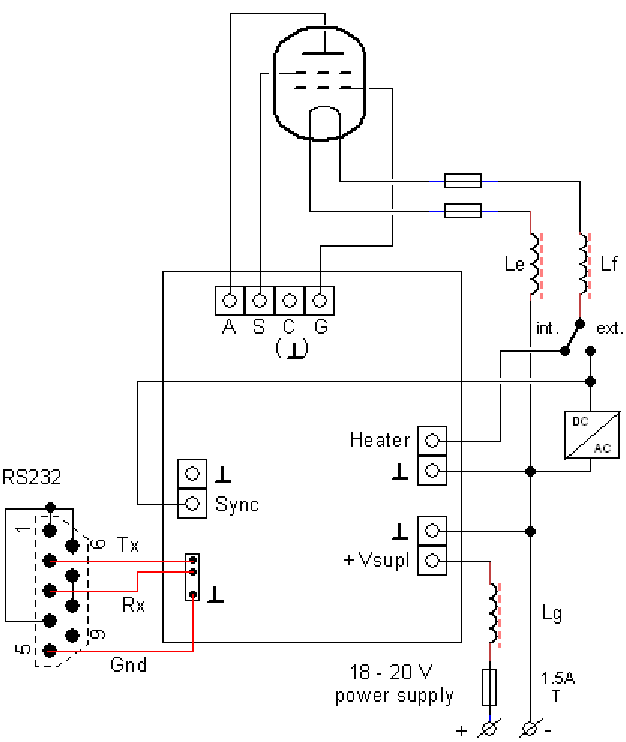
Observe that on the PCB all three terminal blocks have their own ground connection. On the PCB all these ground connections are connected to each other. However, it is recommended to wire your PCB as indicated in the schematic to avoid heavy currents (e.g. from the heater) flowing through the traces on the PCB.

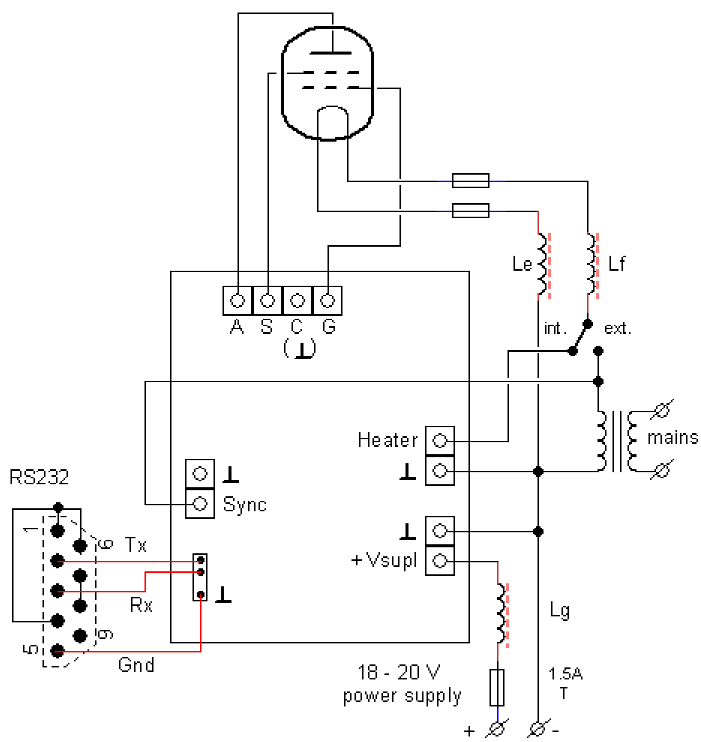
Many users prefer to use an external heater supply for their uTracer.

The left figure below shows the recommended wiring in case an **external power supply** is used for the heater. In this example a directly heated tube is shown, the connections for an indirectly heated tube are the same, apart from the fact that the cathode of the tube needs to be connected to the cathode connection of the uTracerNXT.

In the figure a switch was added to that the user can chose between the internal and the external heater supply.

Note, how the external heater supply is referenced to ground! This is an important difference with the uTracer3 and the uTracer6! This has the important advantage that the power supply that is used to power the uTracerNXT board, can now also be used to power the heater supply if needed.





A new feature of the uTracerNXT is that – in contrast to the uTracer3 and uTracer6 – an (AC) transformer can be used to power the heater of **directly heated tubes!**.

To highlight this, the picture on the right was added where the schematic AC/DC heater supply block is replaced by a transformer.

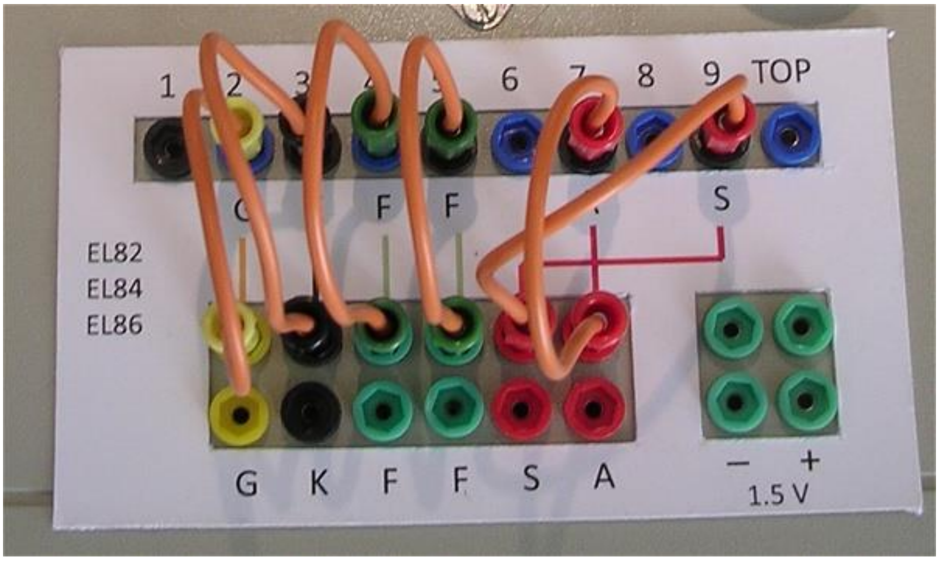
To make this possible a synchronization circuit has been added that synchronizes the measurement pulse with the AC period phase. This requires a connection between the heater circuit to the synchronization input as shown in the pictures. Note that in case a DC heater supply is used, this connection can remain in place as the software recognizes that a DC supply is used.

Tips for connecting the tube sockets

Probably everybody will have his (or hers!) on ideas on how to connect the uTracerNXT 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 uTracerNXT 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 uTracerNXT 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.

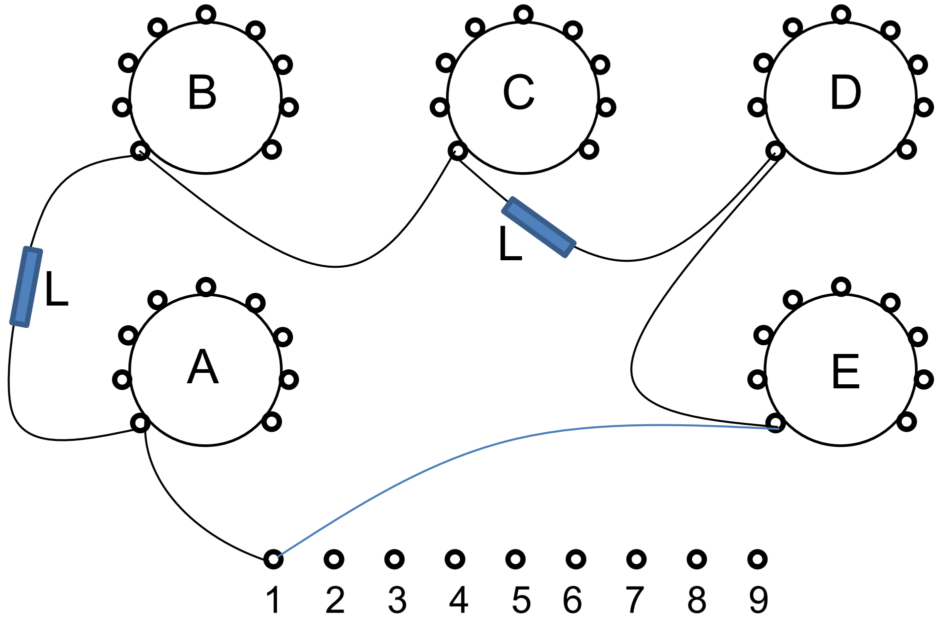
For conventional tube testers, such a plug-and-wire solution would be rather dangerous because high voltages can be present on the terminals. In this case the danger is very limited because the high voltages are only present for 1 millisecond, and even then, the amount of charge stored in the capacitors 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

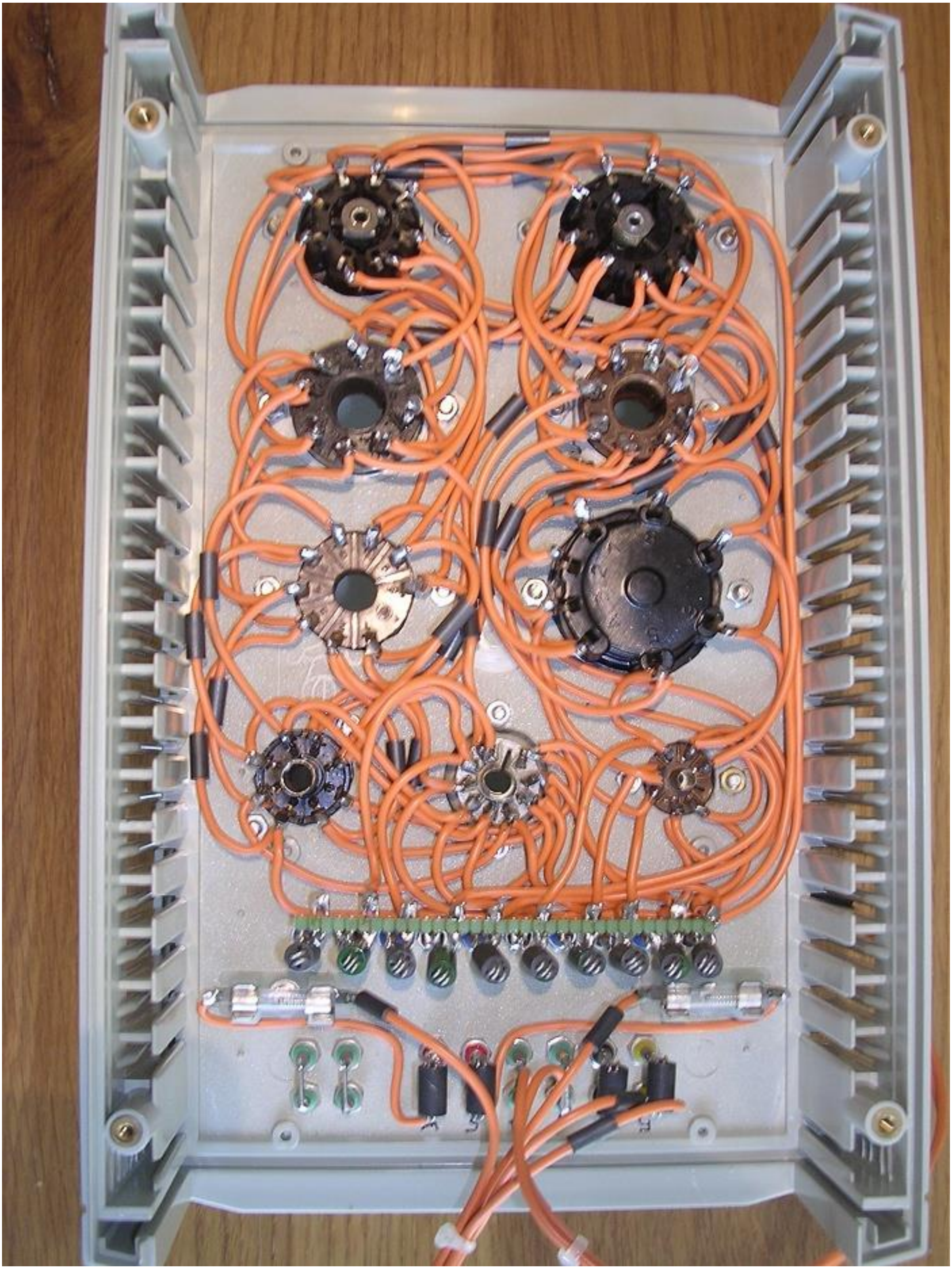
With conventional tube testers, it can very well happen that a tube is destroyed because a destructive oscillation occurs! A big advantage of the uTracerNXT is that the maximum energy stored in the reservoir capacitors is very small, too small to destroy a tube. The oscillations can however disturb the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

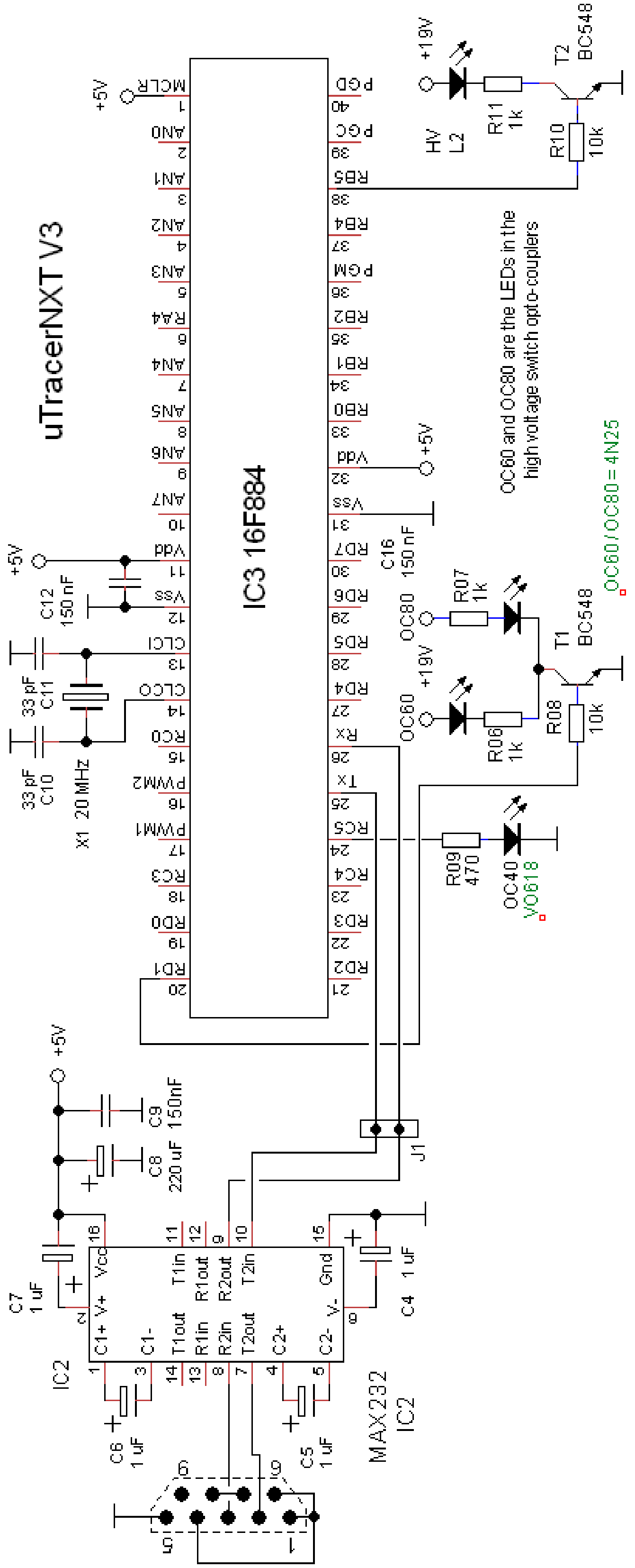
One of the most effective ways to achieve this is to include RF suppression coils in the tube connections. At DC the resistance of these coils is almost zero, while the resistive losses quickly increase for increasing frequencies.

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.

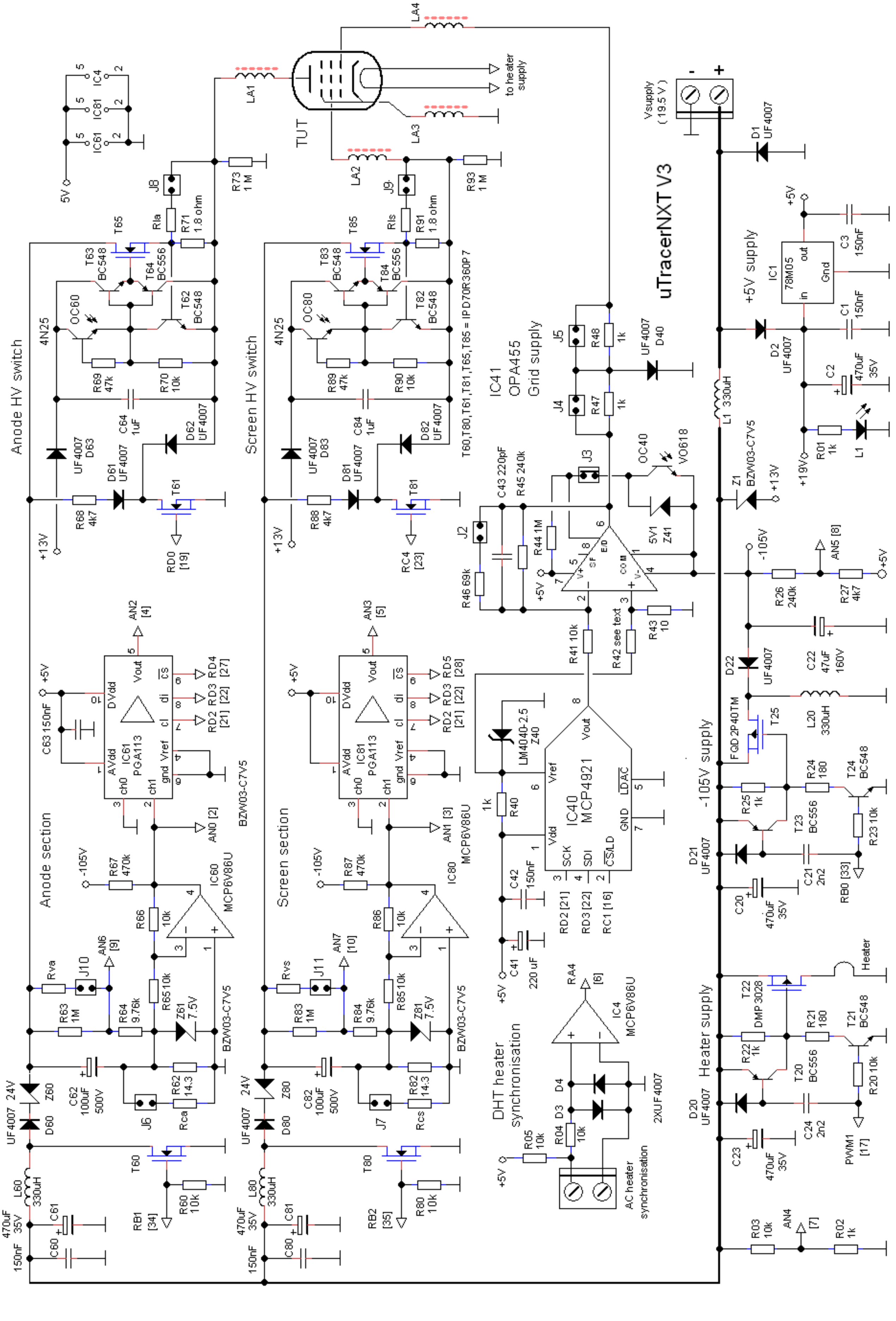


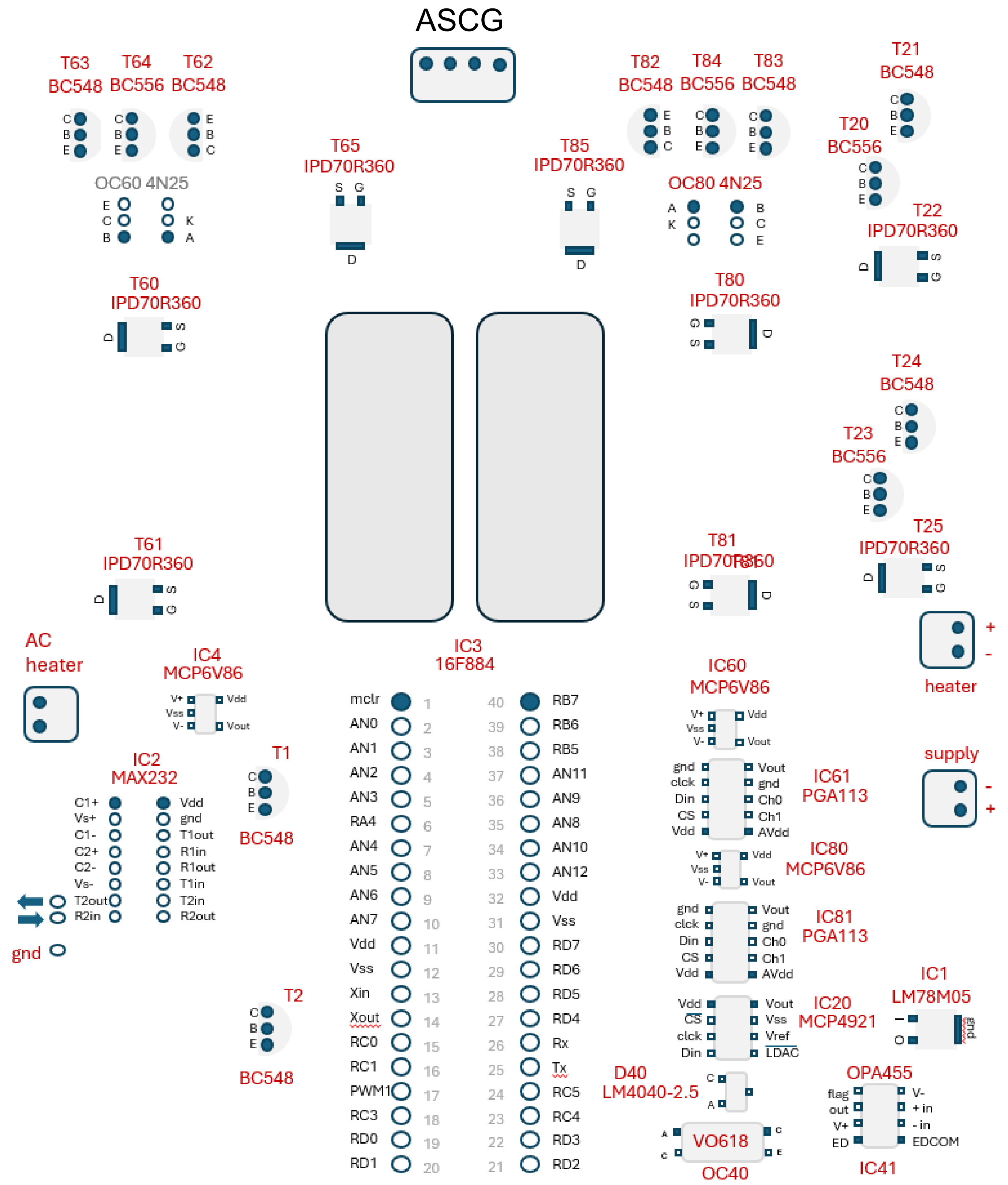
The wire connecting say pin one of each tube socket runs in a loop which also connects to the banana plug or rotary switch. At certain intervals, an RF suppression bead (e.g. Wuerth no: 74270015) is shifted over the wires. The physical lengths of all the loops must be approximately the same. The figure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

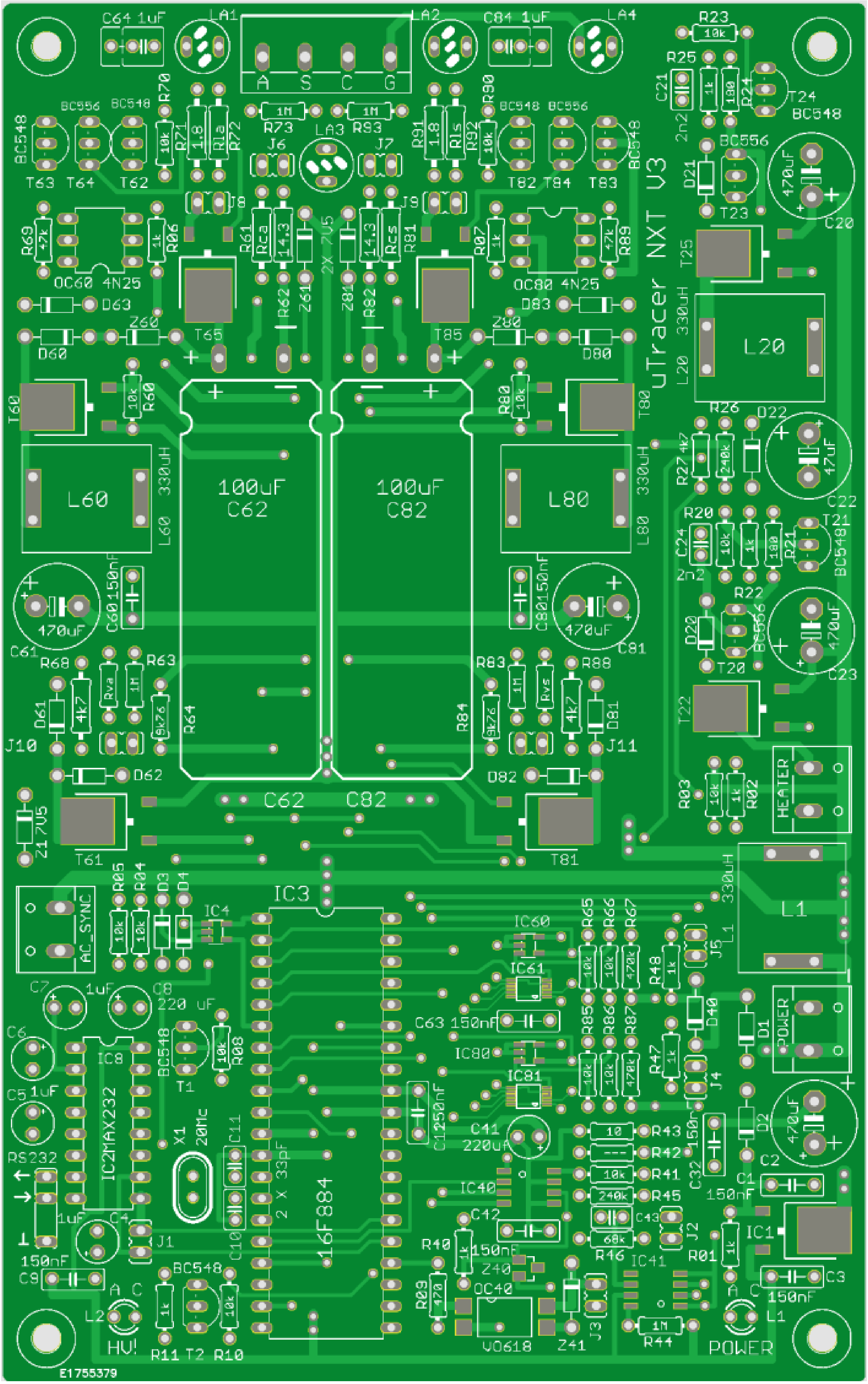




uTracerNXT V3
IC3 16F884
MCLR
AN0
AN1
AN2
AN3
RA4
AN4
AN5
AN6
AN7
Vdd
Vss
CLCI
CLCO
RC0
PWM2
PWM1
RC3
RD0
RD1
PGD
PGC
RB5
RB4
PGM
RB2
RB1
RB0
Vdd
Vss
RD7
RD6
RD5
RD4
Rx
Tx
RC5
RC4
RD3
RD2
+5V
C12 150 nF
33 pF C11
33 pF C10
X1 20 MHz
C16 150 nF
+19V
HV L2
R11 1k
R10 10k
T2 BC548
OC60 and OC80 are the LEDs in the high voltage switch opto-couplers
OC60 / OC80 = 4N25
OC80
R07 1k
OC60
R06 1k
R08 10k
T1 BC548
R09 470
OC40 VO618
C9 150nF
C8 220 uF
C7 1 uF
C4 1 uF
C6 1 uF
C5 1 uF
J1
IC2
MAX232 IC2
Vcc
V+
C1+
C1-
T1in
R1out
R2out
T2in
T1out
R1in
R2in
T2out
C2+
C2-
Gnd
V-


uTracerNXT V3
Anode HV switch
Screen HV switch
Anode section
Screen section
DHT heater synchronisation
Heater supply
-105V supply
+5V supply
IC41 OPA455 Grid supply
T60,T80,T61,T81,T65,T85 = IPD70R360P7
TUT
to heater supply
Vsupply (19.5 V)
IC40 MCP4921
IC61 PGA113
IC81 PGA113
IC60 MCP6V86U
IC80 MCP6V86U
IC4 MCP6V86U
IC1 78M05
AC heater synchronisation
2XUF4007
LM4040-2.5 Z40
R42 see text
FQD2P40TM
DMP3028


ASCG
T63 BC548
T64 BC556
T62 BC548
OC60 4N25
T65 IPD70R360
T85 IPD70R360
T82 BC548
T84 BC556
T83 BC548
T21 BC548
T20 BC556
OC80 4N25
T22 IPD70R360
T60 IPD70R360
T80 IPD70R360
T24 BC548
T23 BC556
T25 IPD70R360
T61 IPD70R360
T81 IPD70R360
AC heater
IC4 MCP6V86
IC3 16F884
IC60 MCP6V86
heater
supply
IC2 MAX232
T1 BC548
IC61 PGA113
IC80 MCP6V86
IC81 PGA113
IC1 LM78M05
IC20 MCP4921
T2 BC548
D40 LM4040-2.5
OPA455
VO618
OC40
IC41
gnd
C1+ Vs+ C1- C2+ C2- Vs- T2out R2in
Vdd gnd T1out R1in R1out T1in T2in R2out
mclr AN0 AN1 AN2 AN3 RA4 AN4 AN5 AN6 AN7 Vdd Vss Xin Xout RC0 RC1 PWM1 RC3 RD0 RD1
RB7 RB6 RB5 AN11 AN9 AN8 AN10 AN12 Vdd Vss RD7 RD6 RD5 RD4 Rx Tx RC5 RC4 RD3 RD2
gnd clck Din CS Vdd
Vout gnd Ch0 Ch1 AVdd
V+ Vss V- Vdd Vout
Vdd CS clck Din
Vout Vss Vref LDAC
flag out V+ ED
V- + in - in EDCOM


uTracer NXT V3
E1755379
16F884
IC3
IC2MAX232
100uF
C62
100uF
C82
L60
L80
L20
L1
HEATER
POWER
AC_SYNC
RS232
HV!
VO618
OC60 4N25
OC80 4N25