How to built a QCX SSB from a new QCX Kit with Rev 5 PCB. Documentation by DL2MAN

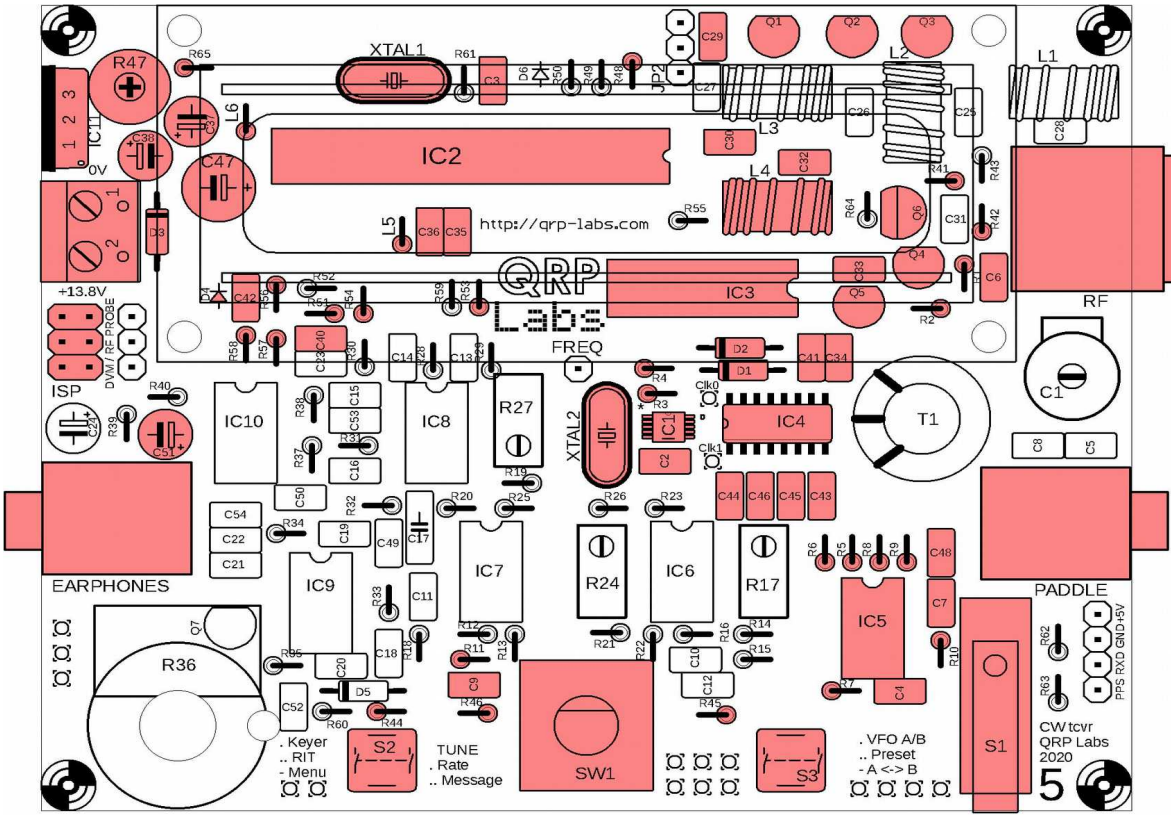
Introduction:
When I did the QCX SSB Modification (by Guido PE1NNZ) following his Instructions, I had several Points, that caused me headache. For Example: Where to put the missing Parts from his Rev 4 PCB in Rev 5 PCB ? So I ended up with 3 printed out Schematics (Rev 4, Rev 5 and QCX SSB), comparing them for hours, and trying to figure out where to put what....
So I had the Idea to do a step by step documentation, so future builders will have it easier.
This is not intended as a full manual for the QCX SSB, but only as building aid for other HAMs.

What do you need ?

1. QCX Kit from QRP Labs http://shop.qrp-labs.com/qcx
2. If you want multiple Bands, a low pass filter for each Band http://shop.qrp-labs.com/kits/LPF
3. Additional small parts for the built: (2x Resistor 82K, 1x Capacitor 220nF)
4. Arduino UNO or Compatible to program the Firmware
5. Arduino IDE Software to do so (https://www.arduino.cc/en/main/software)
6. Some Jumper wires (male – female) for example https://www.amazon.com/cables-multicolor-alambre-tableros-arruinar/dp/B01EV70C78?ref_=fsclp_pl_dp_2
7. The latest firmware file „QCX SSB.ino“ from https://github.com/threeme3/QCX-SSB
8. Some additional small parts depend on the decisions you make down the road....

The tools required are the same, as for the original QCX Kit.

Let´s jump directly into step one:
Place all the red components below and solder them in place. Look carefully to the tables on the next pages, for what value to place where and use the checkbox to verify, you did not miss anything.



Start with IC´s and Crystal's:

PCB Rev5 Label	**Place (QCX SSB Schematic)**	**Value (bold if changed)**	**Remark**	**Checkbox**
IC2	IC2	28 Pin Socket	for ATMega328P Controller	
IC3	IC3	SN74ACT00N		
IC5	IC5	LM4562A		
XTAL1	XTAL1	20 Mhz		

Continue with capacitors

PCB Rev5 Label	Place (QCX SSB Schematic)	Value (bold if changed)	Remark	Checkbox
C3	C3	0,1µF - 104		
C29	C29	0,1µF - 104		
C35	C35	0,1µF - 104	Solder a little higher on PCB, for easier step 3	
C36	C36	0,1µF - 104		
C30	C30	**30pF – 300**	for Multiband	
C32	C32	**10µF (electrolytic, Minus to GND)**		
C6	C6	0,1µF - 104		
C33	C33	1nF – 102		
C41	C41	0,1µF - 104		
C34	C34	0,1µF - 104		
C40	C40	0,1µF - 104		
C42	C42	10nF – 103		
C43	C43	470nF – 474		
C44	C44	470nF – 474		
C45	C45	470nF – 474		
C46	C46	470nF – 474		
C2	C2	0,1µF - 104		
C48	C48	0,1µF – 104		
C7	C7	**1nF – 102**		
C4	C4	**1nF – 102**		
R11	**C39**	**0,1µF – 104**		
R58	CXX	**220nF**	**not in Kit !**	
C37	C37	10µF (check Polarity)		
C38	C38	10µF (check Polarity)		
C51	C51	10µF (check Polarity)		
C47	C47	470µF (check Polarity)		

Continue with resistors:

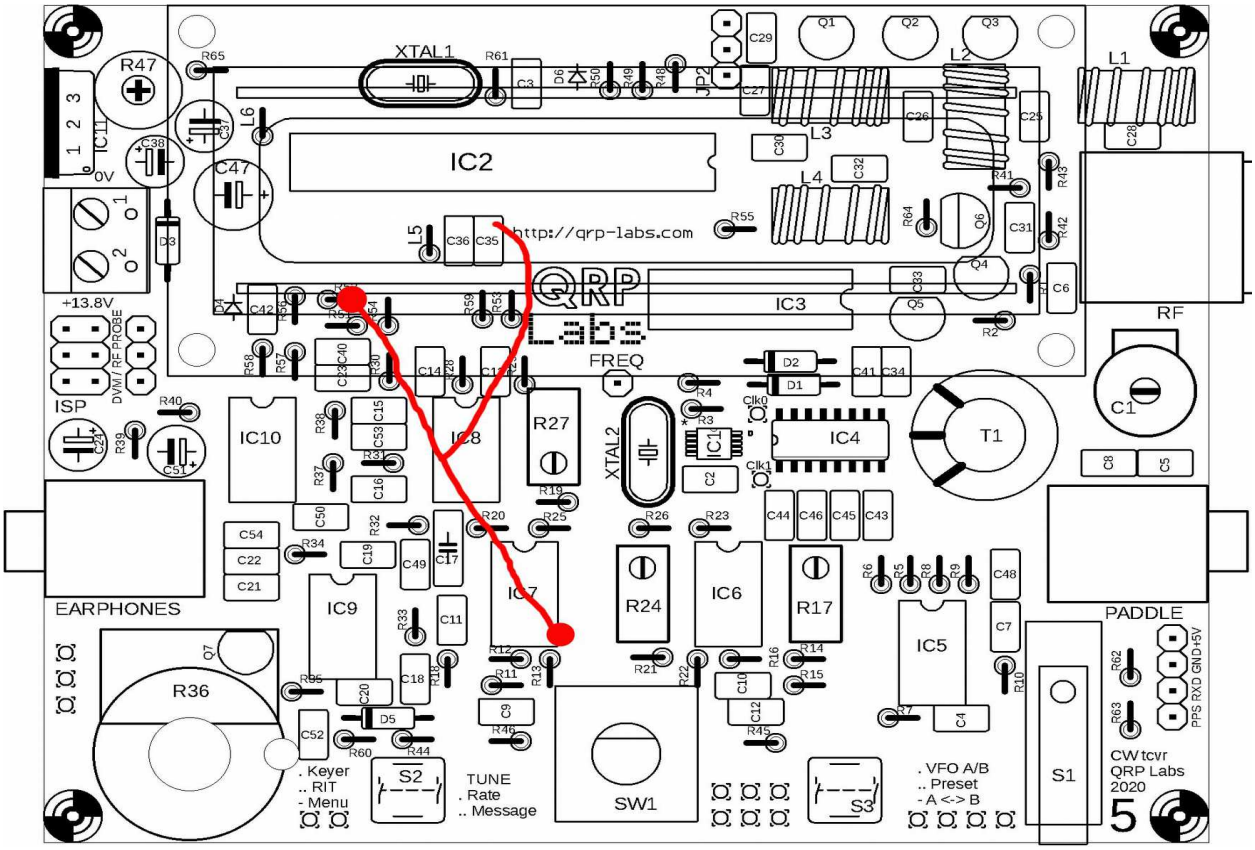
PCB Rev5 Label	Place (QCX SSB Schematic)	Value (bold if changed)	Remark	Checkbox
R65	R65	3,3KOhm		
R48	R48	270 Ohm	**Alternative 1,2K for less Brightness and Current**	
R41	R41	**470 Ohm / in Rev 5 150Ohm**		
R42	R42	**1K / in Rev 5 1,2K**		
R1	R1	10K		
R2	R2	10K		
R3	R3	1K		
R4	R4	1K		
R53	R53	**1K**		
R54	R54	1K		
R51	R51	10K		
R57	R57	10K		
R56	R56	**10K**		
D4	**R(D4)**	**10K**		
R44	R44	3K3		
R45	R45	1K		
R46	R46	10K		
C9	**R49**	**10K**		
R7	R7	**82K**	**not in Kit !**	
R10	R10	**82K**	**not in Kit !**	
R5	R5	100 Ohm		
R6	R6	100 Ohm		
R8	R8	100 Ohm		
R9	R9	100 Ohm		

Finish step one, with the final parts:

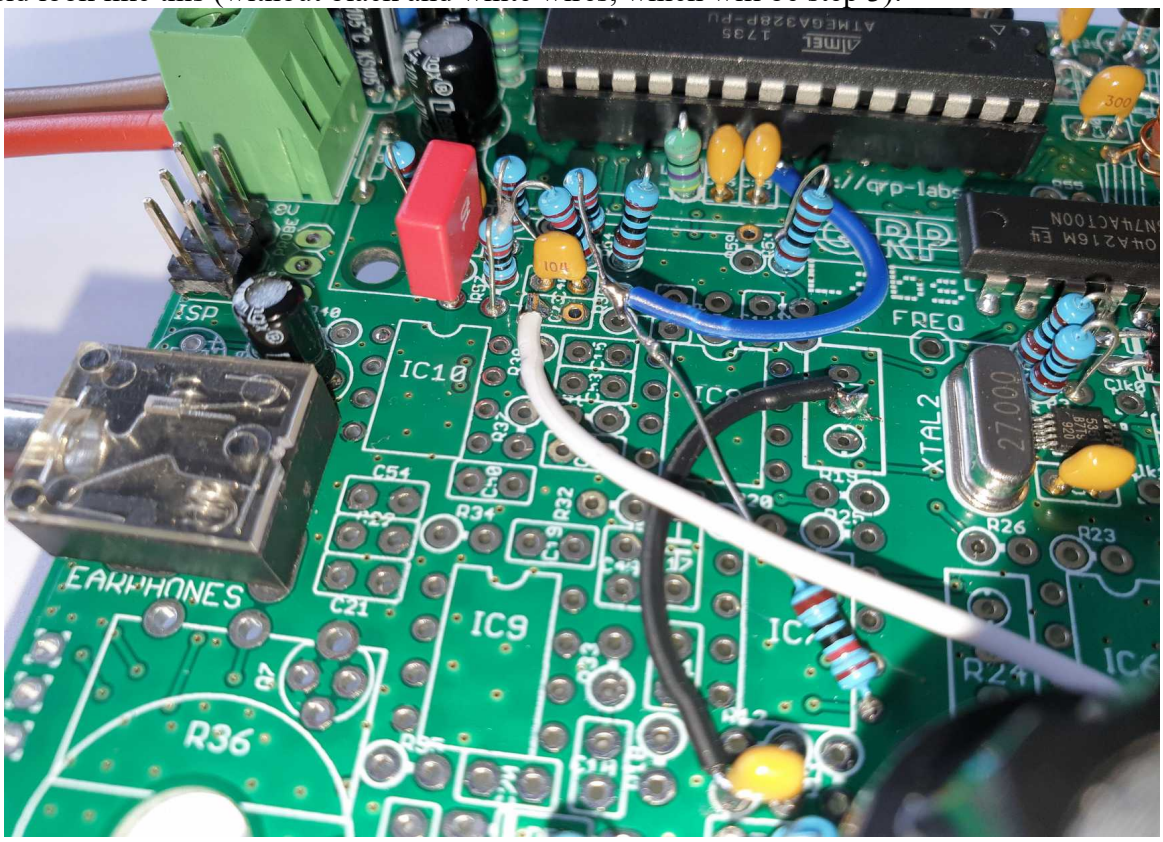
PCB Rev5 Label	**Place (QCX SSB Schematic)**	**Value (bold if changed)**	**Remark**	**Checkbox**
L5, L6	L5,L6	100µH (47µH in Rev 5 – not critical)	fixed Inductors	
L4	L4	1µH / 16 Turns (on T50-2 -red)	**For Multiband**	
D3	D3	1N5189	**(Alt: Wire Bridge, if you know what you do – for Current Saving)**	
D1, D2	SD1,D2	1N4148		
Q6	Q6	MPS751		
Q1,Q2,Q3, Q4,Q5	Q1,Q2,Q3,Q4 ,Q5	BS170		
IC11	IC11	Voltage Regualtor 7805		
R47	R47	Trimmer Poti 100K		
S1,S2,S3	S1,S2,S3	Switches		
SW1	SW1	Rotary Encoder		
Paddle/Ear phone	Paddle/Earph one	3,5mm Jacks		
Power Connector	Power Connector	Power Connector		
RF Jack	RF Jack	BNC Socket		
Display Connector	Display Connector	16-Pin Male -Male on PCB		
ISP	ISP	6 Pole Connector		

You can also put in the screws and spacers for the display, and solder in Male/Female 16 Pin connector to your Display PCB.

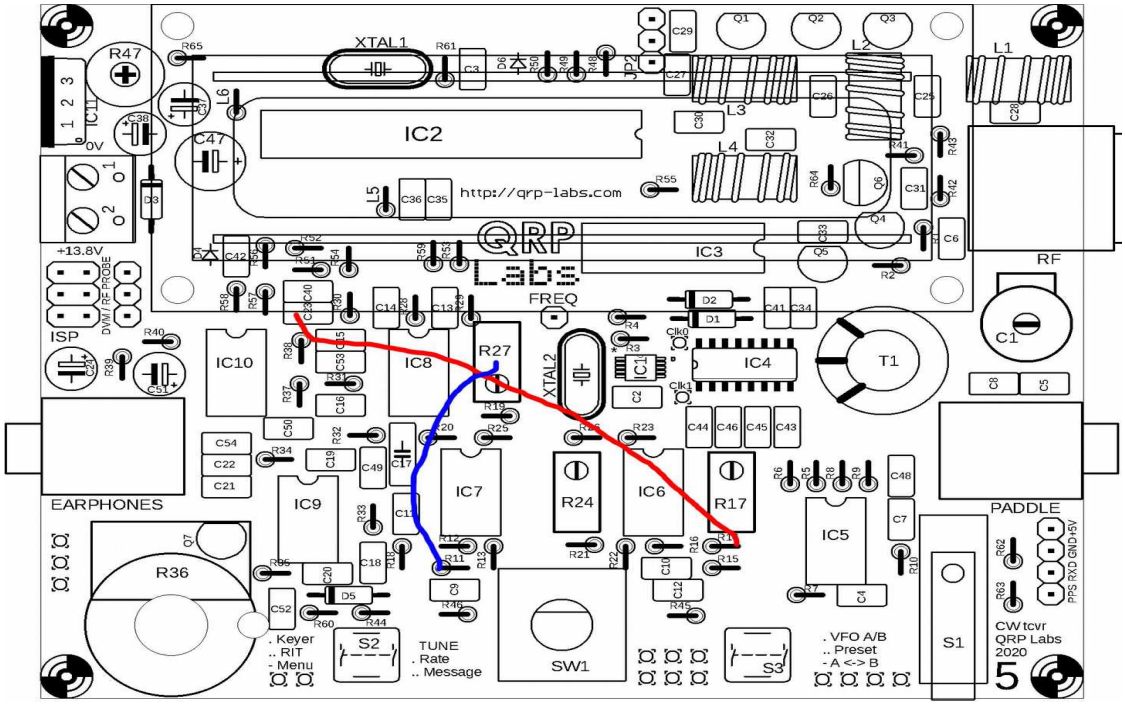
Step 2:
Place 10K resistor at right side of PCB Label R52
Place 10K resistor at Pin 5 of PCB Label IC7 (lower right pin)
Connect the 2 legs of the resistor, and wire them to Upside of C35 (Side towards IC2)



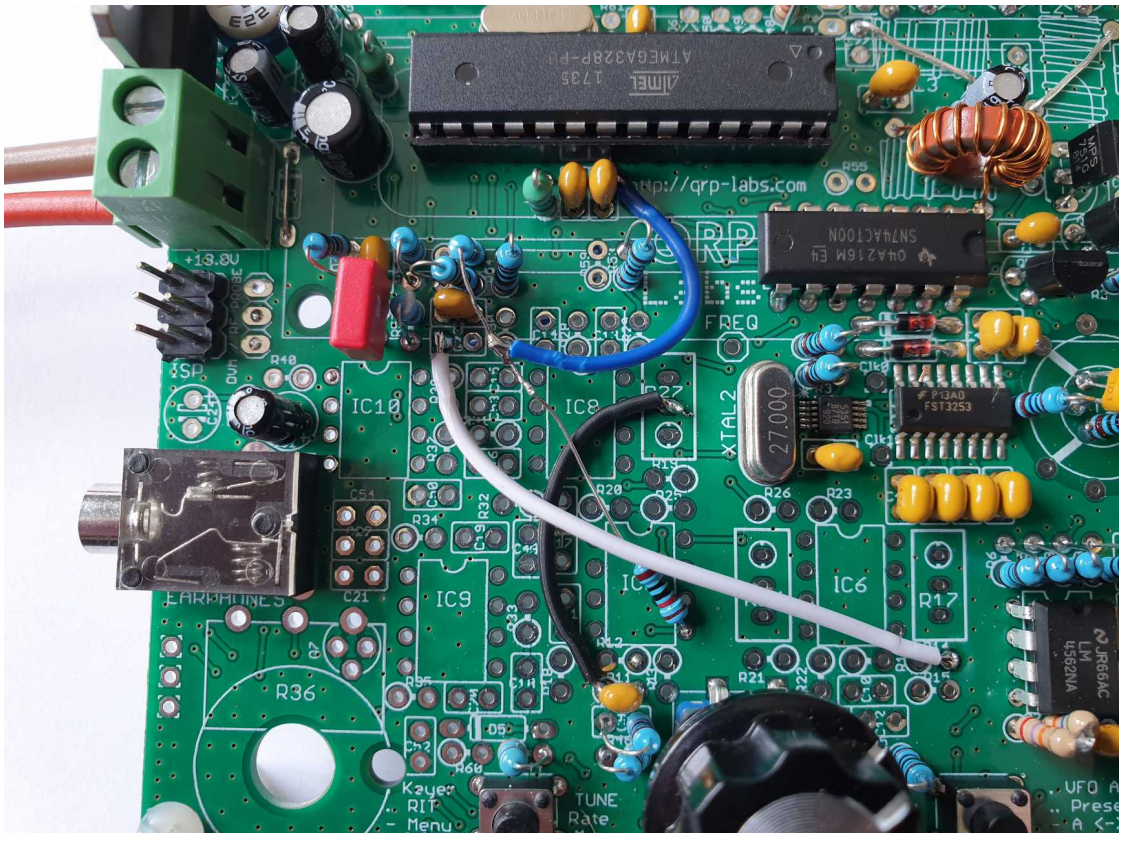
Should look like this (without black and white wires, which will be step 3):



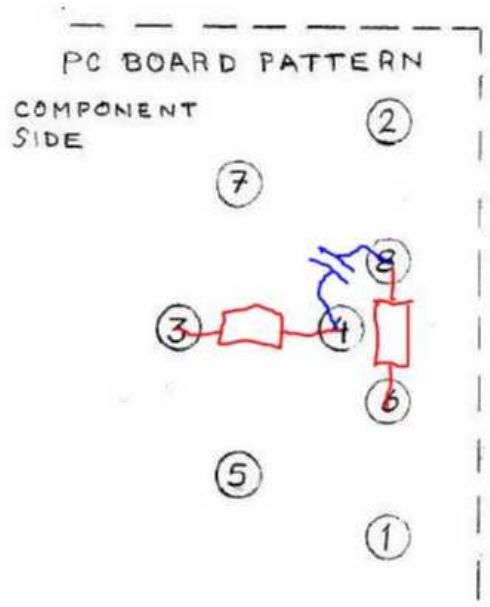
Step 3:
Solder a Wire from left Pin of PCB C23 to Right Pin of PCB R14
Solder a Wire from left Pin R11 (Which is now C39) to middle Pin of PCB R27



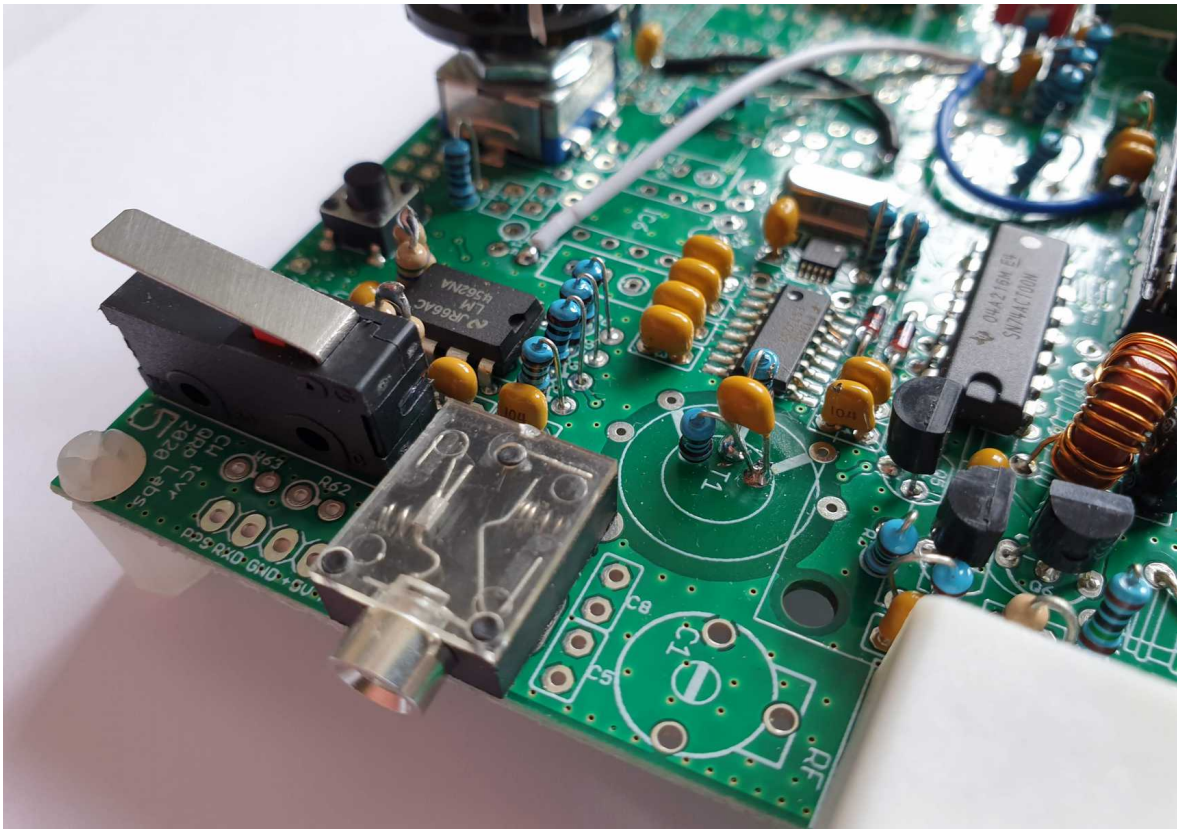
Should now look something like this: (black and white wire)



Step 4:
Place Resistor 1K between Pins 3 and 4 of PCB Pattern T1
Place Resistor 1K between Pins 6 and 8 of PCB Pattern T1
Place Capacitor 10nF between Pins 4 and 8 of PCB Pattern T1

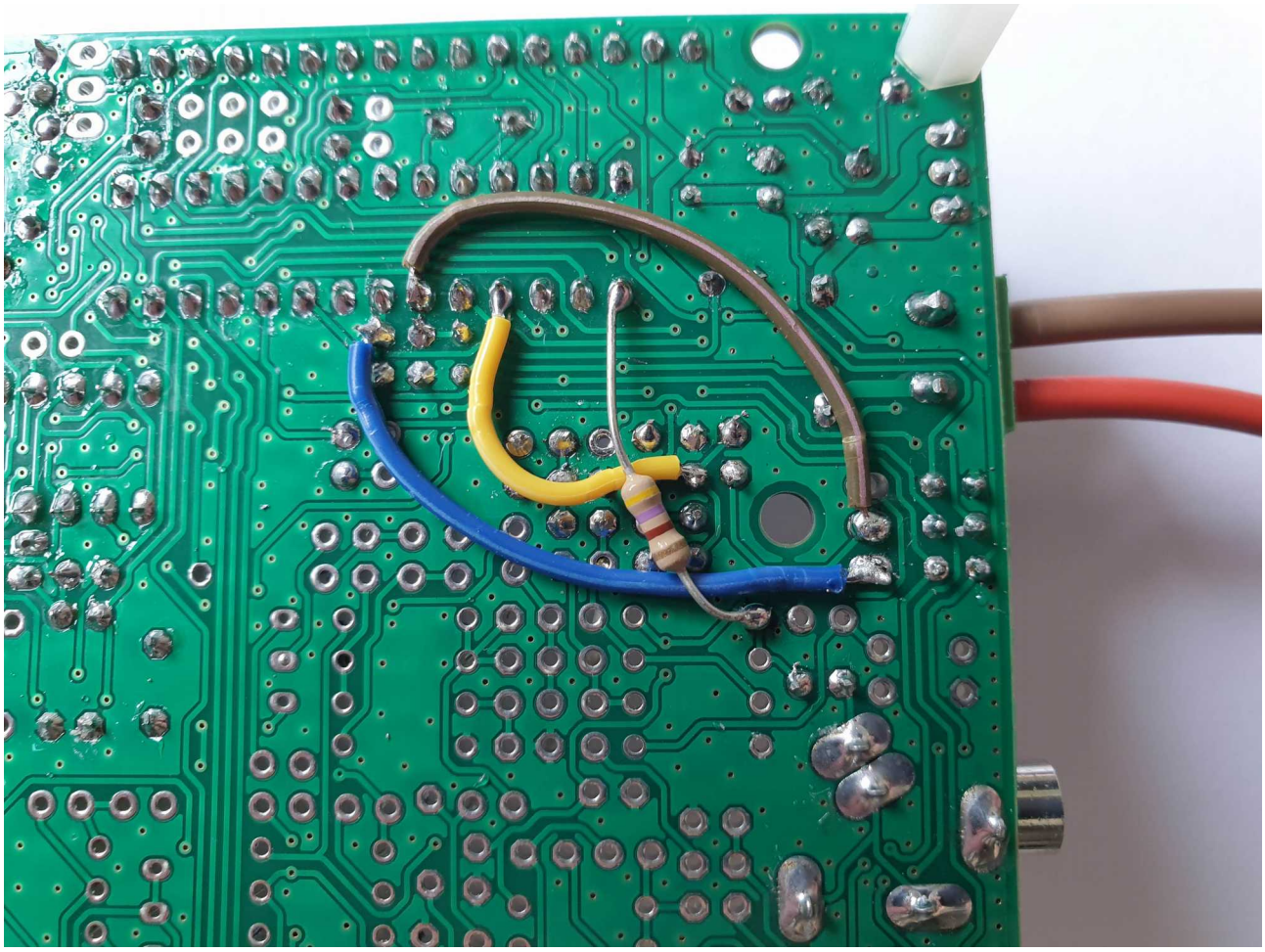


It´s hard to take a detailed picture of this, but you should get the Idea....



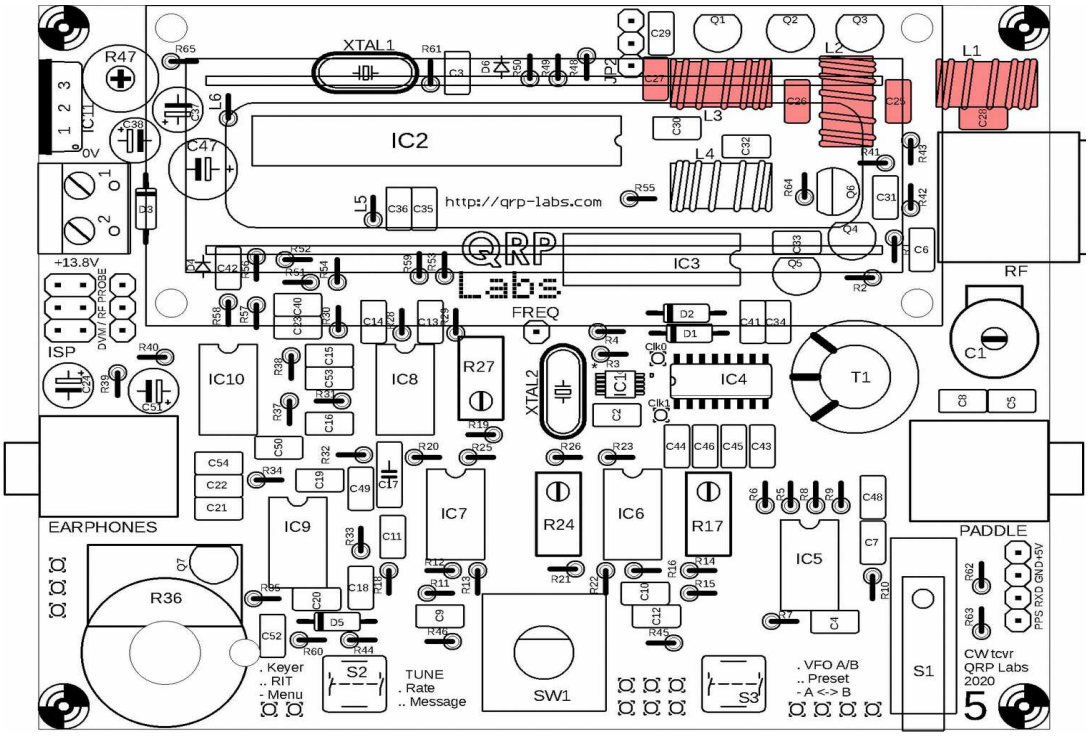
Step 5:
Finally some wiring on the soldering Side of PCB:
Colors are up to you, but for description I will refer to my chosen colors....
Yellow wire connects IC2 Pin 18 to lower Side of C42
Brown wire connects IC2 Pin 20 to middle Pin of DVM Pins on PCB
Blue wire connects IC 2 Pin 21 to lower Pin of DVM Pins on PCB.(In my case I soldered this one to upper leg of C 35, because it is directly connected to IC2 Pin 21)

If you want to use Headphones, or active speakers, solder in 470 Ohm resistor between IC2 Pin 15 and IC10 Pin 1



Step 6:
Now we need to take care about the Low Pass Filter.
For operation without unwanted harmonics, you need a Low Pass Filter, matching the Band of Operation after the Finals. There are several Options.

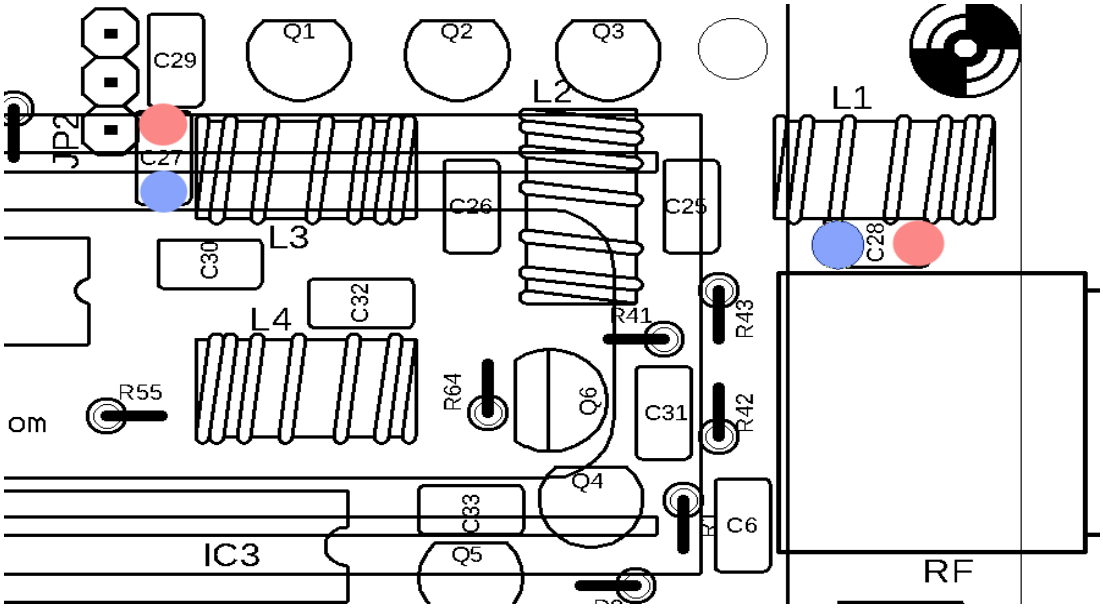
Option 6a)
You could use the LPF Components, supplied with your Kit.
Simply use the components supplied with your Kit and place them according to Original QCX Manual. However this would make a „kind of“ Mono-band QCX-SSB. If you build one for highest Frequency you want to use (For Example 10m or 20m), then you could add additional Filters (Off PCB) for Lower Frequencies. (30m,40m,60m,80m.... You get the idea.) The relays board from QRP Labs for example works this way. However, you always will have loss of 2 filters this way, with the exception of your highest used frequency.



I´v taken all those possible values for QCX LPF, and put them together in one Table for you:

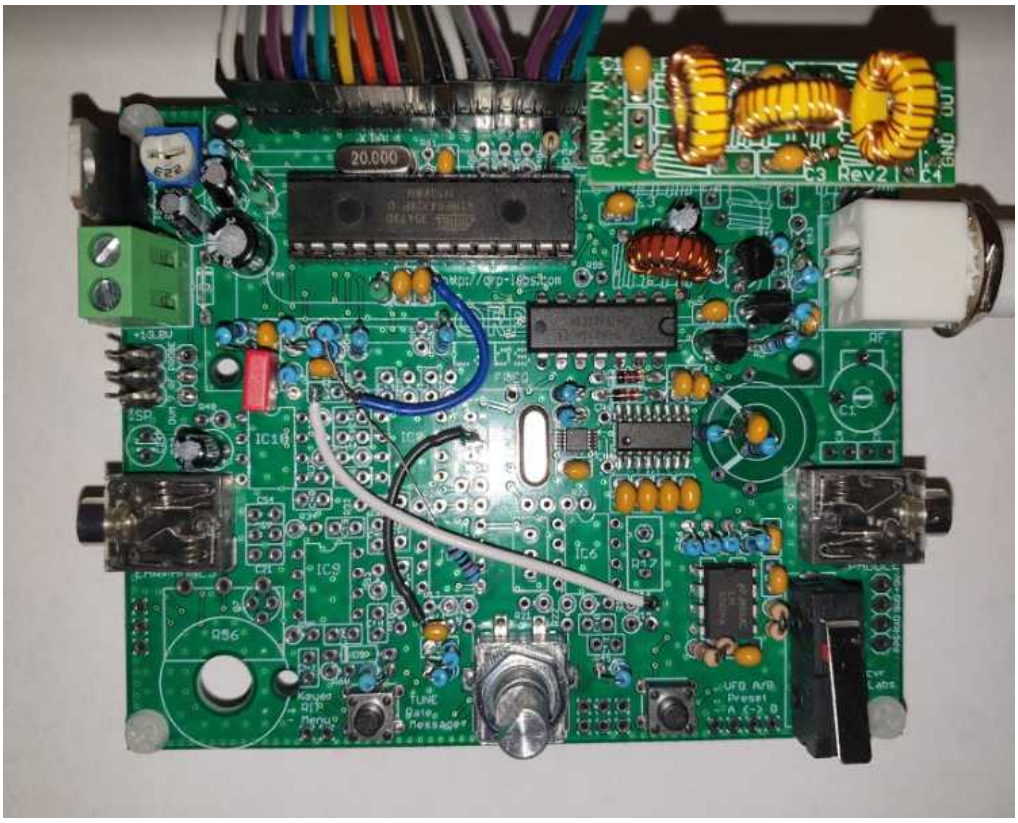
QCX LPF	C25/C26	C27/C28	C30	L2			L1 & L3		
Band	Value	Value	Value	Toroid	Value	Windings	Toroid	Value	Windings
80m	1200pF	470pF	180pF	T37-2	3,0µH	27	T37-2	2,4µH	25
60m	1200pF	680pF	30//56pF	T37-2	2,3µH	24	T37-2	2,1µH	23
40m	680pF	270pF	56pF	T36-6	1,7µH	24	T36-6	1,4µH	21
30m	560pF	270pF	30pF	T36-6	1,3µH	20	T36-6	1,1µH	19
20m	390pF	180pF	30pF	T36-6	0,9µH	17	T36-6	0,77µH	16
17m	270pF	100pF	30pF	T36-6	0,67µH	15	T36-6	0,55µH	13

Option 6b)
External LPF, still using QCX BNC Socket for Antenna Output. You could insert external Low Pass Filters, and still use the On-Board RF Out BNC Jacket to feed the Antenna:



Use PCB C27 to feed Input of external LPF
Use PCB C28 to feed back Filtered RF to BNC Jack
Red is RF, and Blue is GND.
If you -eg solder Coax Cable there- red needs to be inner wire, and blue needs to be shield.

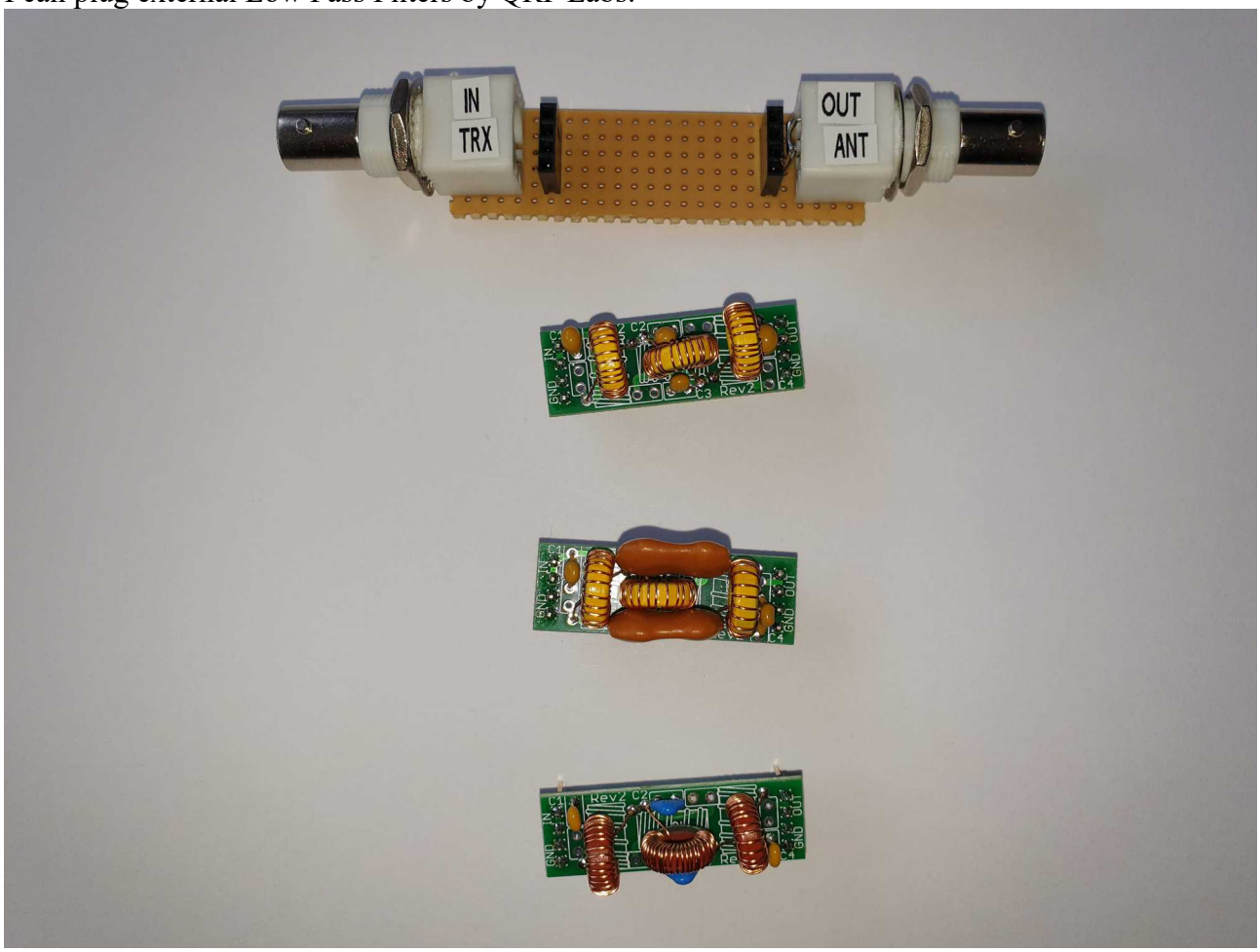
This would be an example, where I soldered wires on component side of PCB with connectors to the points shown above, and simply plug in external LPF´s. This way I had to extend display connectors. This can be avoided, by soldering the LPF connections on the Downside of the PCB.



Option 6c)
Wire final Transistors directly to BNC-Jack and use external LPF´s in Coax Line. Like this:

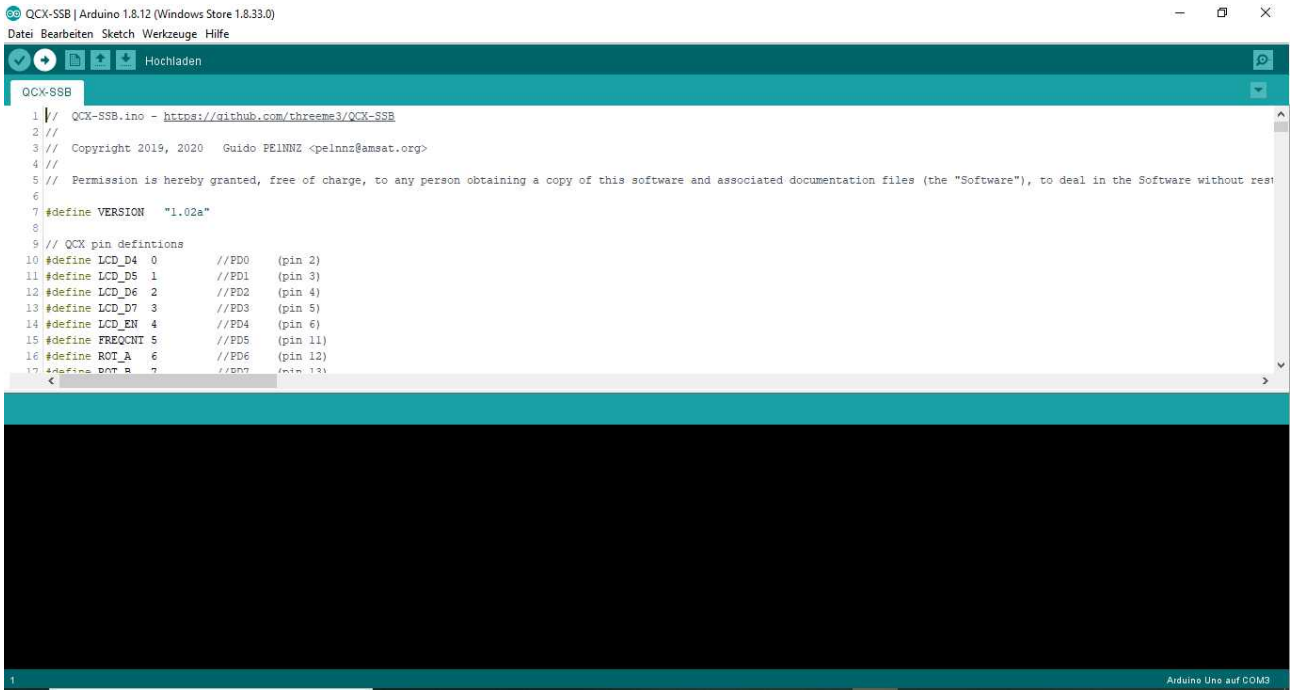
There are other ways to do it, but this way, you can use un-isolated wires with low chance of shorting something.

Then you need to get your LPF´s into Coax Line. I´ve build a small PCB, with 2 BNC Jacks, where I can plug external Low Pass Filters by QRP Labs:

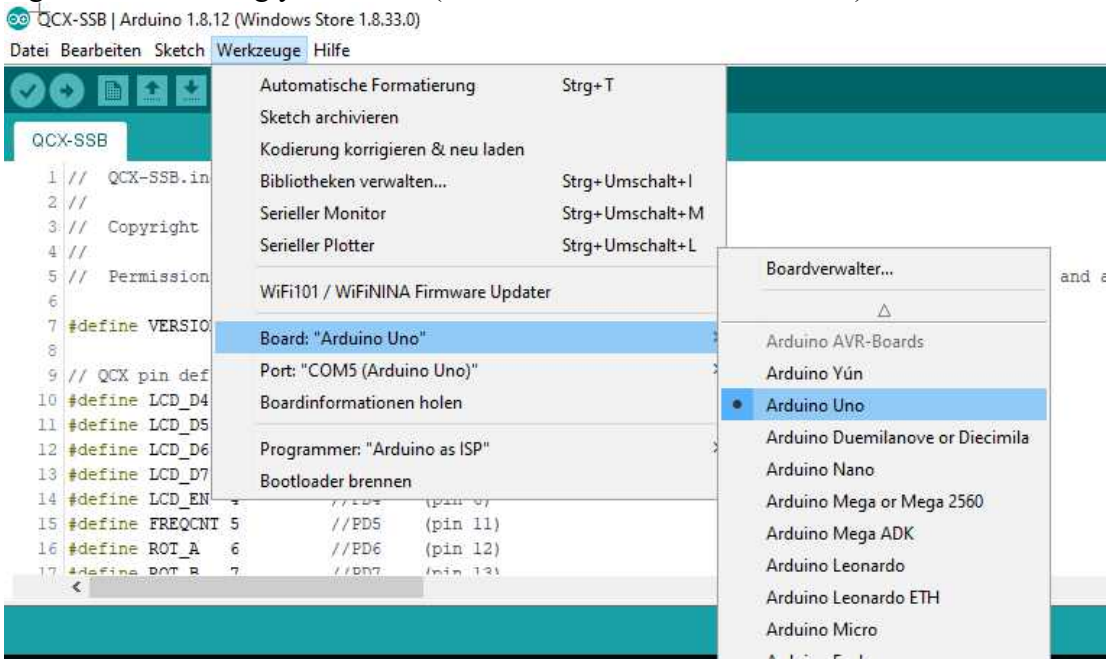


Step 7:
Making it alive.... or putting firmware on it....
This is actually pretty simple, if you carefully follow step by step.
First you need to install Arduino IDE (Source: https://www.arduino.cc/en/main/software)
Then you need to Download the Firmware File „QCX SSB.ino“ from
https://github.com/threeme3/QCX-SSB
When you´re using Guidos source code (The INO File, instead of the Hex File) there´s no need for an additional software tool...

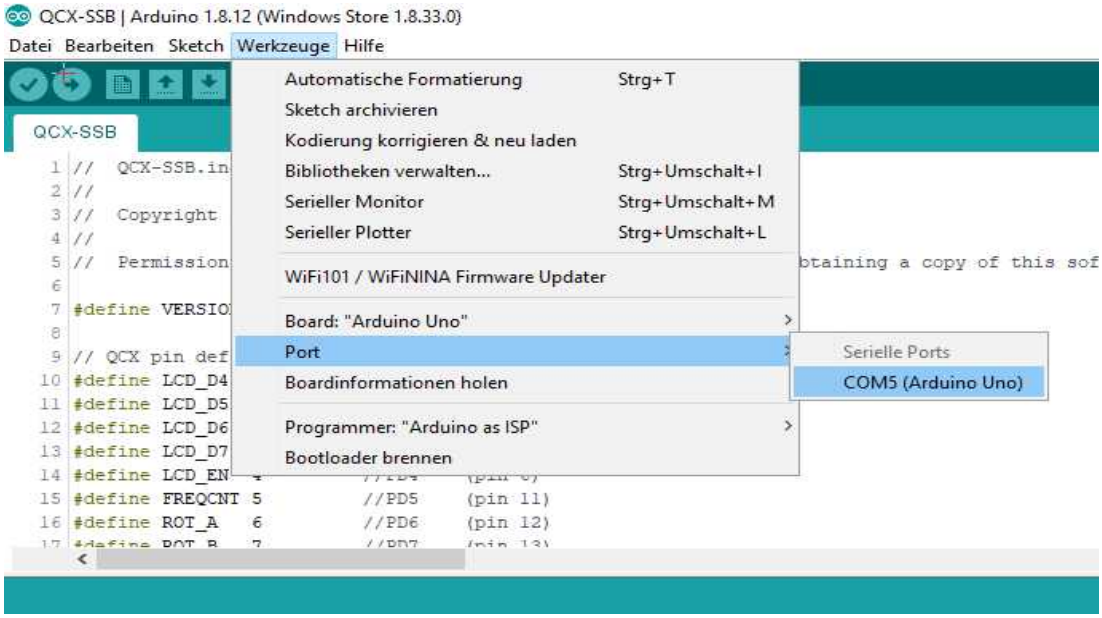
So, you installed Arduino IDE, and have Arduino UNO connected to your computer by USB ? Let´s get started. If your Installation is working, by double clicking the Downloaded „QCX SSB.ino“ Arduino IDE should start and should look like this. Except for your language ;)



First thing to do is choosing your Board (Tools → Board → Arduino Uno):



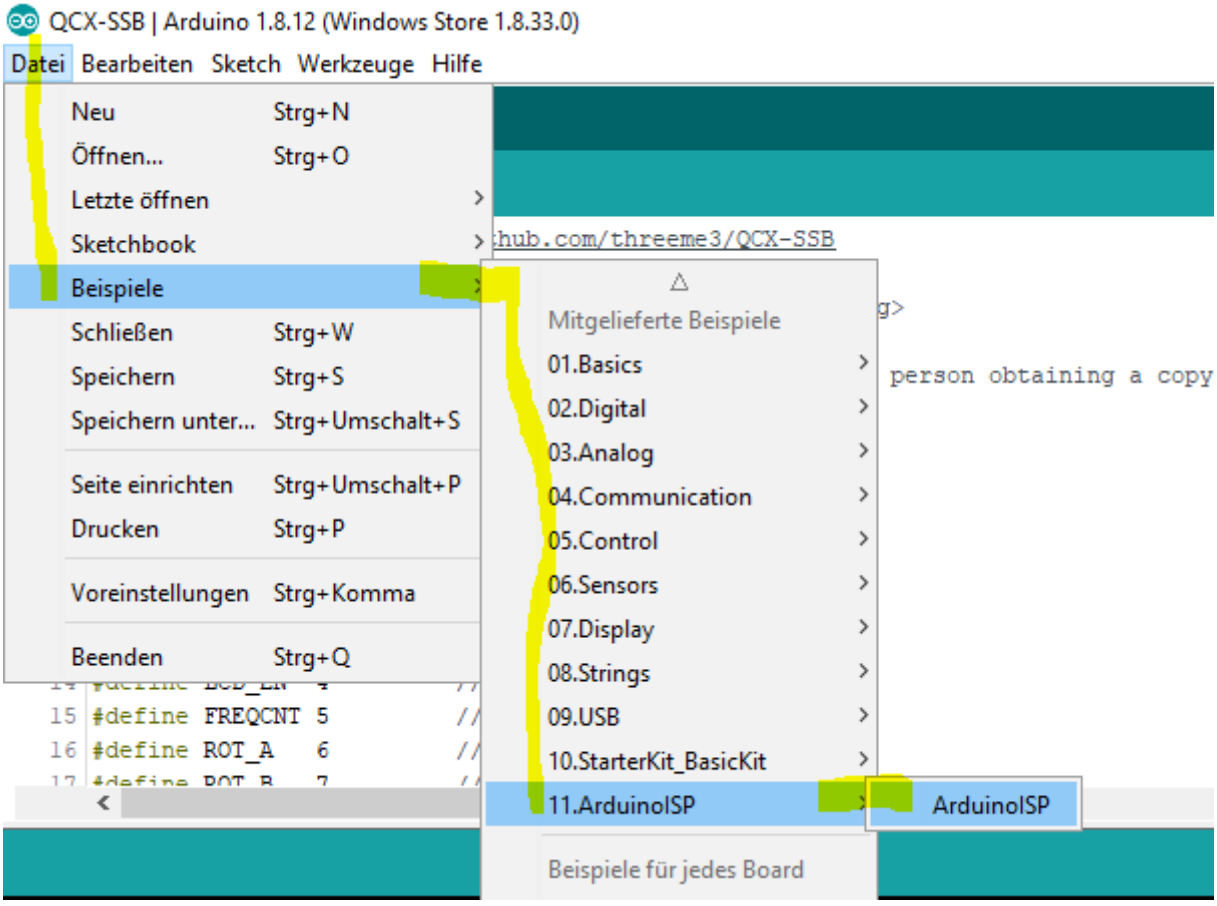
followed by setting the right com Port for your Arduino UNO:



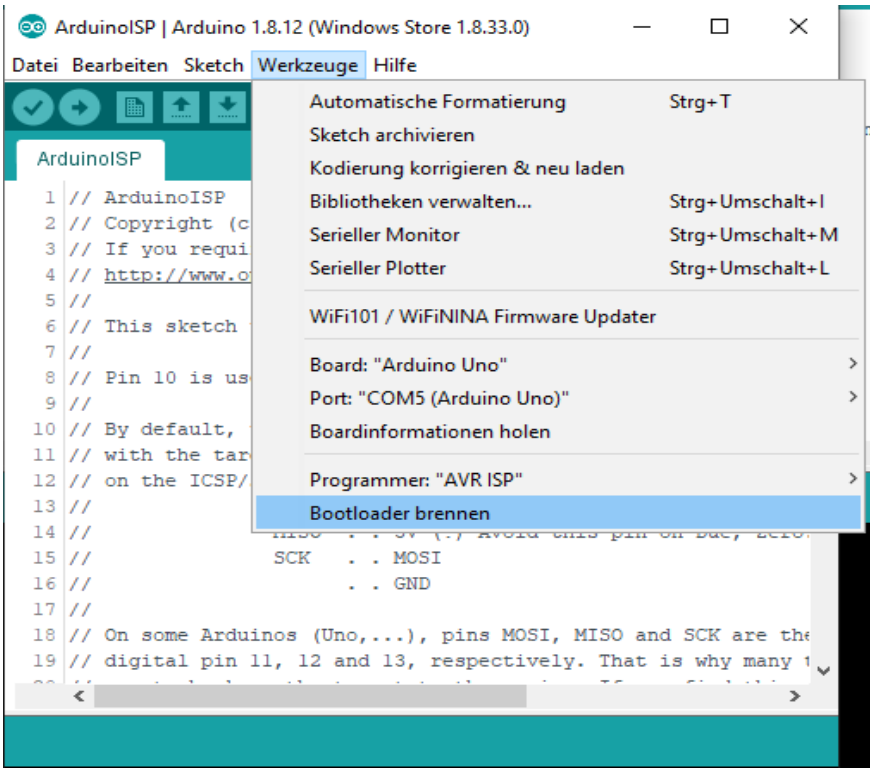
After that, you should see something like this in the lower right corner of the Arduino IDE Window:

Next, you need to prepare the Arduino UNO, so it will serve as Programmer for the QCX.

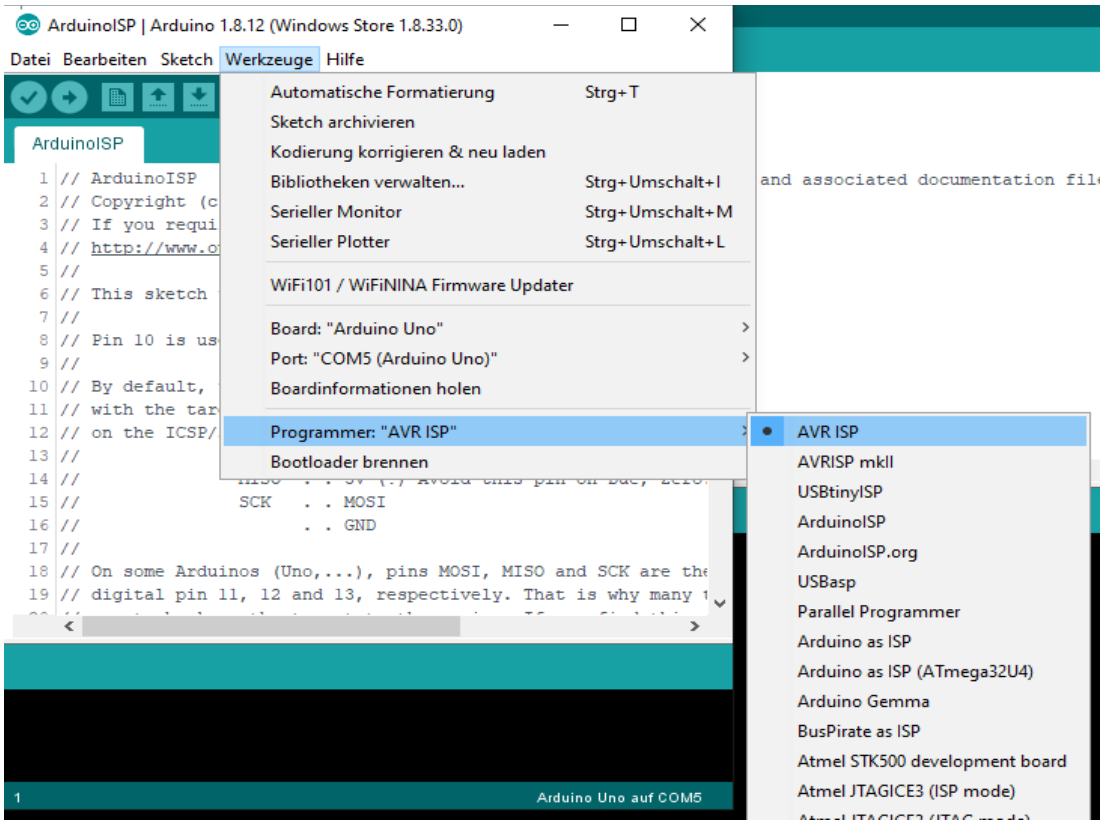
To do so choose File → Examples → ArduinoISP → Arduino ISP



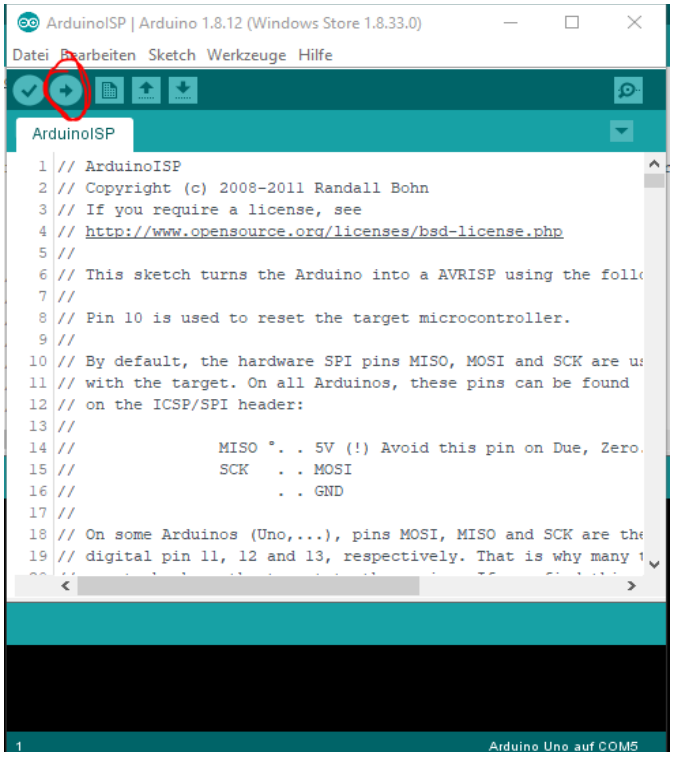
A new sketch window will open.
In this Window, first select Tools → Burn Bootloader.



When this is done, select Tools → Programmer → AVR ISP

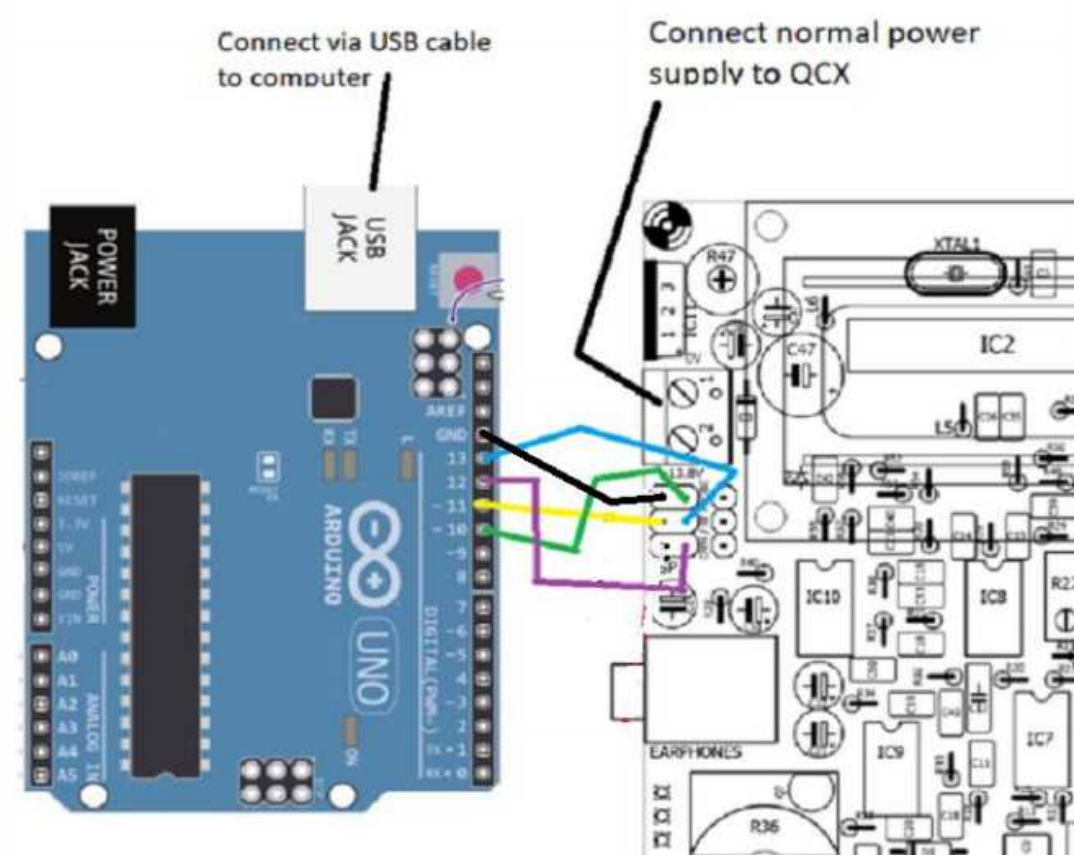


Then you need to push this button:

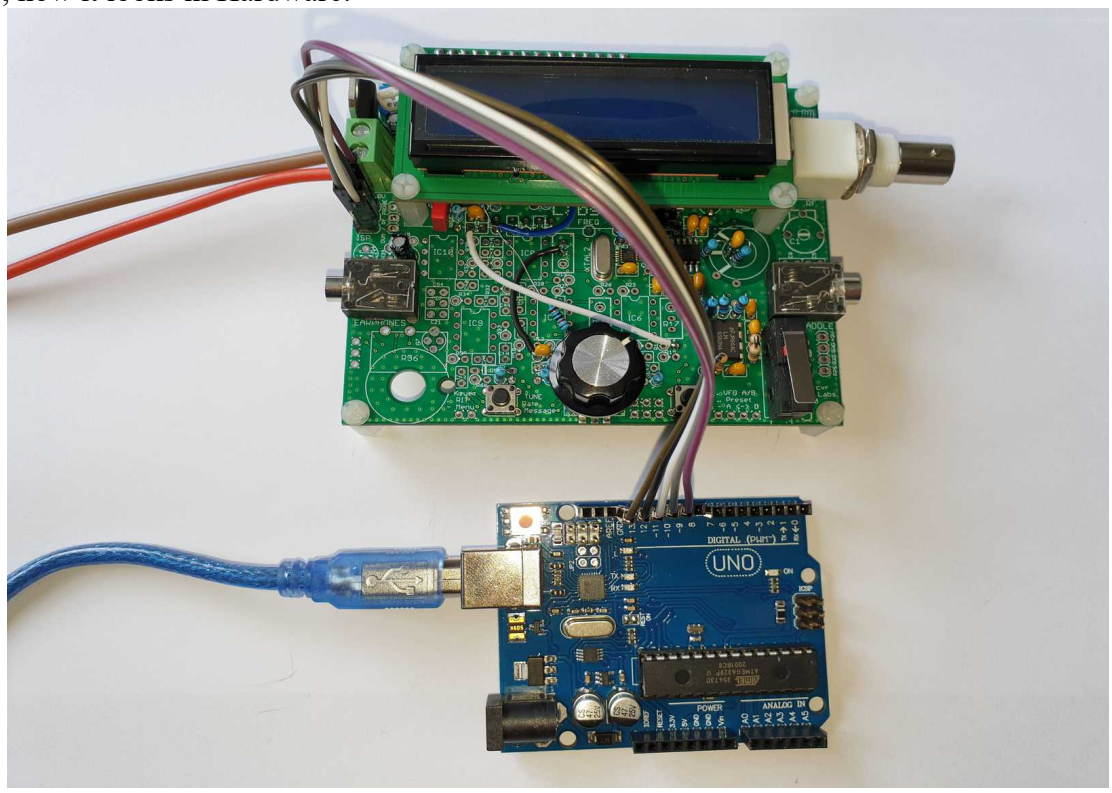


and after a few seconds, the Arduino UNO will be ready to serve as Programmer for our QCX.
We can close our Arduino ISP Window now, as we don´t need it anymore, and we now should have only QCX SSB Window remaining.

Now disconnect the Arduino from PC (USB) and connect the Arduino UNO and the QCX by using the wire bridges. (Male / Female)
This Picture from QRP-Labs Manual shows how:

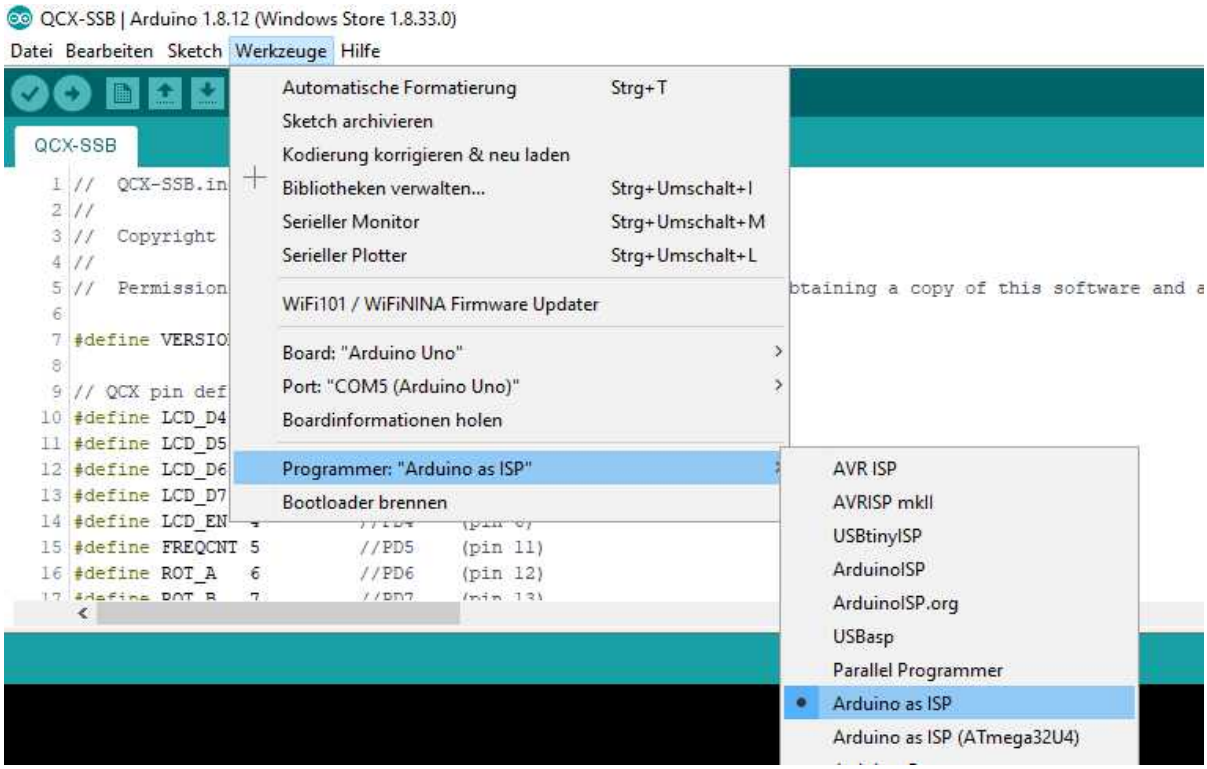


This is, how it looks in Hardware:

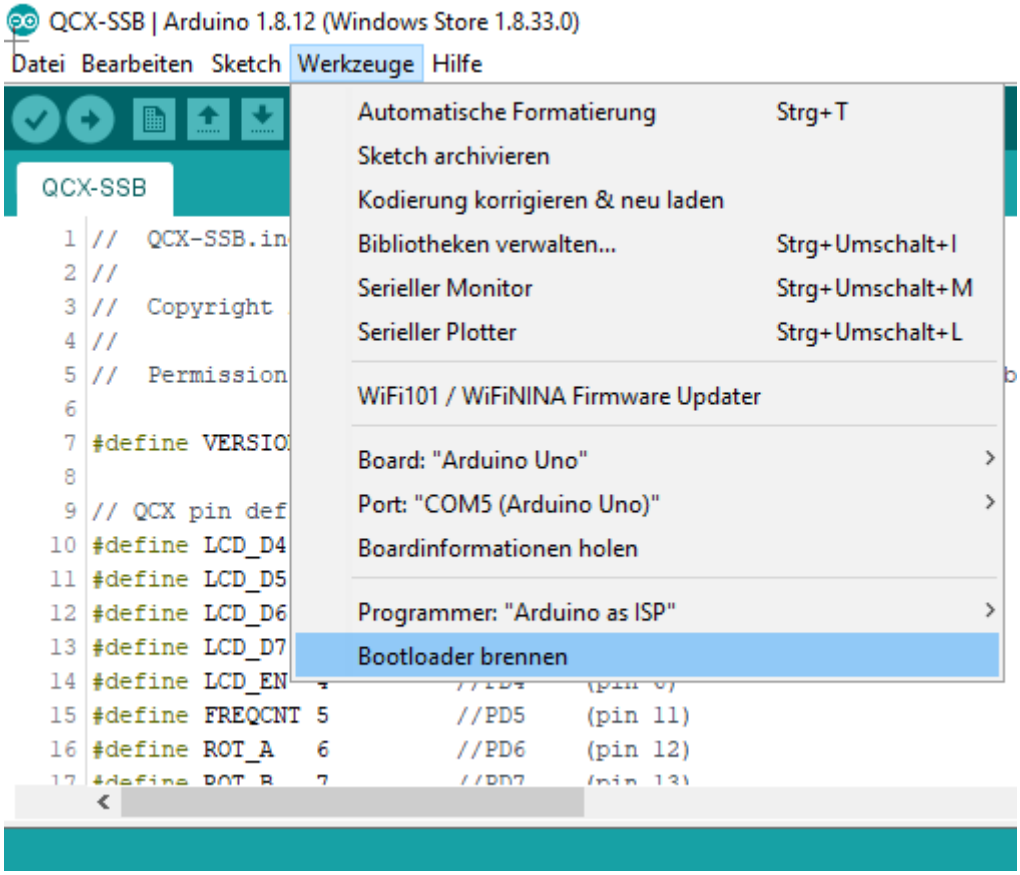


After you have double checked, that the wiring is right, you can connect Arduino´s USB cable to your PC again, but do not yet Power up the QCX !

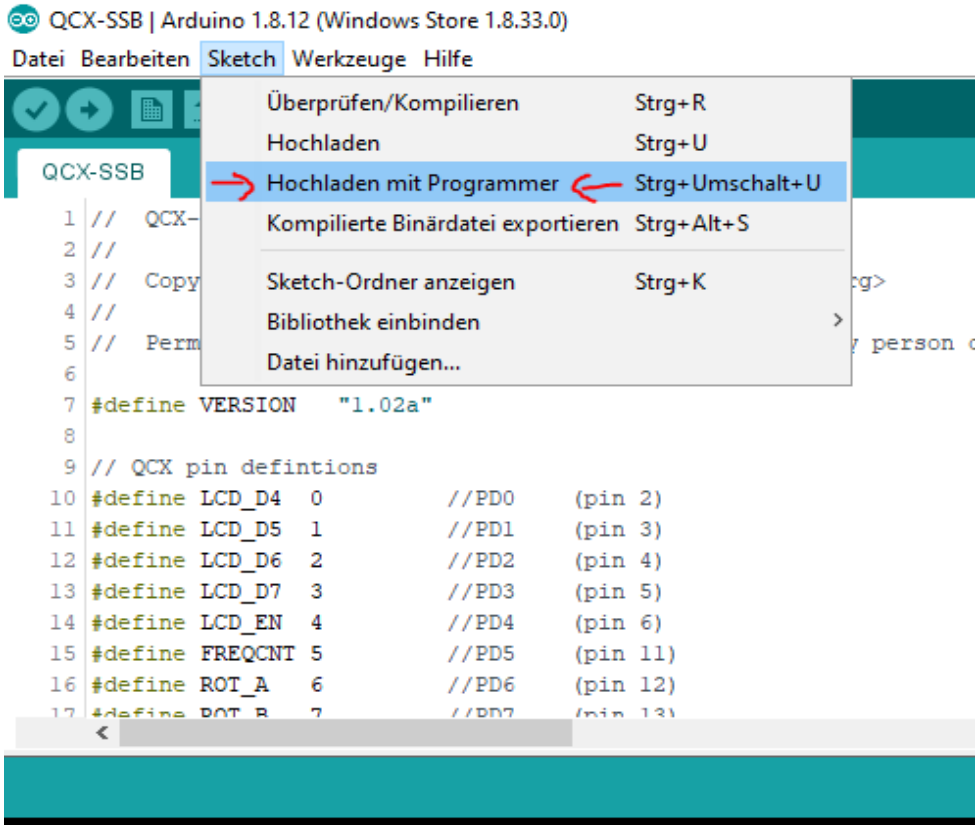
Board Settings and Com Port should still be the same as before, but we need to change the Programmer now, as we use the UNO to program the external QCX now. Choose Tools → Programmer → **Arduino as ISP**



Next Step is to power up the QCX by external Supply Voltage first, and then burning the Bootloader again, but this time to the QCX:



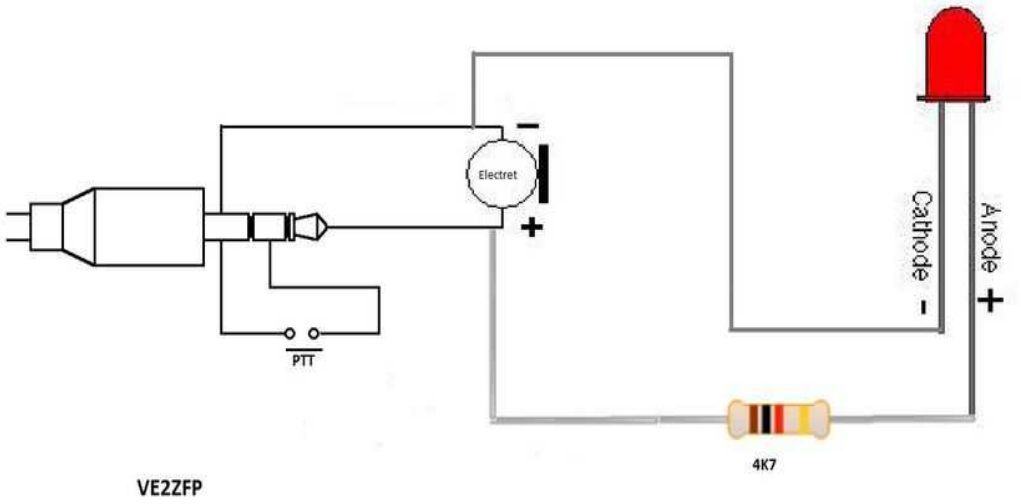
And after that has been done, finally we will send the firmware to the QCX by using **Upload with Programmer**. This is very important, otherwise you´re Programming the Arduino UNO, and not your QCX ! Ask me, how I know this.... ;)



After removing the ISP cable and rebooting QCX, if you did all your soldering correct, and Adjusted your Display Brightness by R47, you should see the Menu of the QCX SSB, and your adventure starts.

To connect a CW Key, you need to solder a special cable or re-solder your plug, because the 3,5mm (1/8 Inch) 3-Pin Plug is not in the usual layout.
Only straight keys or external keyers can be used.
Those need to be connected like the PTT switch (Middle to GND) from Picture below. PTT and Key share a common Pin.

Microphone should be connected as below. **LED and Resistor are optional, not required**.



Picture Source: https://x1m.fandom.com/wiki/Microphone_pinout?file=X1M_mic_pinout_diagram.jpg

I used an old cheap microphone that was left from Yaesu VX-7R. I re-soldered a 3,5mm Stereo Plug, and I had to change the internal wiring to PTT and Mic to match the diagram.

Have fun, and please give feedback via Groups.io with your experiences.
A project like this can only evolve, if many people try it, and give their point of view.

73 Manuel; DL2MAN

Thanks to Guido, PE1NNZ for his modification and his help so far !
Thanks to Hans, G0UPL for his Original QCX Project, and his friendly permission to use his Pictures from the manual.

Versions:
Version 1.01 of the Manual, minor corrections, added Mic and Key (DL2MAN; 08. May 2020
Version 1.0 of the Manual, First Release by DL2MAN
for QCX PCB Board Revision 5 and QCX SSB Firmware 1.02a