DPScope Assembly Guide

Version 1.2 (April 28, 2010)

Congratulations for purchasing the DPScope oscilloscope kit! This guide will lead you through all the steps required to put it together so you'll be measuring signals in no time.

While we made every effort to create a kit that is robust and simple to assemble, there are some components that are sensitive to mishandling, e.g. putting them in with wrong polarity, so please pay close attention to the description for every step. Moderate soldering experience is required.

You will need a few tools for the assembly:

- Small soldering iron (about 17 Watts power) with sufficiently fine tip
- Solder wire
- Flat-nosed pliers (to bend component leads)
- Small wire cutter (to cut off component legs)
- 14mm wrench (to fasten the BNC connectors to the front panel)
- Small Philips screwdriver (to adjust the trimmer resistors and for the enclosure
- Small non-metal screwdriver (to adjust the trimmer capacitors)

The time required for putting the scope together will depend on your experience – a seasoned hobbyist should be able to do it in around one hour, but if you are new to this it will take longer.

If you have any questions or feedback, do not hesitate to contact us:

Email: support@dpscope.com
Webpage: http://www.dpscope.com

Step 1: Unpacking the Kit

Below you see the kit as it comes out of the box:

- Probe cables
- Enclosure, which holds all the small components
- FTDI USB cable

Unscrew the two screws on the bottom of the enclosure and *carefully* lift the bottom – make sure not to drop any components.

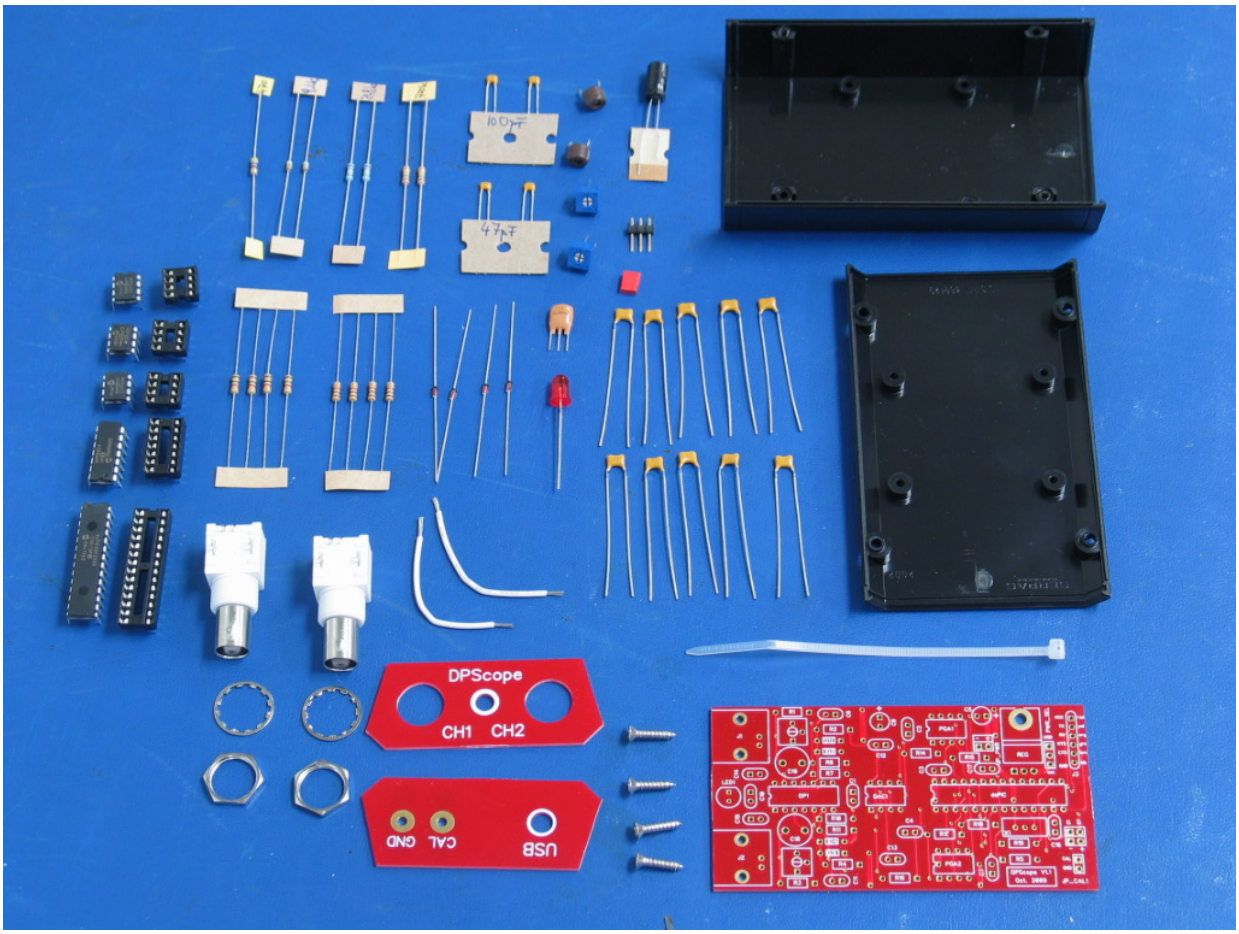
We don’t need the probes and the USB cable right now, so put them to the side for the moment.

Step 2: Components

Below you see all the components spread out. Everything you need to build the oscilloscope is included. Please use the component list (on the next page) to verify that you have all the parts shown below.

Some general remarks:

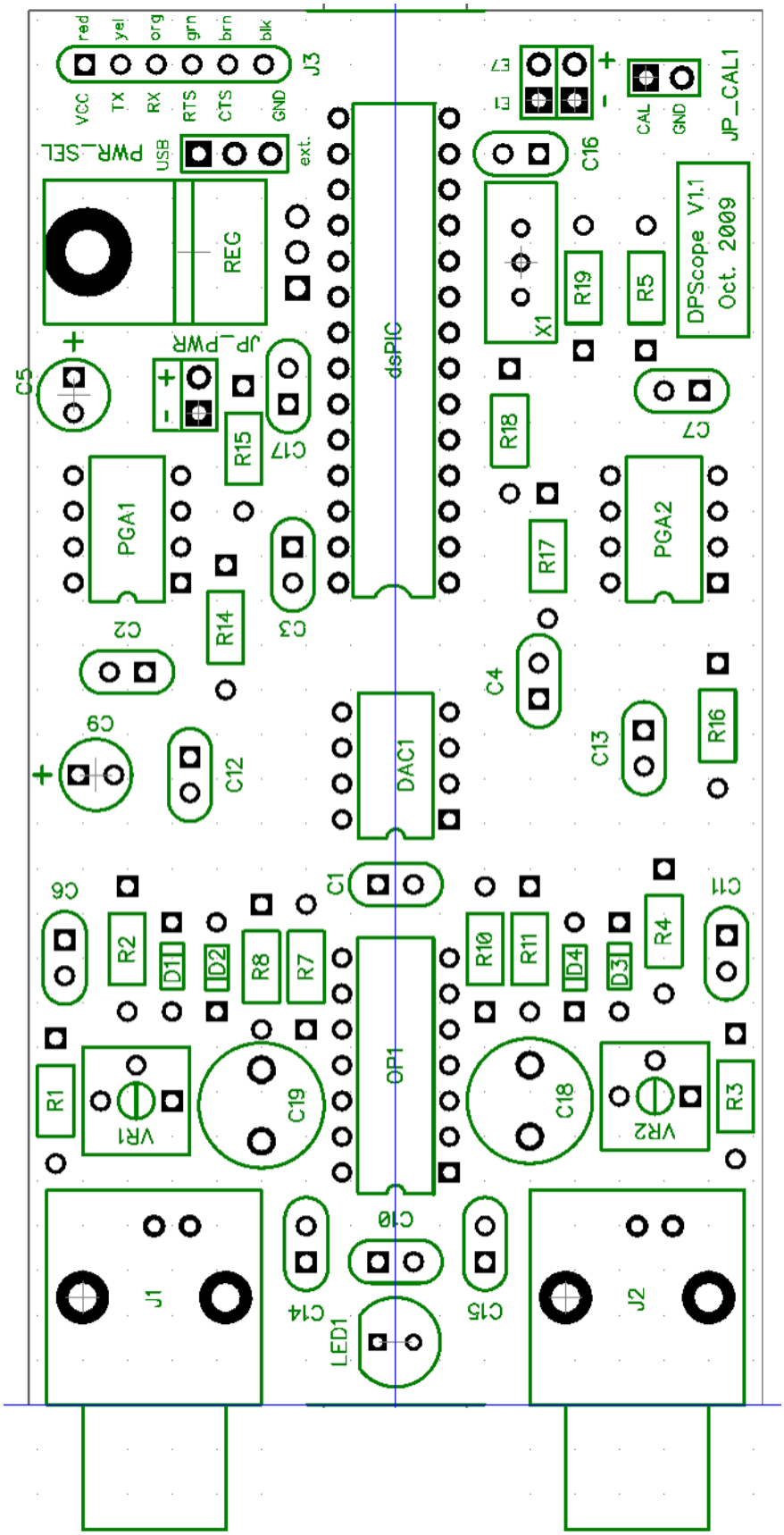
- All components that are difficult to distinguish (e.g. resistors) are clearly labeled with their respective value (e.g. “100 pF”).
- The integrated circuits and the diode are sensitive to electrostatic discharge – it is good practice to use a grounded wrist strap to avoid damage to them during assembly, and to place all the components on an antistatic surface. Don’t wear clothes that get easily charged up (e.g. wool sweater).



Step 3: Component List

RefDes	Description	Value	Quantity
R19	resistor, 0.25W	470	1
R5, R8, R11, R14, R15, R16, R17, R18	resistor, 0.25W	1k	8
R7, R10	resistor, 0.25W	9.1k	2
R2, R4	resistor, 0.25W	249k	2
R1, R3	resistor, 0.25W	750k	2
VR1, VR2	trimmer, 0.25W	500k	2
C6, C11,	ceramic capacitor	47p	2
C14, C15	ceramic capacitor	100p	2
C1, C2, C3, C4, C7, C10, C12, C13, C16, C17	ceramic capacitor	0.1u	10
C9	electrolytic capacitor	100u	1
C18, C19	trimmer capacitor	10...60p	2
D1, D2, D3, D4	diode	1N914A	4
X1	ceramic resonator, 3 pin	16 MHz	1
LED1	LED	red, 1 3/4	1
DAC1, PGA1, PGA2	DIP socket	8 pin	3
OP1	DIP socket	14 pin	1
dsPIC	DIP socket	28 pin, 0.3"	1
DAC1	Integrated Circuit	MCP4822	1
OP1	Integrated Circuit	MCP6024	1
PGA1, PGA2	Integrated Circuit	MCP6S22	2
dsPIC	Integrated Circuit	dsPIC30F2020	1
	Terminal turret for Cal & GND		2
PWR_SEL	3-way jumper		1
	Shorting block		1
J1, J2	BNC connector		2
	Nut & washer for BNC connector		2
	Enclosure		1
	Screws for enclosure		4
	Printed circuit board		1
	Front panel		1
	Back Panel		1
	Hookup wire		2
	Cable binder		1
	USB-to-serial converter cable		1
	BNC-to-Microhook probe cables		2

Step 4: Component Placement

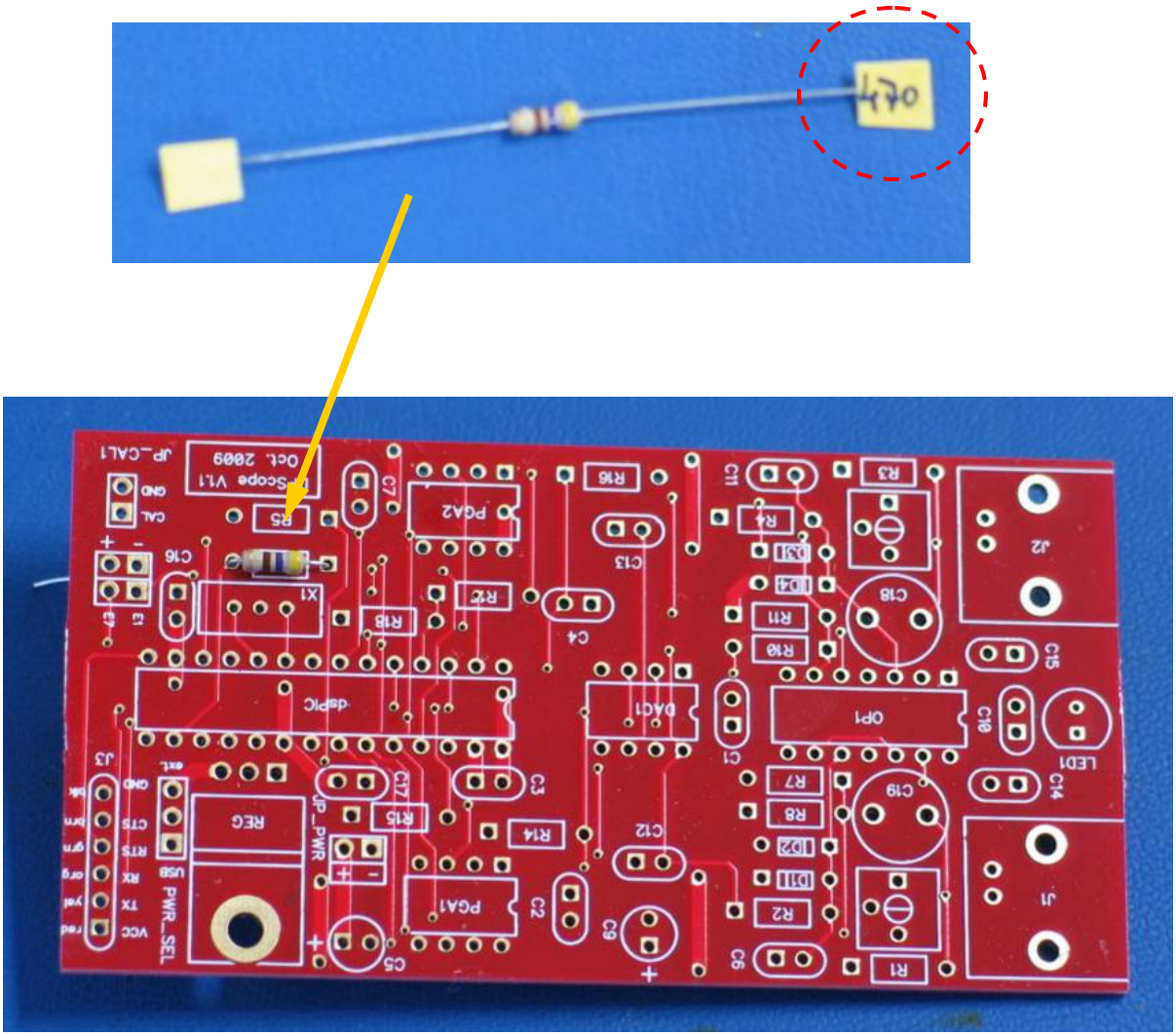


Step 5: Installing the First Part – R19

Let's start out with a simple part – resistor R19, which has 470 Ω. All the other parts will get assembled in a very similar way.

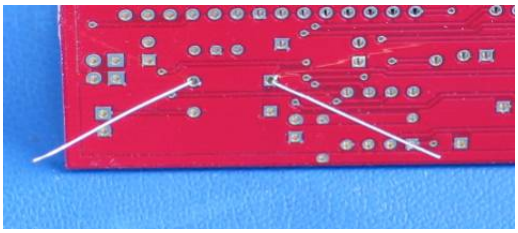
Find the resistor among all the parts – it is labeled with "470" (see picture below).

On the printed circuit board (PCB) locate the component outline labeled "R19". Bend the resistor leads and stick them through the board. The final result is shown below.

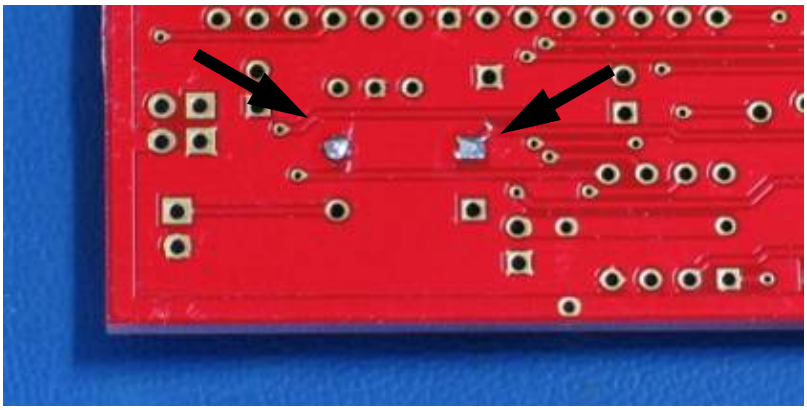


Step 6: Soldering the First Part

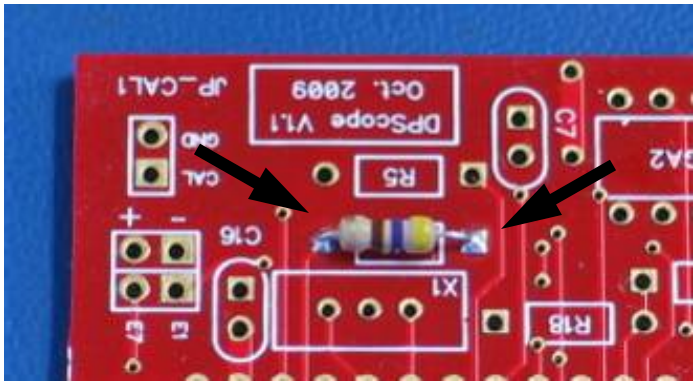
On the PCB bottom bend the component leads apart – this will hold the resistor securely when you turn over the PCB to solder it on:



To solder, touch the component lead (wire) with the soldering iron and the solder wire at the same time. Add enough solder so it completely fills the hole and leaves a small “hill” of solder. Remove the solder wire and the soldering iron but don’t move the board before the solder has become completely solid again. The solder joint should be shiny and bright. Repeat for the other lead.

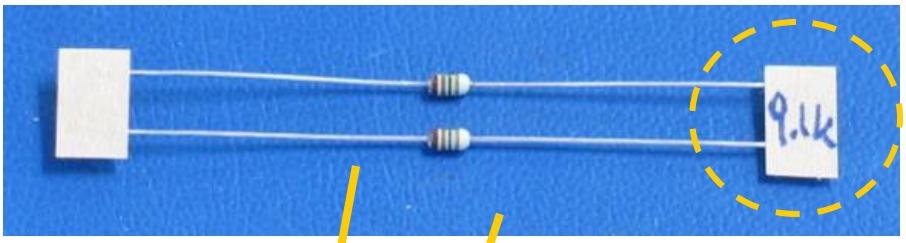


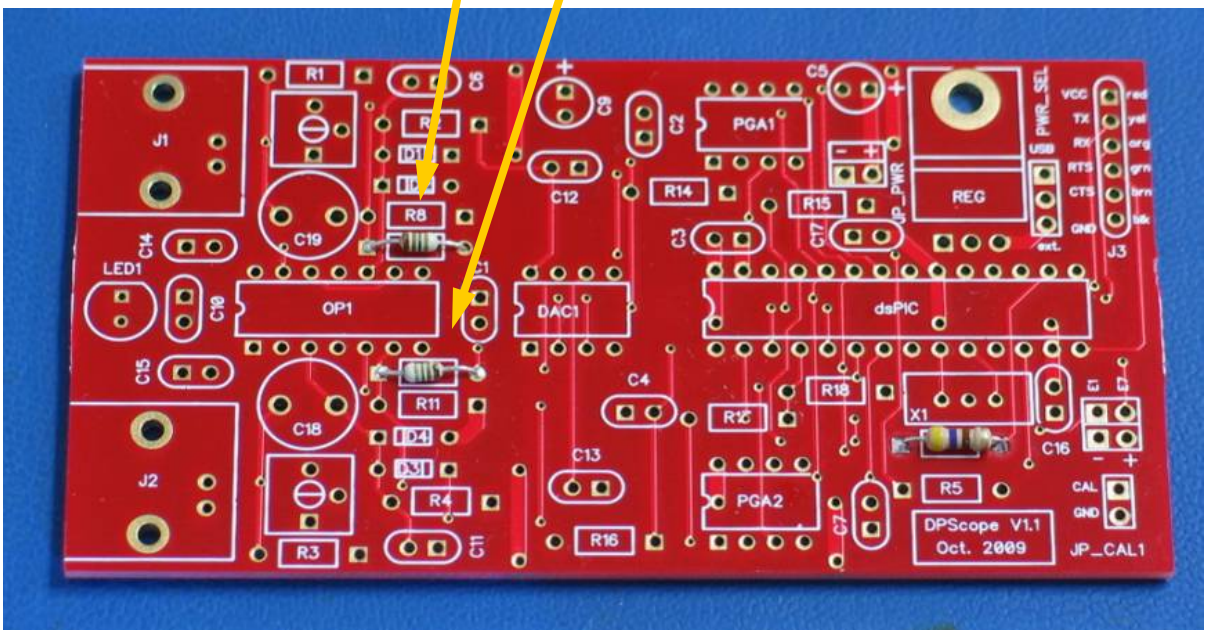
Check the topside of the board – you should see solder protruding a bit from the via holes – this shows the holes are nicely filled with solder (see picture below):



Step 7: Next Components

Next is a pair of resistors – R7 and R10 (the 9.1 kΩ resistors, thus labeled with “9.1k”). Installation and soldering works just like for R19. Below you see what the board looks like with them installed.

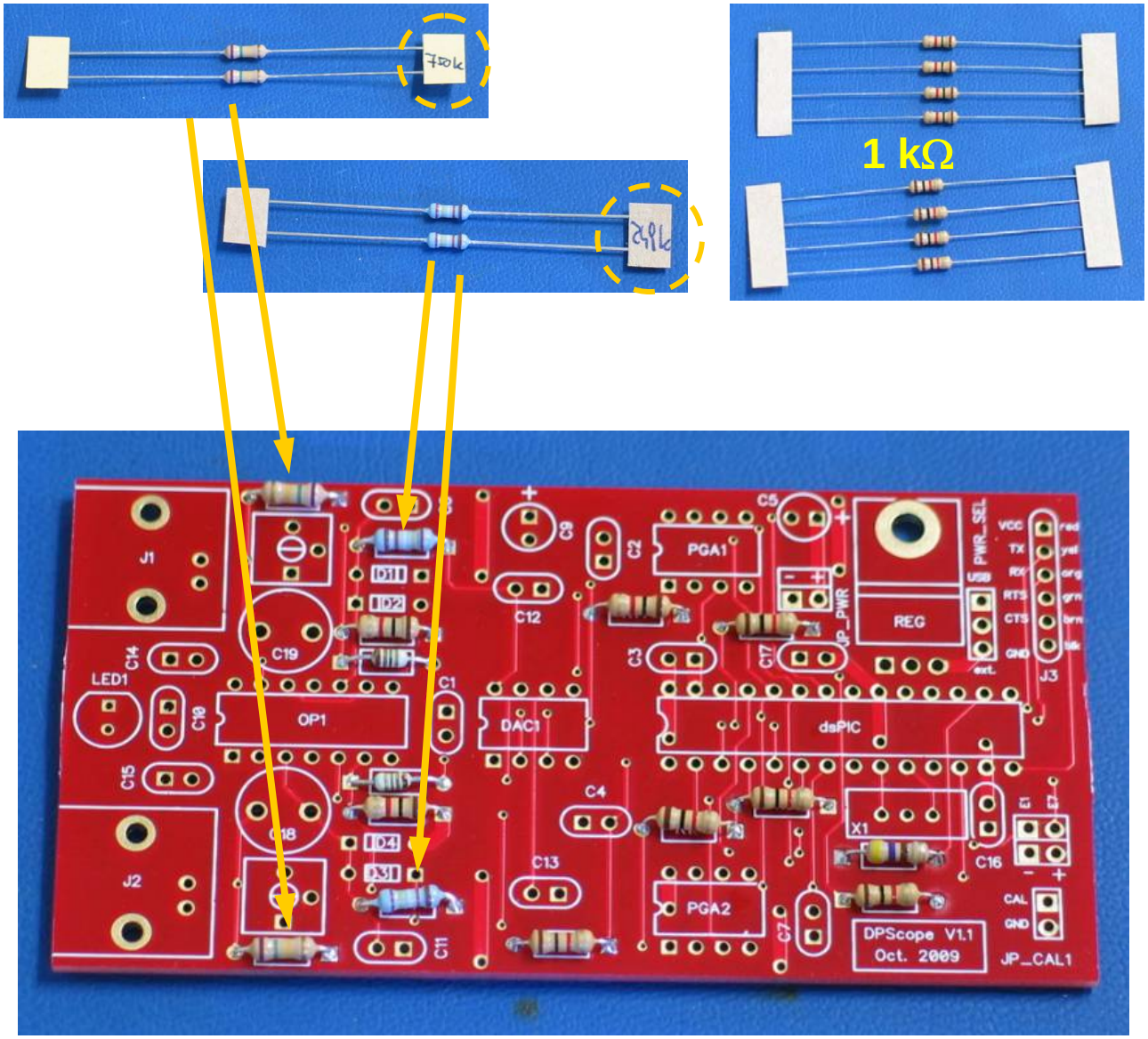




Step 8: Remaining Resistors

Now lets finish up the resistors. We still got R2 and R4 (249 kΩ), R1 and R3 (750 kΩ), and 8 pieces Of 1 kΩ resistors which are not labeled because they are the last resistors left. Install the two pairs first, and then install the 1 kΩ resistors in all remaining resistor outlines on the PCB (refer to the parts list if find out which resistor is which value).

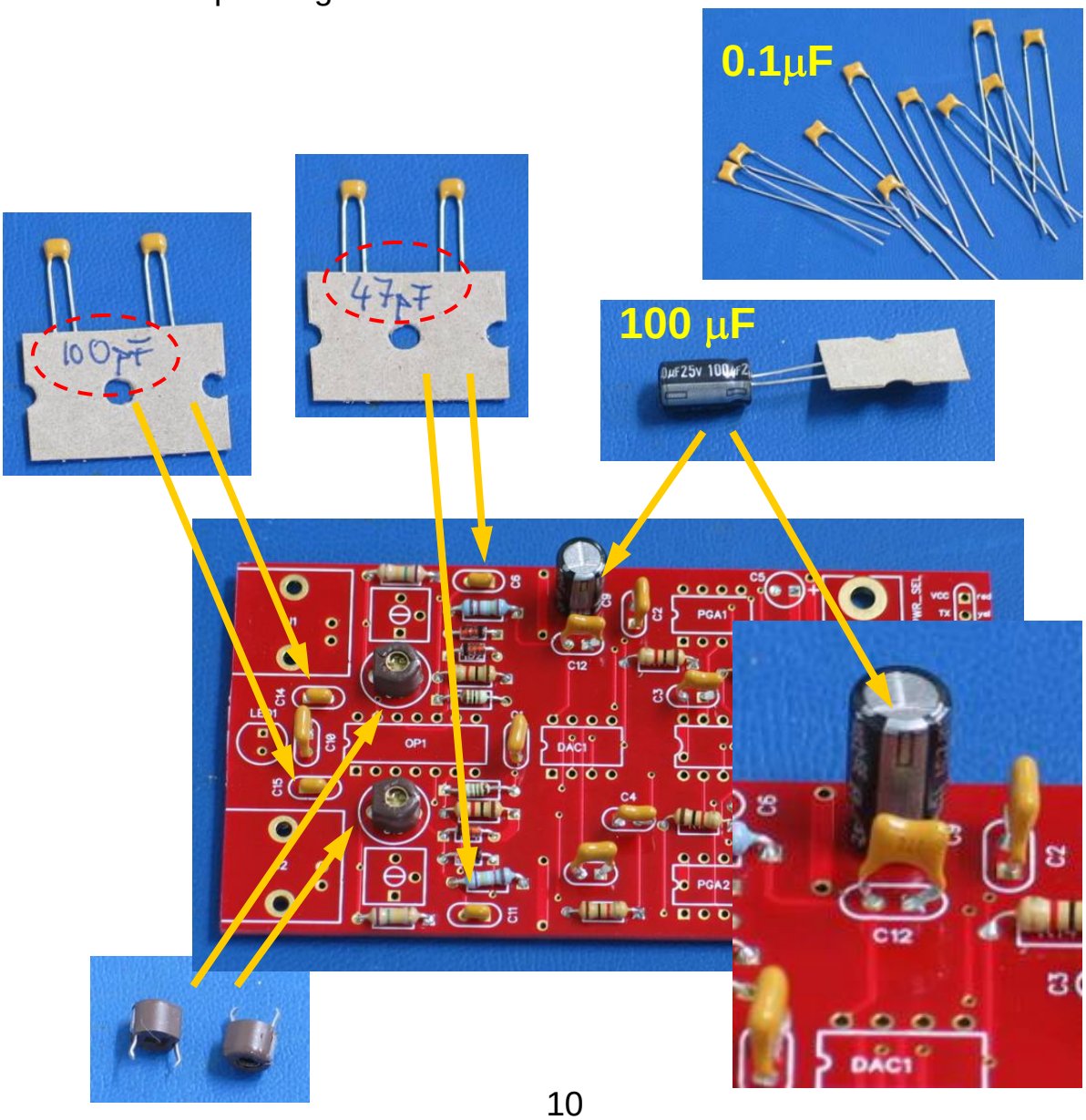
The picture on the bottom shows the PCB populated with all the resistors.



Step 9: Capacitors

Installing the capacitors works the same as for the resistors. We have 5 different types: C6 and C11 (ceramic, 47 pF), C14 and C15 (ceramic, 100 pF), C9 (electrolytic, 100 µF), C18 and C19 (trimmer), and 10 pieces of 0.1µF cermic capacitors (not labeled because they are the only type left).

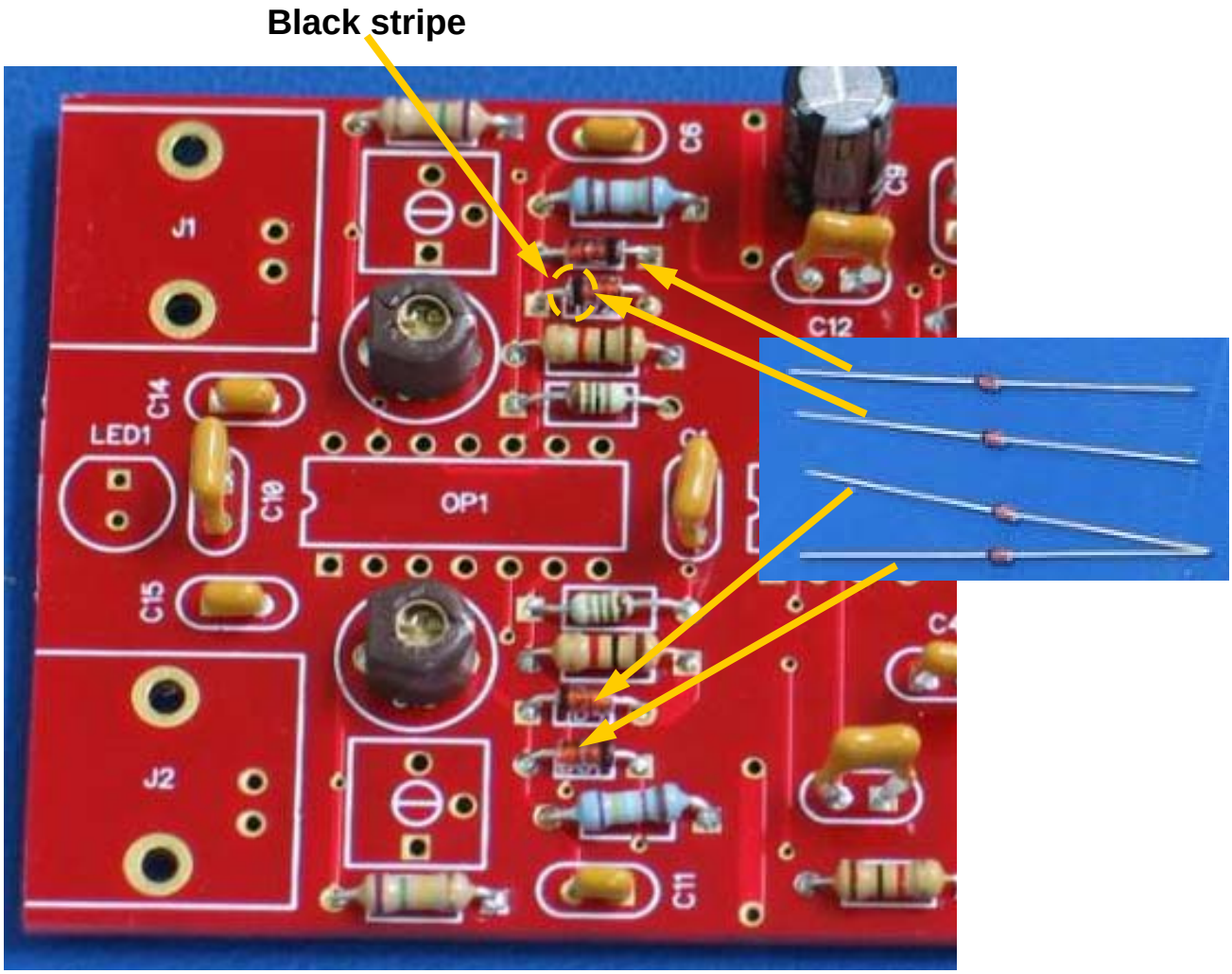
Note: Be careful when installing C9 – it has to be put in with correct polarity. The negative side of C9 is clearly labeled with a white stripe and "-" (minus) symbols – make sure you install it as shown in the pictures below with this white strip facing inwards on the PCB!



Step 10: Diodes

Next in the row are the clamping diodes. These components need more care than the resistors.

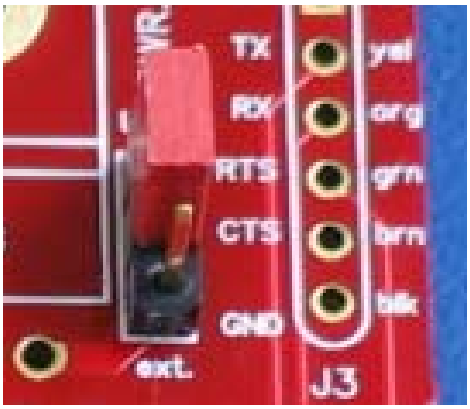
- Make sure you insert them with correct polarity (correct orientation); the negative end is denoted by a **black stripe** around the diode body. The silkscreen outline also shows a (white) stripe – this is the side the black stripe must lie. Note that the orientation is *not* the same for all diodes.
- Diodes are quite sensitive to heat. Thus try to minimize soldering time. The best approach is to first solder only one end of all diodes, then the other end – this gives them enough time to cool down in the time between.



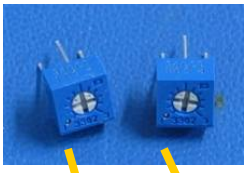
Step 11: Trimmers, Resonator, Jumper

A few more small components are left: The two trimmers (VR1 and VR2, the blue blocks with the adjustment knob on the top), the ceramic resonator (X1), and the power selection jumper (PWR_SEL). Install them as shown below and **put the red shorting block on the jumper in the position indicated in the zoomed-in picture (labeled “USB”)**; this jumper is used to select the optional external power supply & voltage regulator as the scope’s power source; per default these are not installed because the instrument gets its power through the USB connection.

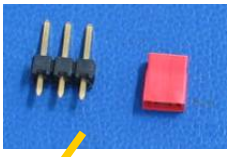
Correct jumper position:

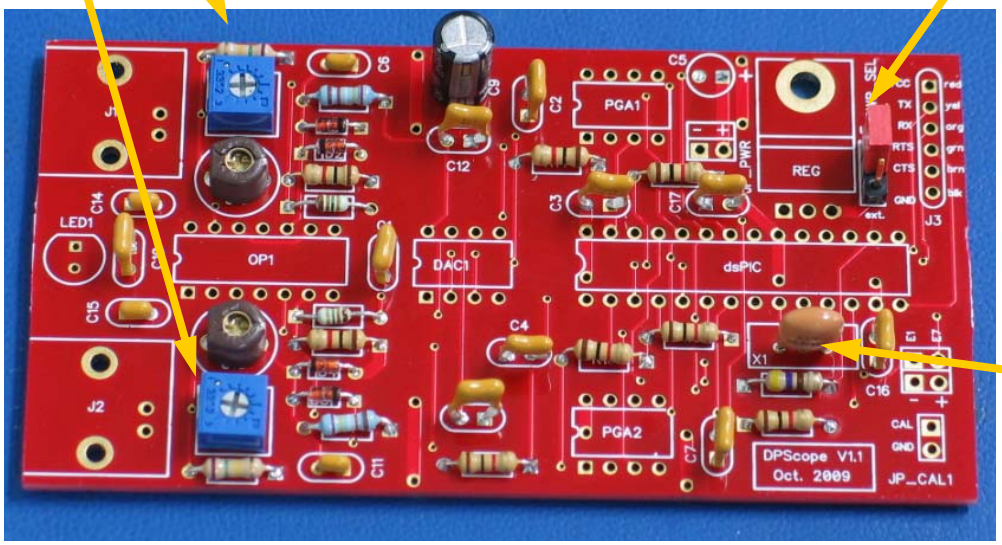


trimmers



power selection jumper



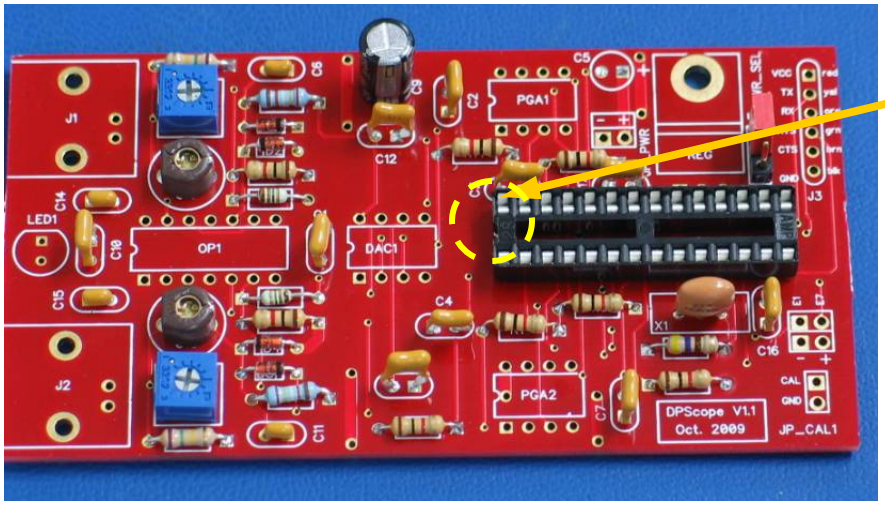


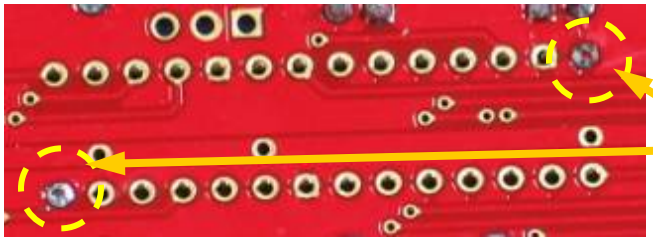
resonator

Step 12: First Socket

Now comes the first socket – lets start with the 28-pin one, which goes into the outline labeled “dsPIC” and will later hold the microcontroller. Be careful with its installation because once soldered down it is almost impossible to remove. The best is to do it step by step:

1. Place the socket on the board as shown in the big picture. Note the **position of the notch** on the left side of the silkscreen outline – make sure to orient the notch on the socket to the same side. This will make installing the chip less error-prone.
2. Turn the board around an **solder only two of the corner pins** as shown in the zoomed-in picture below. The reasons is simple – two pins diagonally opposed will securely hold the socket in place, but still allow you to make corrections.
3. Press the socket onto the board and re-heat both of these solder joint – this allows the socket to sit flush against the board. Visually inspect the socket to make sure this is really the case.
4. Only now solder all the other pins.

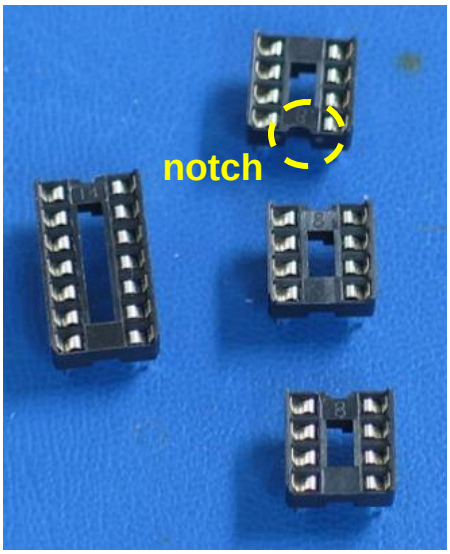


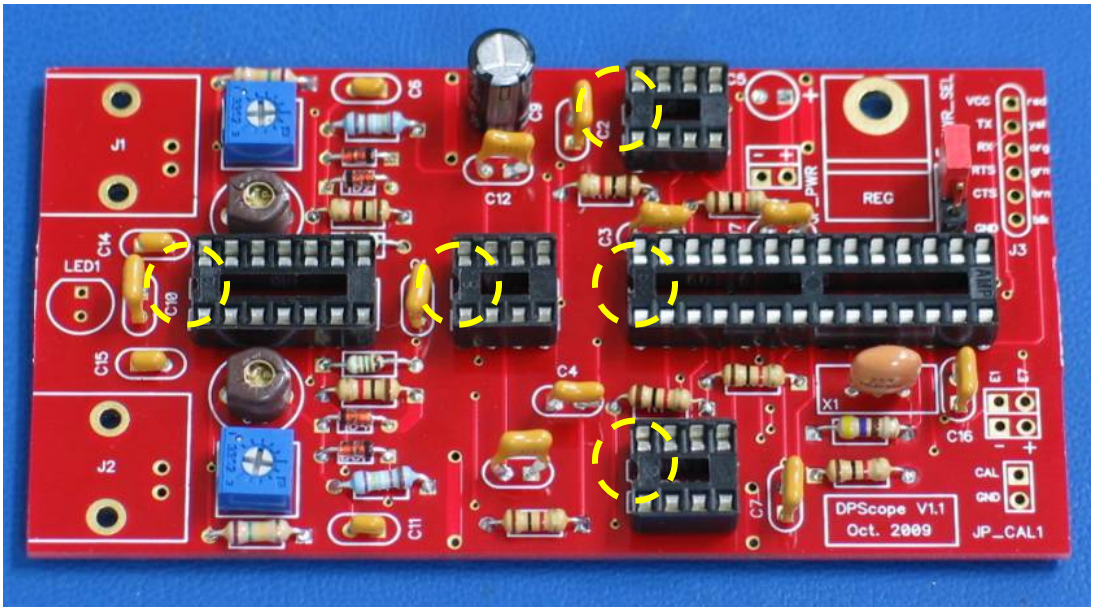


Step 13: Remaining Sockets

Now install the remaining sockets – one 14-pin and three 8-pin ones. Proceed in the same manner as for the first socket. **Again make sure all the notches match the silkscreen outline.** (they are all on the left side in the picture.

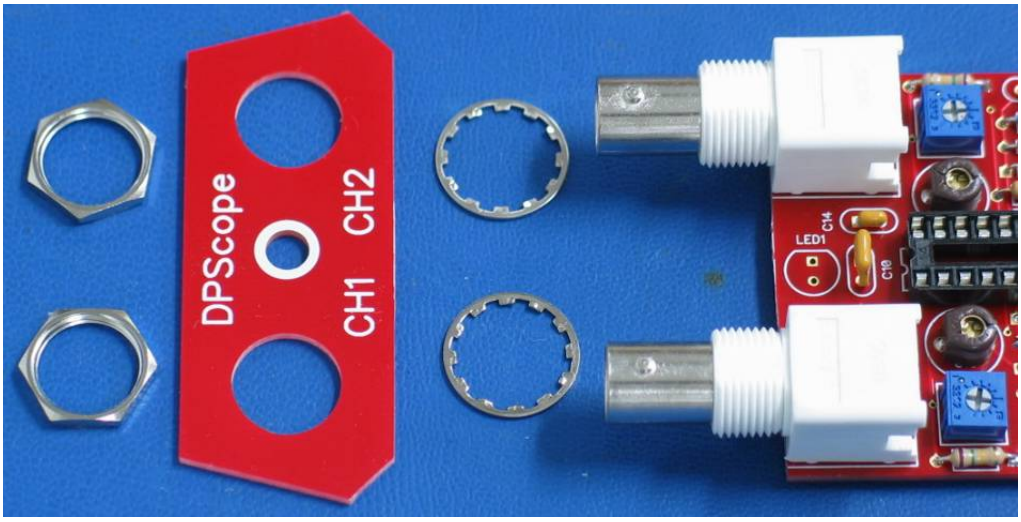
Below you see how the board should look like after this step.



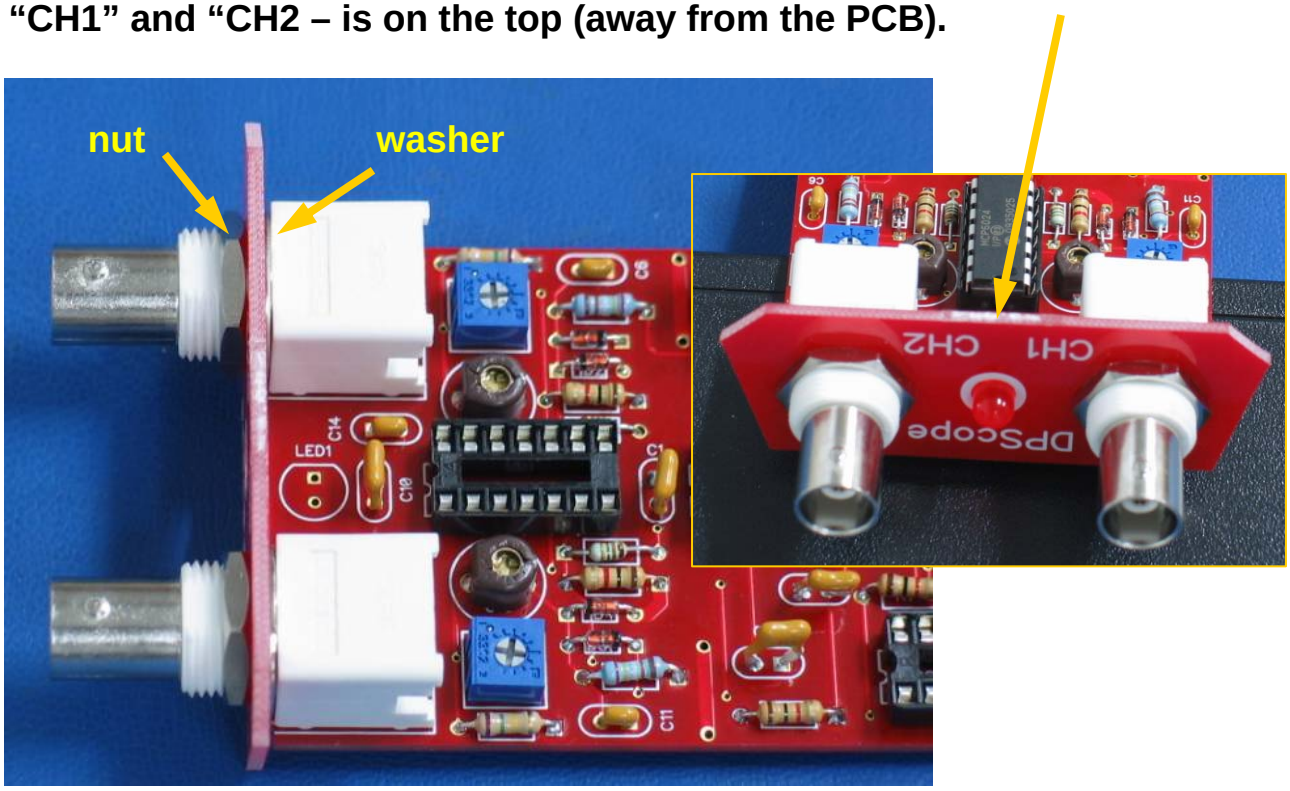


Step 14: BNC Connectors; Frontpanel

Snap the BNC connectors in place. **Don't solder them down yet!**



Add the washers (if included), the frontpanel, and finally the nuts. Tighten the nuts by hand (not too strongly, just so the frontpanel no longer moves freely). **Note that the frontpanel's narrow end – close to the "DPScope" label" – is on the bottom (PCB) side, and the wide end – close to labels "CH1" and "CH2 – is on the top (away from the PCB).**



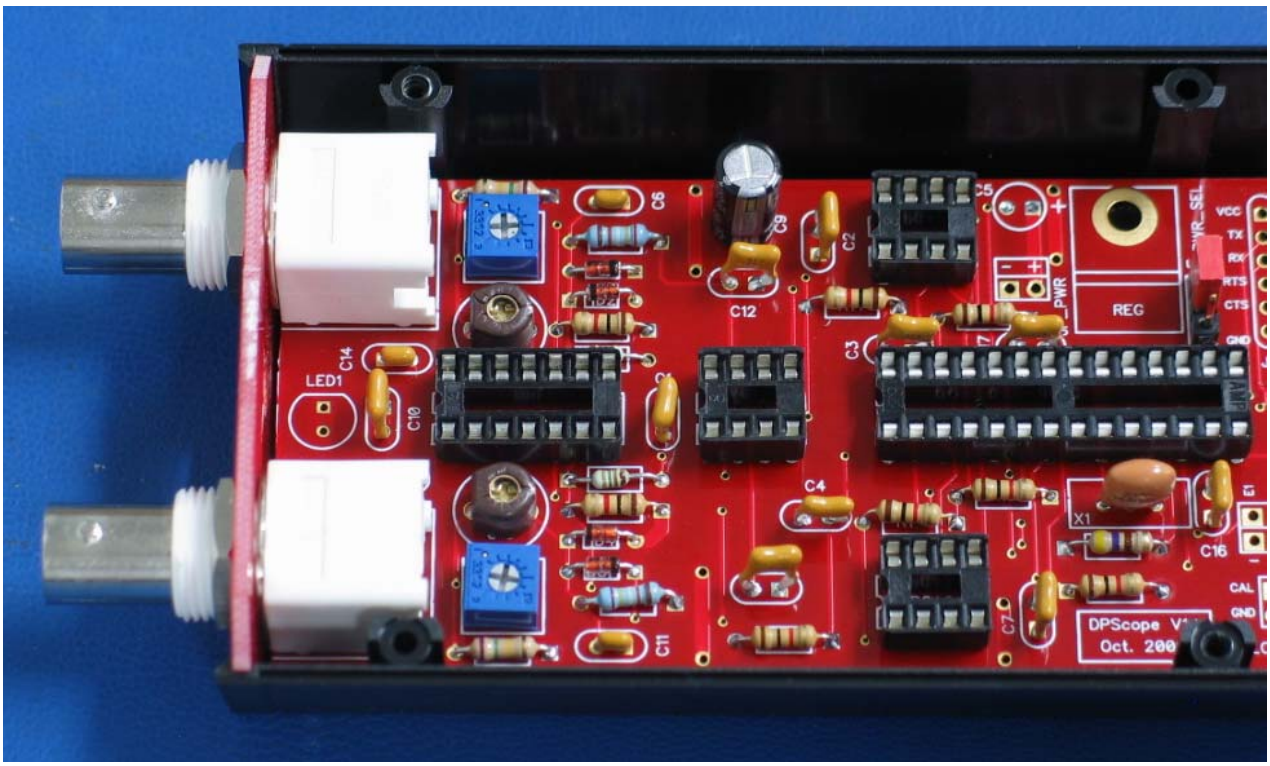
Step 14: BNC Connectors; Frontpanel

Place the board with BNC connectors and frontpanel into the enclosure as shown (into the deep half of the enclosure).

Make sure the board fits and sits loosely on – but does not push too hard against – the standoffs inside the enclosure. If necessary loosen the nuts a bit so the connectors can move against the frontpanel

Tighten the nuts again sufficiently to hold the BNC connectors in place.

Only now solder the BNC connectors onto the board – start with the clamped-in feet, and finish with the signal wires. The clamped feet will need a lot of solder to fully fill up the mounting holes – don't be shy, because that's the only thing holding the PCB in place.

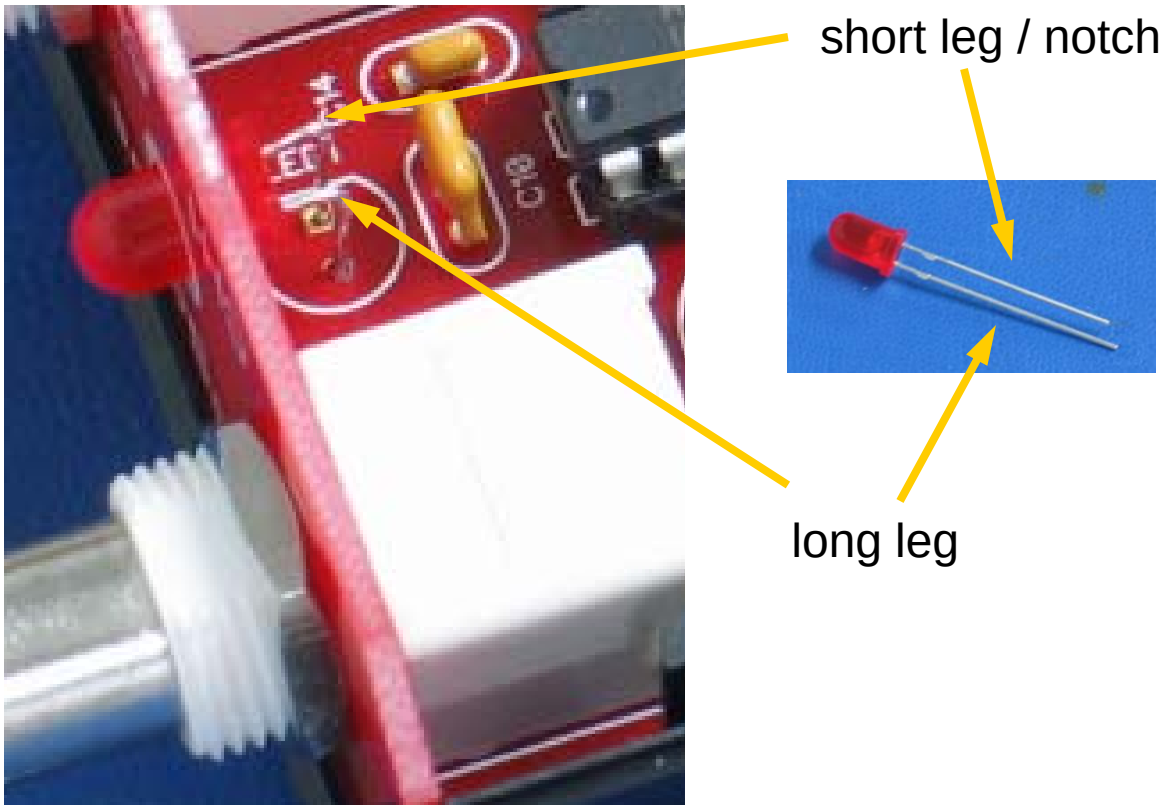


Step 15: LED Indicator

Remove the frontpanel again. Take the LED (light emitting diode) and bend its legs by 90 degrees as shown in the picture. Fit it into the board. **Make sure the short leg of the LED goes into the side where the silkscreen circle has its notch (flattened side).** The diode body also has a notch at the same side.

Now put on the frontpanel – the LED must go through the center hole. Put on the nuts again and tighten them down with a wrench. **Be careful not to over-tighten them, damaging the connectors threads!**

The frontpanel now holds the LED securely in place. Turn the board around and solder the LED's legs onto the board.

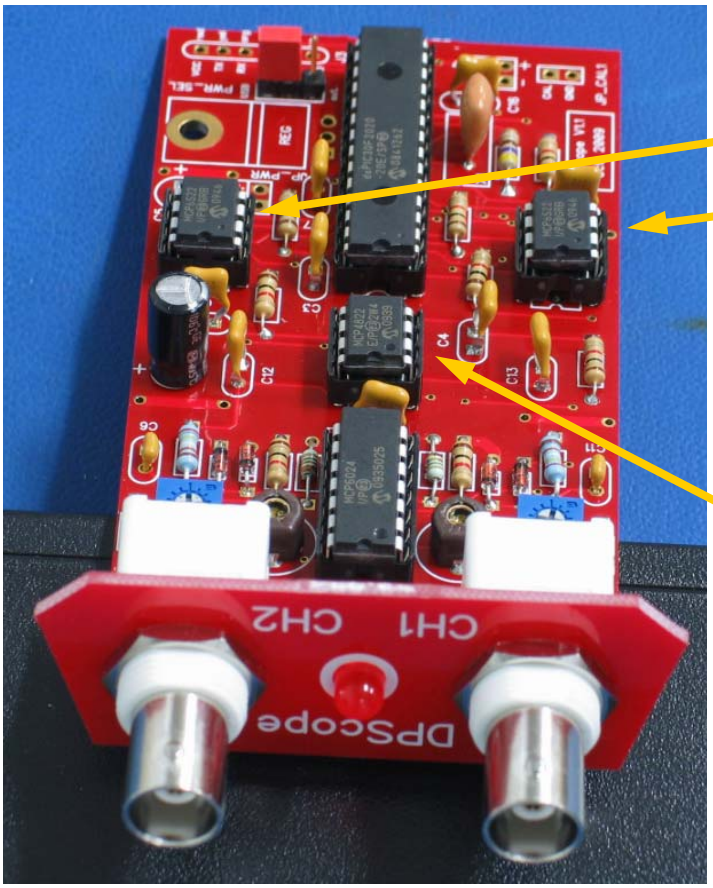


Step 16: IC Installation

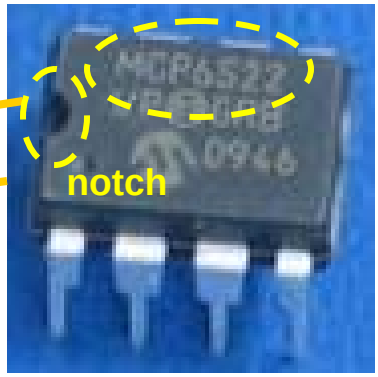
Now it's time to install the ICs (integrated circuits). There are five of them.

Note 1: Make sure to put the chips on in the correct orientation. Each chip has a notch on one end – this notch must go on top of the notch in the silkscreen outline (and the notch in the socket if you installed those correctly!).

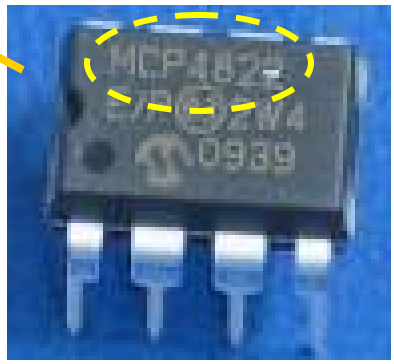
Note 2: There are two different types of the 8-pin ICs – make sure you install them in the correct locations as shown in the picture below. Installing them in the wrong place (or the wrong orientation) will destroy them when you power up the oscilloscope. You can distinguish the chips by the labels printed on them (two are labeled MCP6S22, one is labeled MCP4822).



MCP6S22



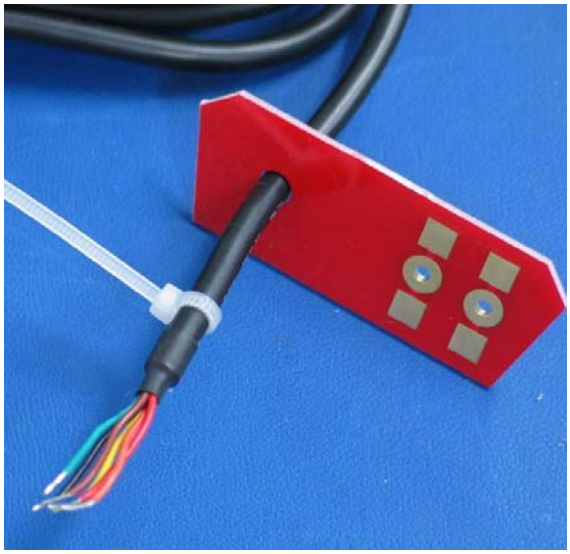
MCP4822



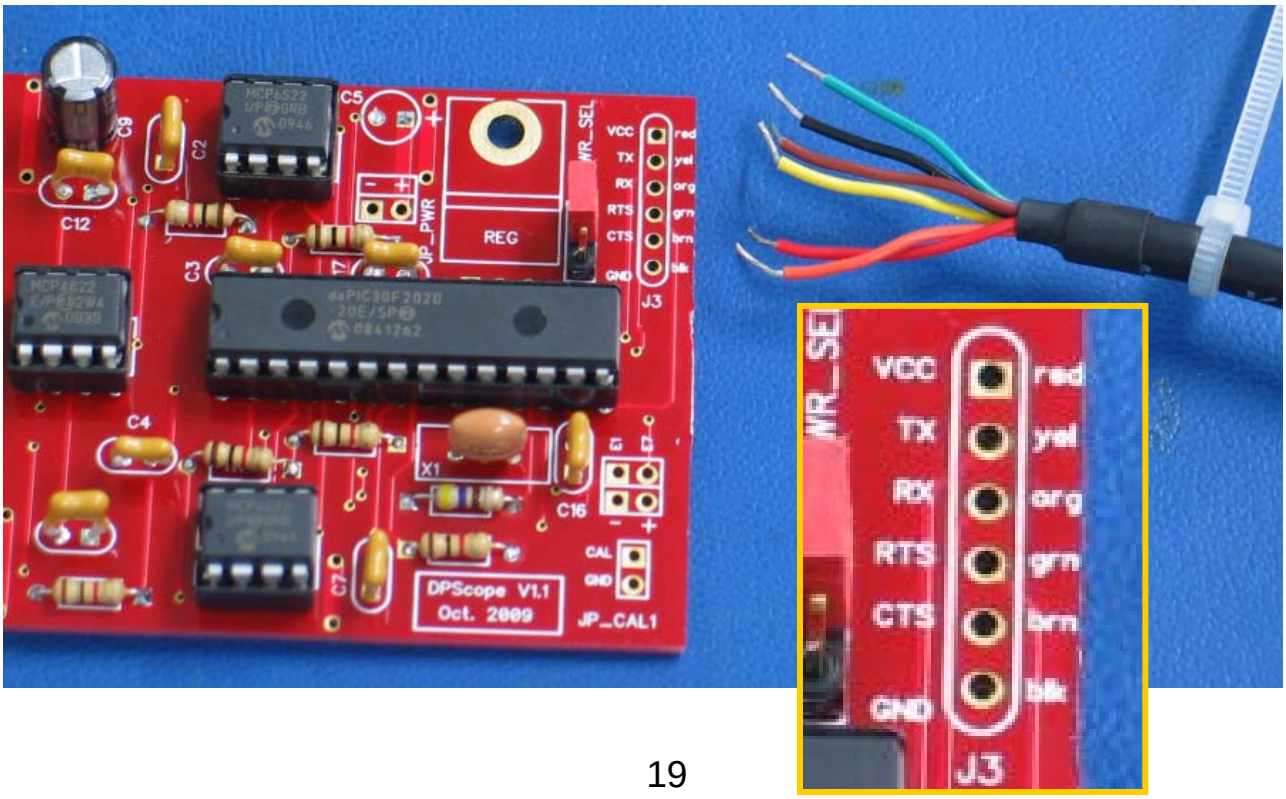
Step 17: USB Cable

Take the USB cable out of its antistatic bag and feed its wire ends through the hole in the backpanel plate. Secure the cable binder tightly around the cable – this will act as a stress relief so the cable can't pull on the solder joints later. Snap off the protruding part of the cable binder.

The cable has 6 wires in 6 different colors. Solder them into the respective hole of J3 (all holes have labels indicating the proper color, as shown in the zoomed insert). The sequence from top to bottom is:



Red (red)
Yellow (yel)
Orange (org)
Green (grn)
Brown (brn)
Black (blk)

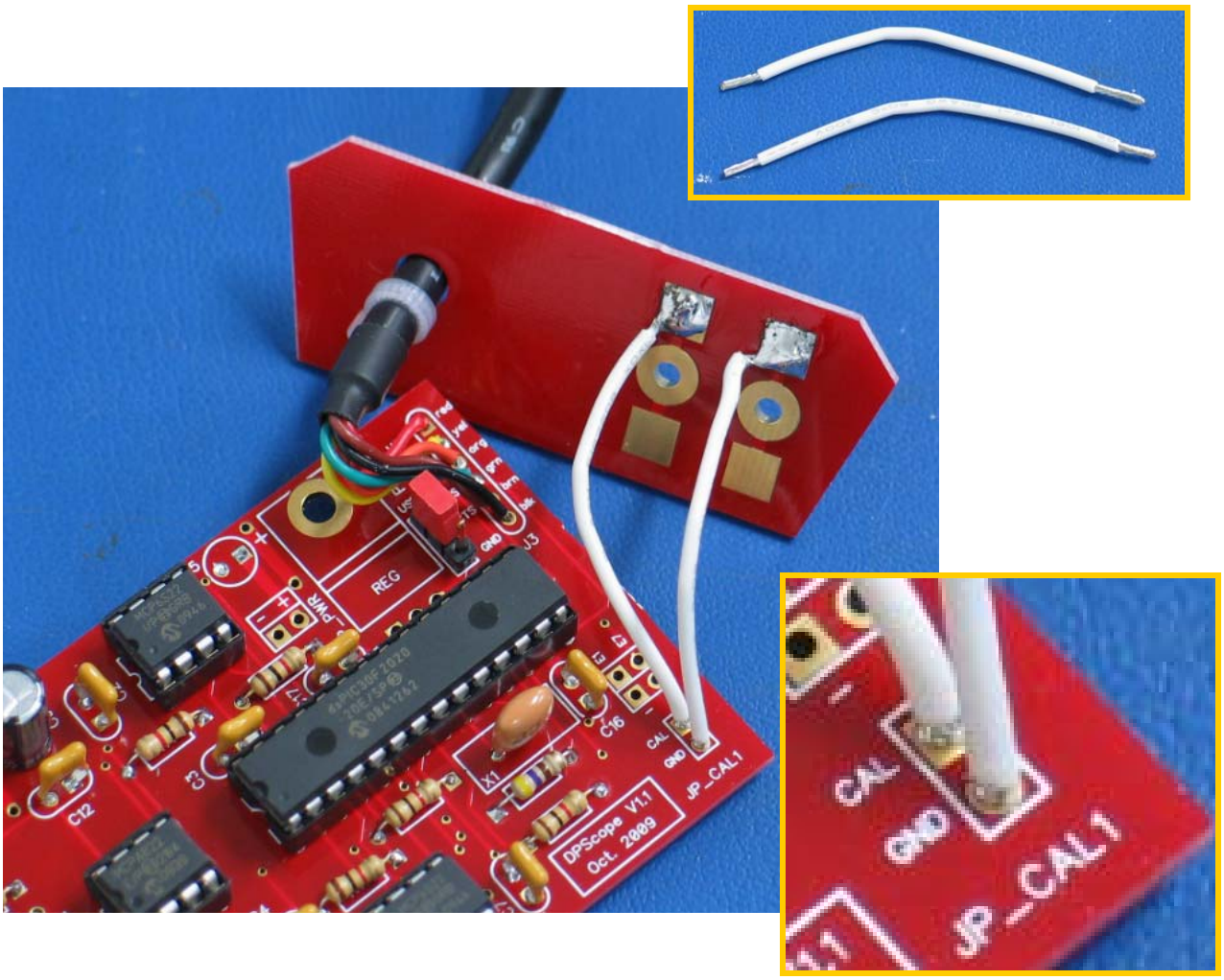


Step 18: Probe Calibration Output

Take the two hookup cables and solder them into the board and onto the solder pads on the backpanel as shown.

Make sure to connect the hole labeled “CAL” with the backpanel pad going to the “CAL” hole on the backpanel, and the same for “GND”.

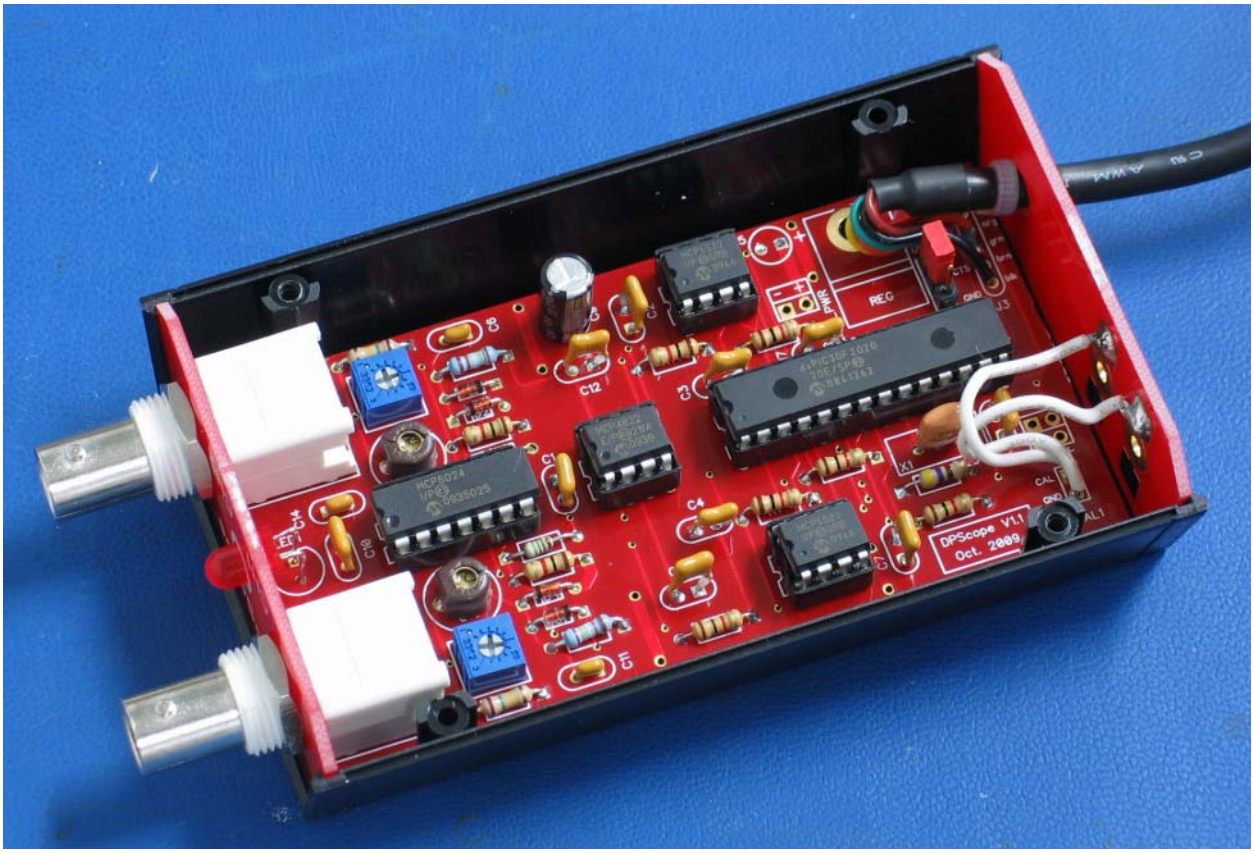
Put the two terminal turrets (not shown) into the holes and solder them on. The long part of the turret points to the outside of the backpanel.



Step 19: Putting It All Together

Now put the board back into the enclosure box, and snap in frontpanel and backpanel.

Congratulations – your oscilloscope is fully assembled!



Step 20: Software Installation

If you haven't already done so, go to the DPScope website (http://www.dpscope.com) and download:

- The oscilloscope software
- The FTDI USB driver
- The USB driver installation guide

If you procured the USB cable yourself then you will also need to burn the USB configuration file onto the converter cable. (If the cable came as part of the kit or the fully assembled scope then the cable is already configured and you do not need to do this). In this case also download

- The PROG configuration program
- The USB configuration file

First install the USB driver – follow the installation guide for that. Note that the installer will actually install two different drivers on your computer, i.e. go through two installation cycles. Make sure to complete both of them.

Then install the DPScope software as well (unpack the files and click on Setup.exe).

After that, if necessary burn the USB configuration file into the cable – as described on the following page.

Step 20: Software Installation

Note: If your USB cable came with the kit or the scope, skip this step.

1. Remove the jumper on the J_PWR selector. This will prevent the scope from drawing excessive current before the USB cable has been configured.

2. Plug the USB converter cable into a free USB port on your computer.

3. Start the MPROG programming software. You should get a message "Found New Hardware", followed after a while by a message "Your new hardware is installed and ready to use" (or similar).

4. Remove any other peripherals (except for mouse) attached to a USB port.

5. Start the MPROG configuration software (see next page for screenshot).

6. Select **Device → Scan**. After a short moment it should report 0 (zero) blank devices and 1 (one) programmed device (which is the USB cable that has a default factory configuration).

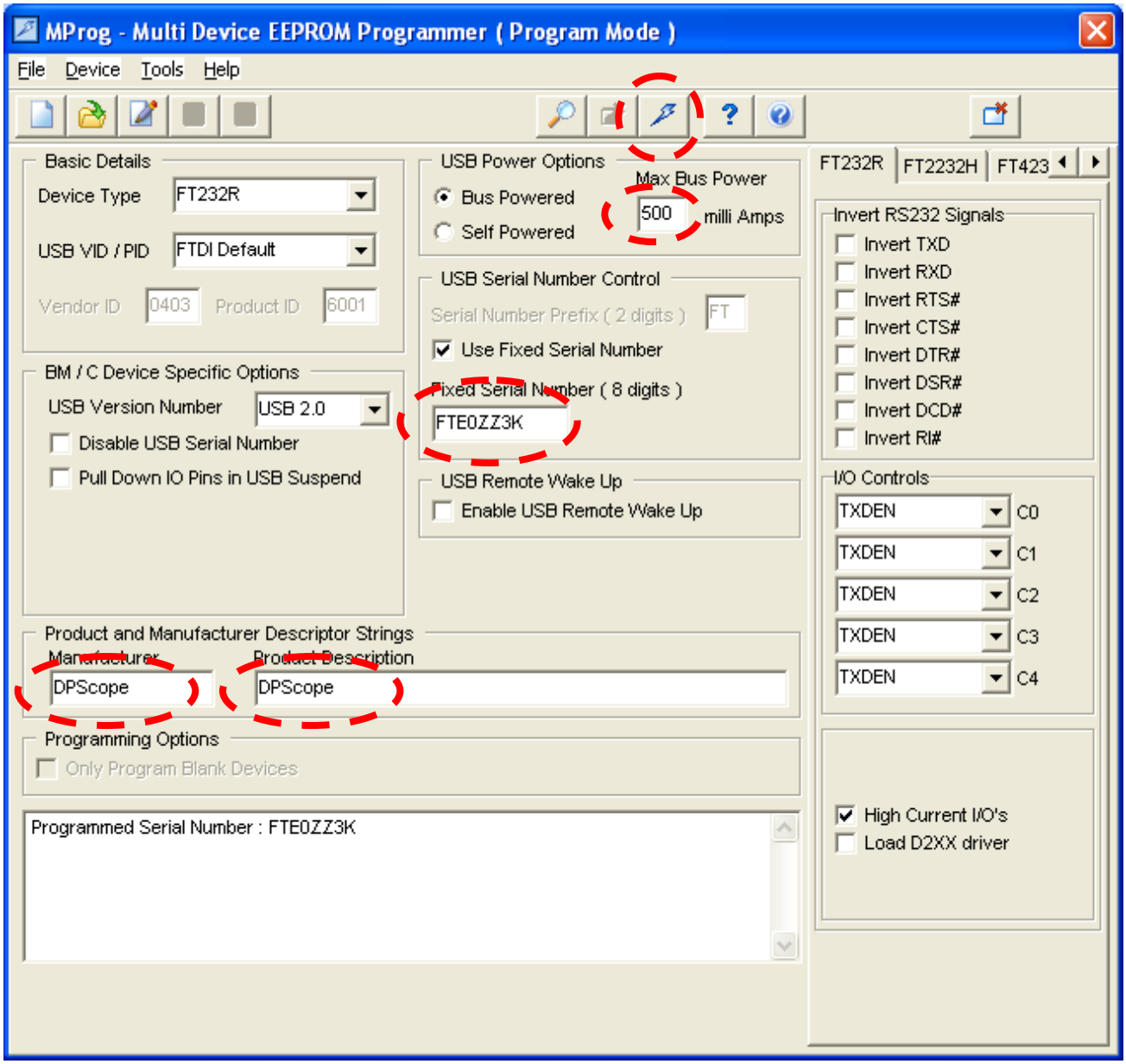
7. Open the DPScope USB configuration file that you just downloaded before (**File → Open**). The configuration data should now show a fixed serial number "FTE0ZZ3K" (the DPScope software recognizes the attached DPScope based on this serial number, so don't change it!), and manufacturer and product description string should both say "DPScope".

8. Press Ctrl-P or click on "**Program All Existing Devices**" (the menu button with the Flash symbol).

9. After a short while the status display on the bottom should now say "Programmed serial number FTE0ZZ3K".

10. Close MPROG. Put back the J_PWR jumper on the "USB" position.

Step 20: Software Installation



Step 20: Software Installation

Attach the DPScope to a free USB port on your computer. Wait for a minute or two to give the computer time to recognize the new instrument (you should get a screen message when that happens.

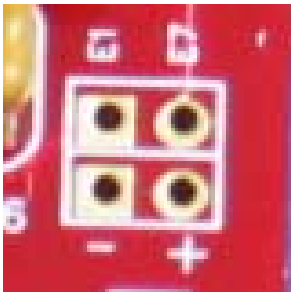
The DPScope's frontpanel LED should blink a few times and then stay on. *The blinking should last for about one second total*. If that's the case then your oscilloscope has just passed the first functional test! (The LED will only come on if the microcontroller actually starts successfully. If the blinking speed is much slower this would indicate a problem with the 16 MHz resonator or its connection to the microcontroller).

Note 1: The microcontroller (dsPIC30F2020) can get quite hot under operation – typically about 65 degC, which is already quite uncomfortabke to touch. However this is normal and no reason for concern (the dsPIC30F series is known to consume quite some power, and the DPScope runs the device at maximum speed).

Note 2: The DPScope software needs a screen resolution of at least 800 x 600 pixels. If your screen is 1024 x 768 pixels or larger it will automatically select a somewhat larger layout for better visibility. Also note that increasing the Windows font size (through the Display control in the Setup panel) will make the windows larger, which can prevent you from running the program on smaller screens (even though they may be 800 x 600 pixels or larger). You will get a message if that happens. In this case try reducing the Windows font size and start the program again.

Step 21: External Power Supply

Note 3: It is recommended to connect the DPScope to a USB port of your computer itself, or to a *powered* USB hub. Unpowered hubs tend to have large voltage drops, and the DPScope's level accuracy is dependent on a steady 5V supply voltage from the USB. You can test the voltage by connecting a voltmeter to the pins labeled "+" and "-" on the expansion header (bottom right on the PCB).

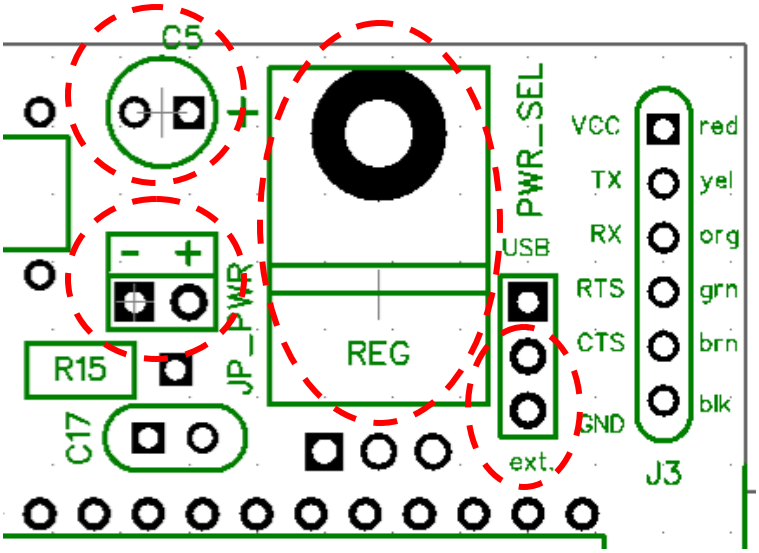


Note 4: If you absolutely cannot find a USB port which supplies close to 5V, or if excessive supply noise from the USB port affects the scope display, you can also use an external DC power supply. The DPScope's printed circuit board is already prepared for such an upgrade.

You will need

- a DC power supply, 9V ==, rated for at least 500mA.
- a LM7805 voltage regulator (easy to get, every Radio Shack has them).
- a 100 μF electrolytic capacitor, radial, 0.1" pin spacing, 10V or higher.

Solder in the regulator (REG) and the capacitor (C5). Screw down REG with a metal screw for better heat dissipation. Move the J_PWR jumper selector to the "ext." position. Attach the power supply cable the to "+" and "-" through-hole close to REG.

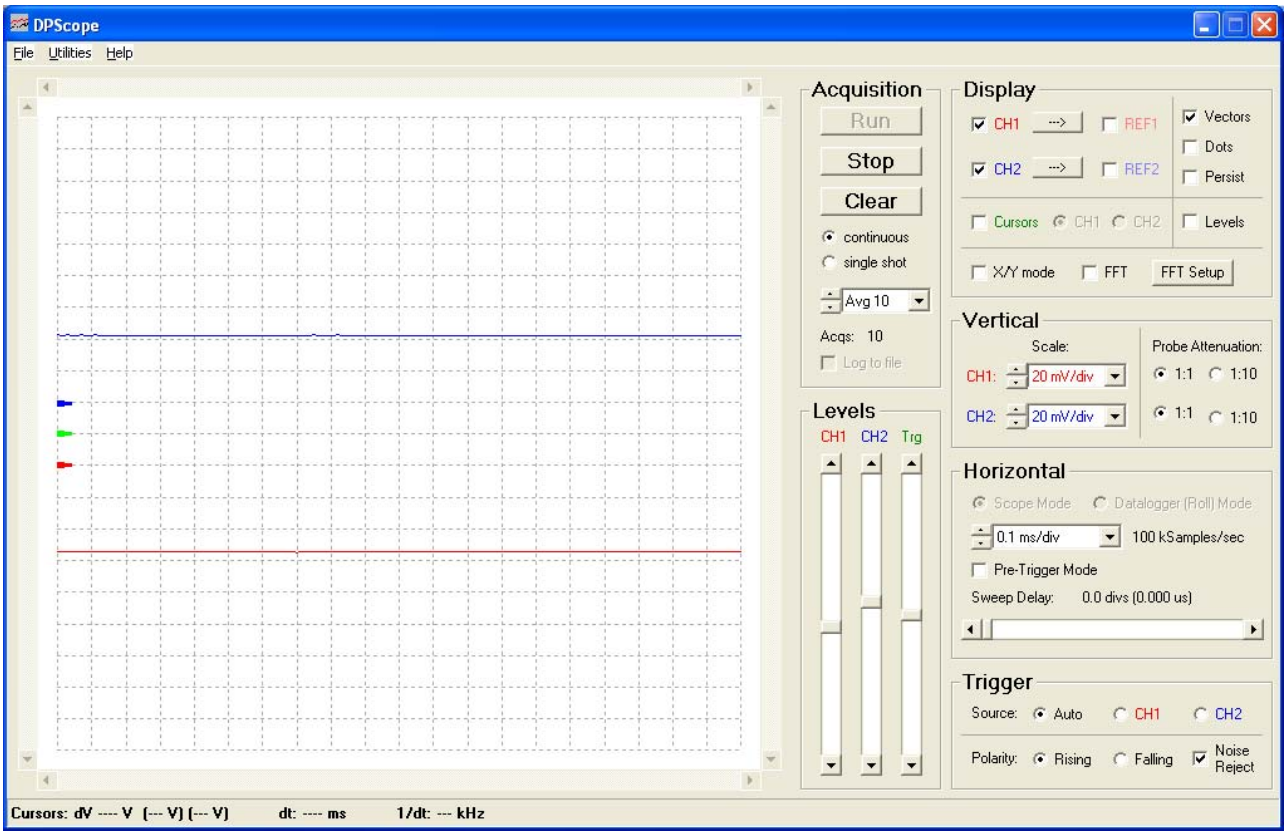


Step 22: Software Start

Launch the DPScope software. It should look like the picture below. Press the “Run” button – the two scope traces should come alive.

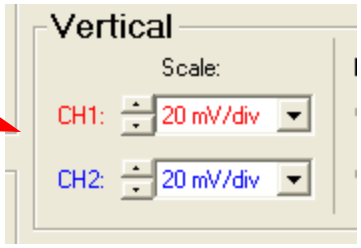
Attach the two probe cables to the BNC connectors (CH1 and CH2 on the frontpanel).

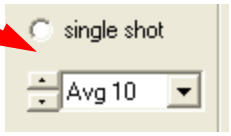
Now we need to make two simple adjustments to optimize the scope performance.

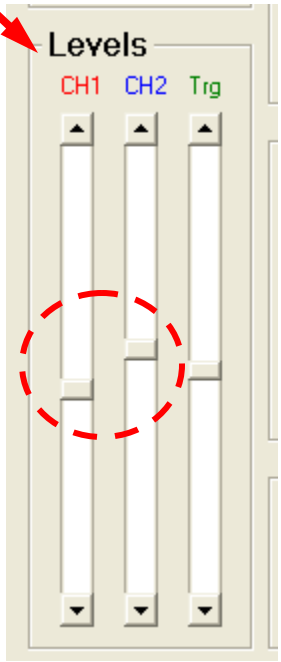


Step 23: Offset Adjustment

- In the “Vertical” menu change the scale for both channels to 20mV/div.
- In the “Acquisition” menu change the averaging to “Avg 10”.
- In the “Levels” menu move the sliders “CH1” and “CH2” to the middle. The ground level indicators (blue and red arrow on the left in the waveform display) will be in the middle as well.
- Short the probes, i.e. connect red grabber and black grabber together.

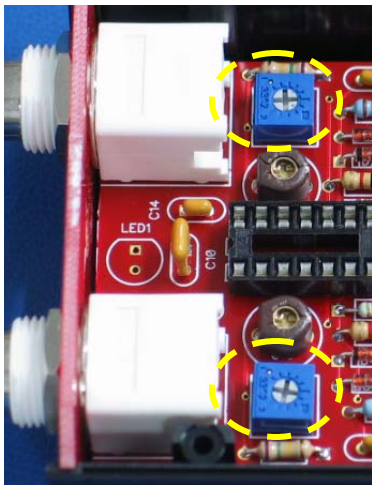


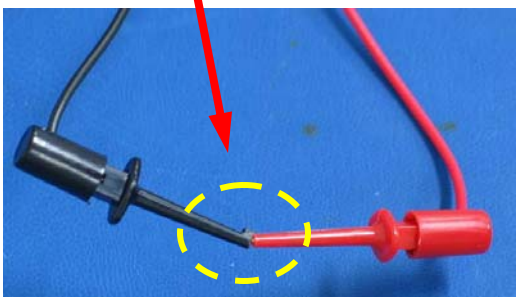




With a small screwdriver you can now adjust the offsets of the two channels:

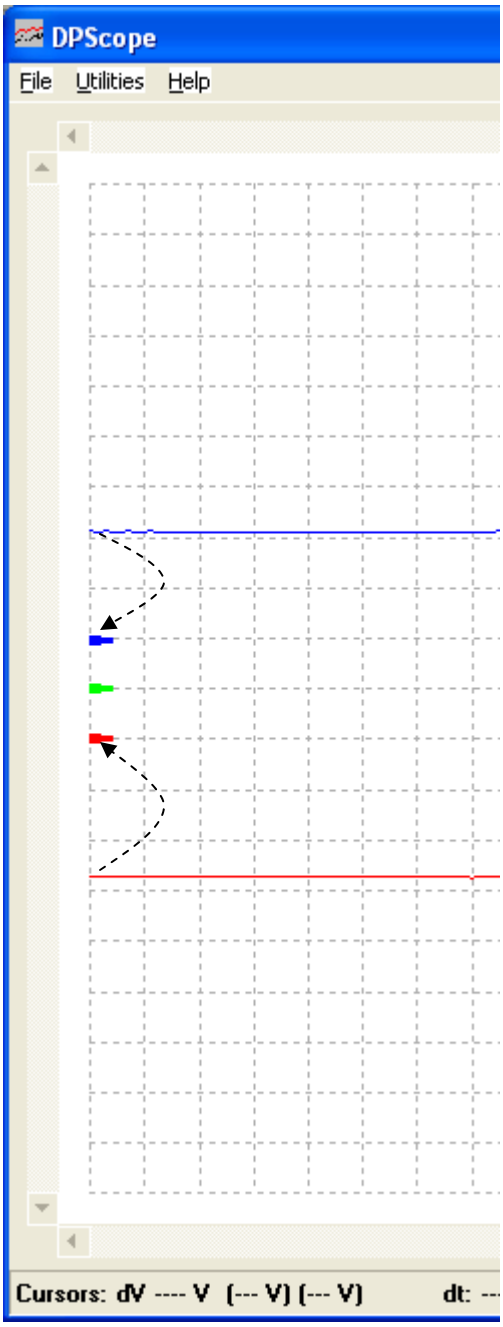
- Adjusting the two blue square trimmers (VR1 and VR2) will move the respective trace (red = CH1 and blue = CH2) up and down.
- Adjust the trimmers so the red trace is exactly at the height of the red arrow on the left, and the blue trace is exactly at the height of the blue arrow. Done!



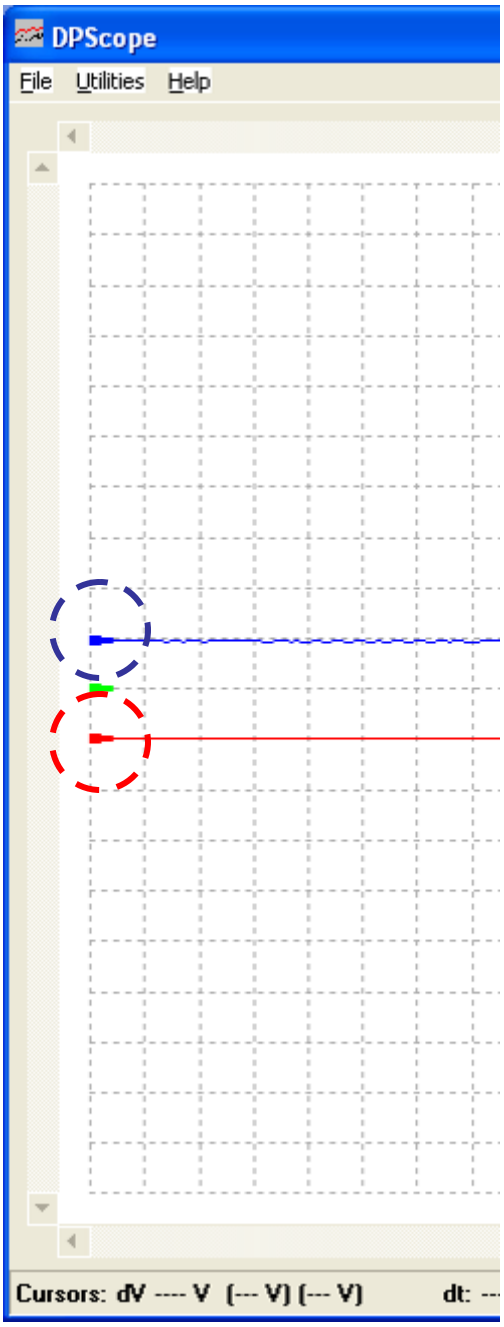


Step 23: Offset Adjustment

Before offset adjustment:

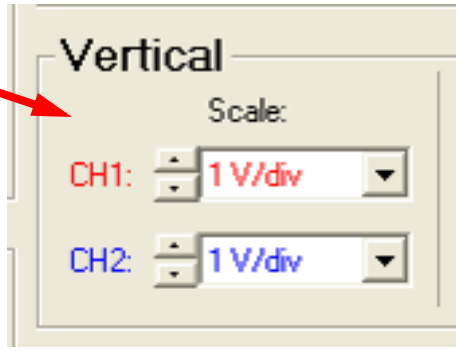


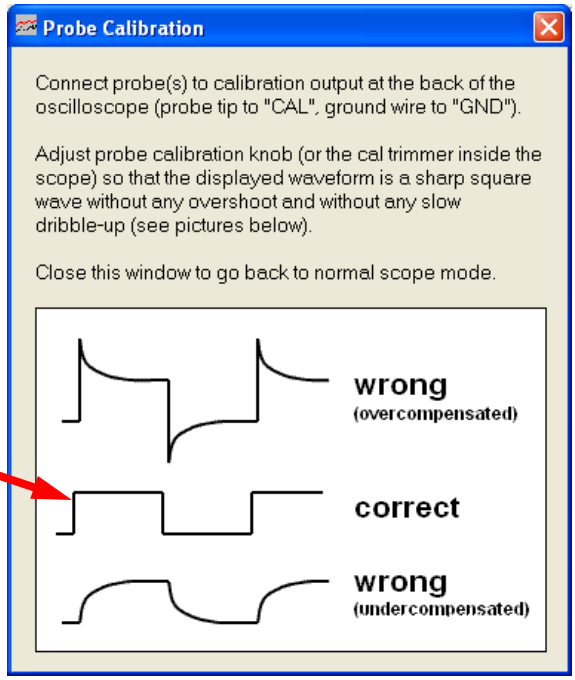
After offset adjustment:



Step 24: Probe Compensation

- In the “Vertical” menu change the scale for both channels to 1V/div.
- In the “Acquisition” menu leave the averaging at “Avg 10”.
- In the “Levels” menu move the sliders “CH1” and “CH2” a bit below the middle.
- Connect the probes to the calibration outputs on the back side of the oscilloscope:
  - Red grabber connects to “CAL” post
  - Black grabber connects to “GND” post
- In the DPScope window on your PC select Utilities → Probe Compensation. A small window with instructions will pop up.
- With a small *non-metal* screwdriver you can now adjust the probe compensation capacitors (C18 and C19, respectively).
- On the right side you see examples for overcompensated, undercompensated, and compensated probes.
- The adjustment is correct when the displayed signals are nice square waves with sharp corners, i.e. when there is neither overshoot (sharp peaks after each transitions) nor slow settling (rounded edges).
- At the same time, you have tested the scope’s acquisition circuitry.





All Done!

Congratulations, you have successfully assembled, set up and tested your new oscilloscope!

Now put the bottom cover on the instrument and screw it shut with the four Philips screws.

If you still have questions:

Email: support@dpscope.com

Webpage: http://www.dpscope.com