

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		
XTAL2	XTAL2	27 Mhz		

Continue with capacitors

PCB Rev5 Label	Place (QCX SSB Schematic)	Value (bold if changed)	Remark	Checkbox
C3	C3	0,1 μ F - 104		
C29	C29	0,1 μ F - 104		
C35	C35	0,1 μ F - 104	Solder a little higher on PCB, for easier step 3	
C36	C36	0,1 μ F - 104		
C30	C30	30pF – 300	for Multiband	
C32	C32	10μF (electrolytic, Minus to GND)	leave out for Guido's magic witchery mod	
C6	C6	0,1 μ F - 104		
C33	C33	1nF – 102		
C41	C41	0,1 μ F - 104		
C34	C34	0,1 μ F - 104		
C40	C40	0,1 μ F - 104		
C42	C42	10nF – 103		
C43	C43	470nF – 474		
C44	C44	470nF – 474		
C45	C45	470nF – 474		
C46	C46	470nF – 474		
C2	C2	0,1 μ F - 104		
C48	C48	0,1 μ F – 104		
C7	C7	1nF – 102		
C4	C4	1nF – 102		
R11	C39	0,1μF – 104		
R58	CXX	220nF	not in Kit !	
C37	C37	10 μ F (check Polarity)		
C38	C38	10 μ F (check Polarity)		
C51	C51	10 μ F (check Polarity)		
C47	C47	470 μ F (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	leave out for Guido's magic witchery mod	
R42	R42	1K / in Rev 5 1,2K	leave out for Guido's magic witchery mod	
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		

Remark: If you don't have 82K Resistors, you can put in Series:

$$68K + 15K = 83K$$

$$56K + 27K = 83K$$

Most important: R7 and R10 need to be the same Value.

Less then 82K = Less Gain / More then 82K = More Gain, but you lose dynamic Range.

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 T37-2 -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	leave out for Guido's magic witchery mod	
Q1,Q2,Q3, Q4,Q5	Q1,Q2,Q3,Q4 ,Q5	BS170	leave out Q4 for Guido's magic witchery mod	
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/Earphone	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		

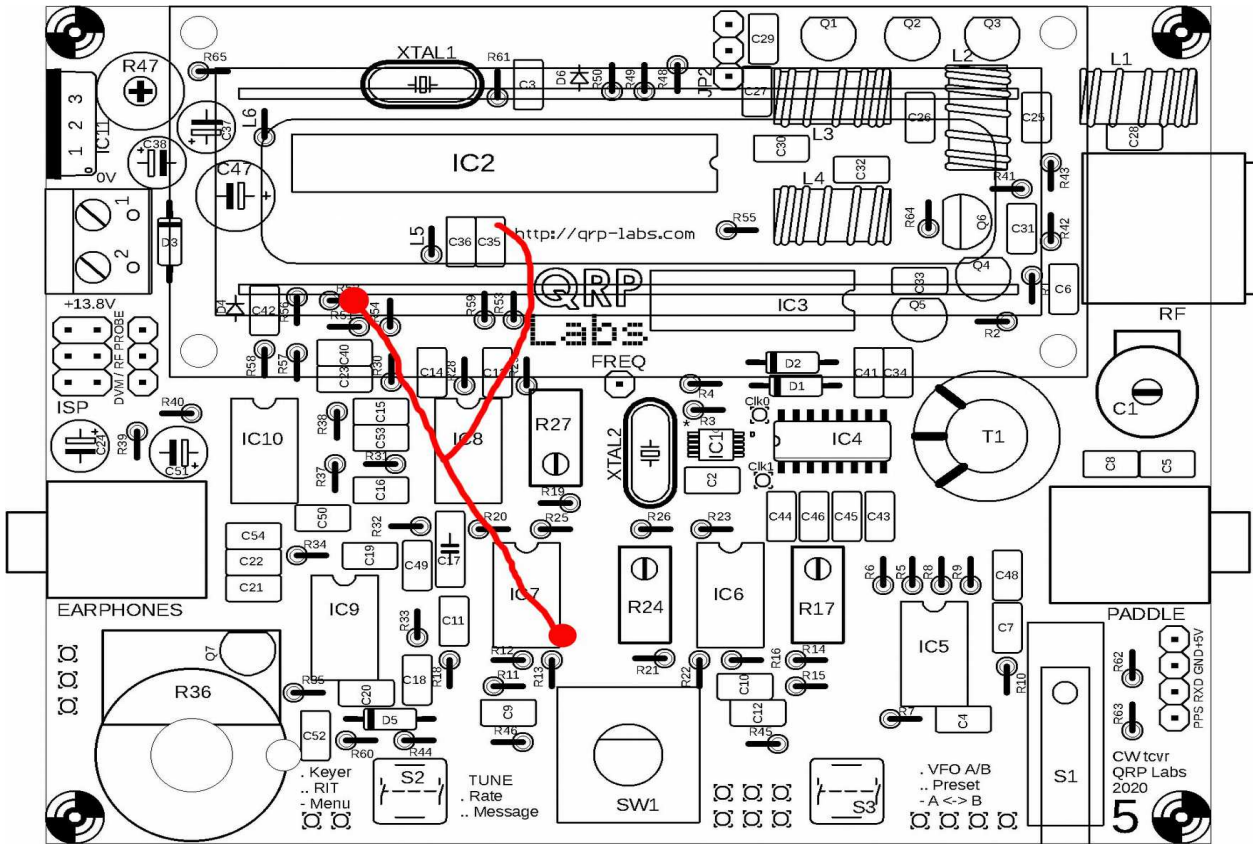
You can also put in the screws and spacers for the display, and solder in Male/Female 16 Pin connector to your Display PCB.

Step 2:

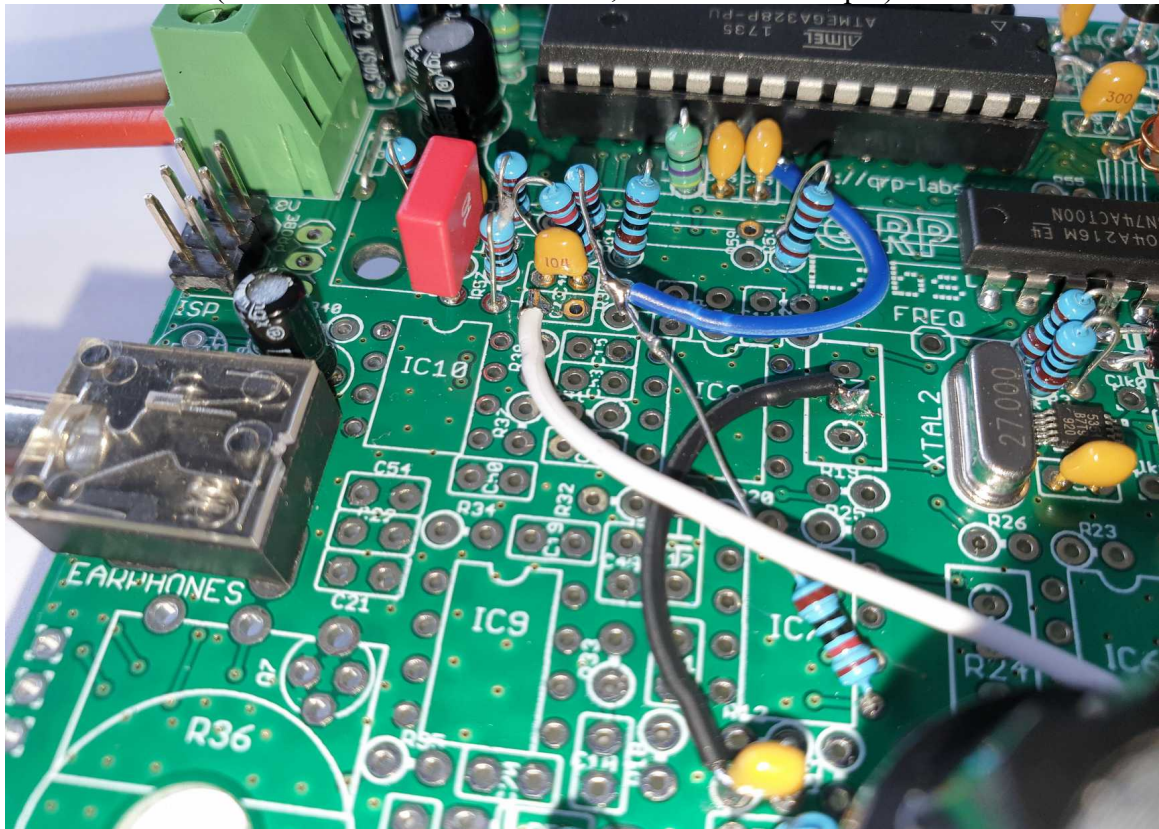
Place 10K resistor at right side of PCB Label R52

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)



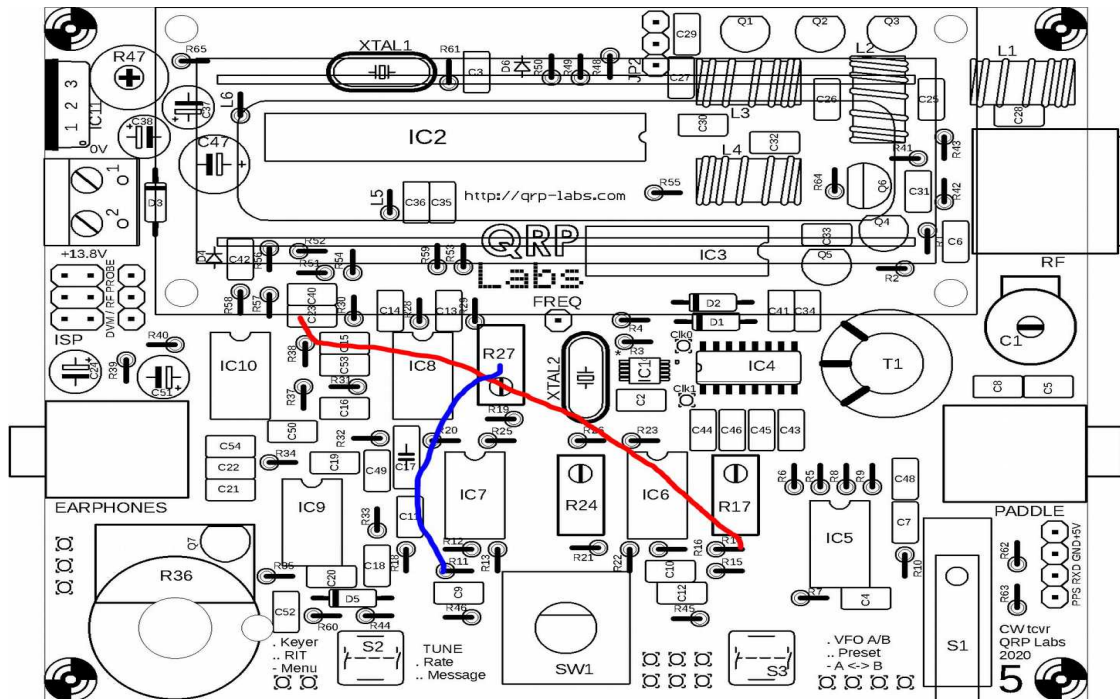
Should look like this (without black and white wires, which will be step 3):



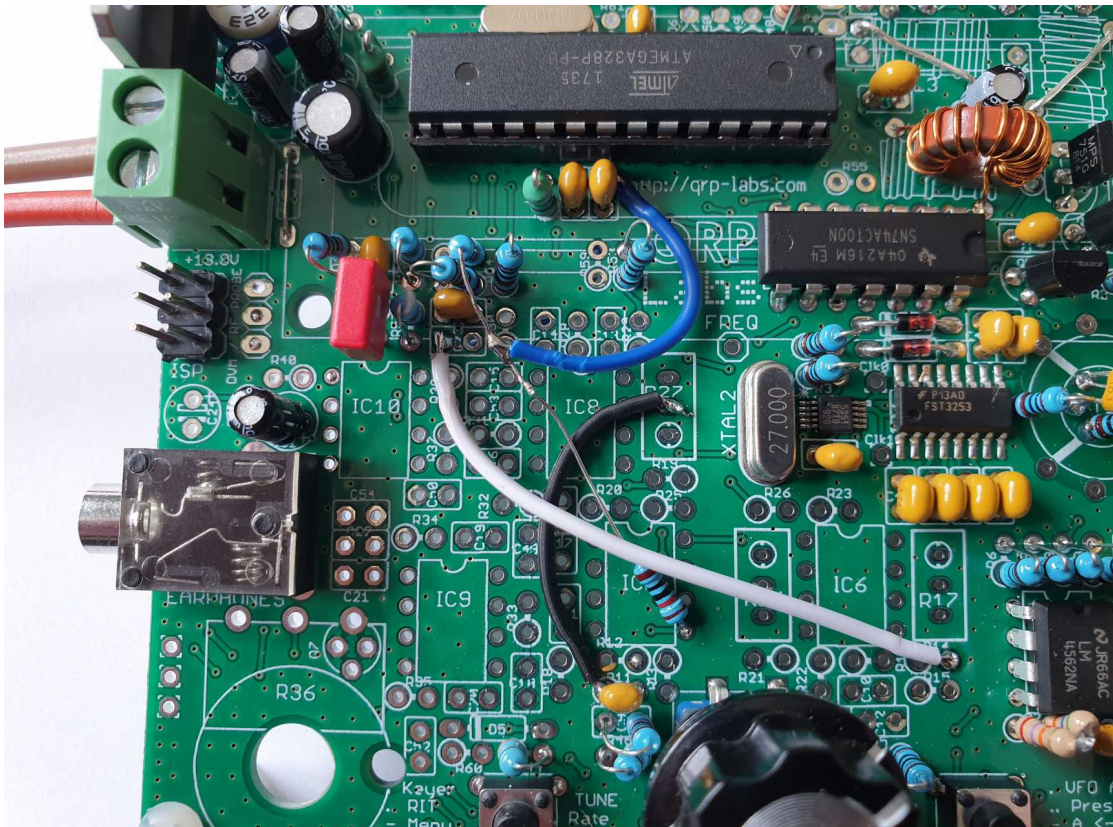
Step 3:

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)

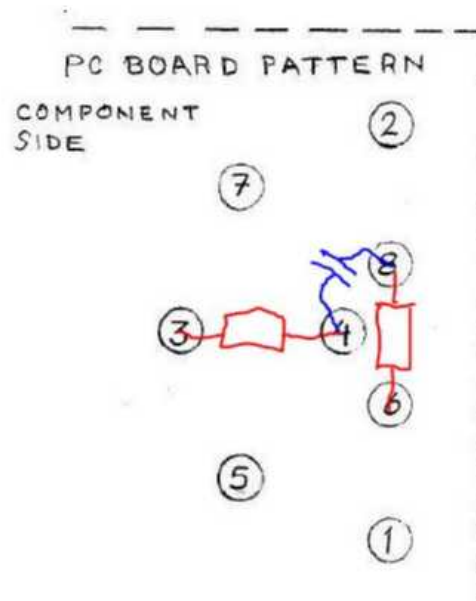


Step 4:

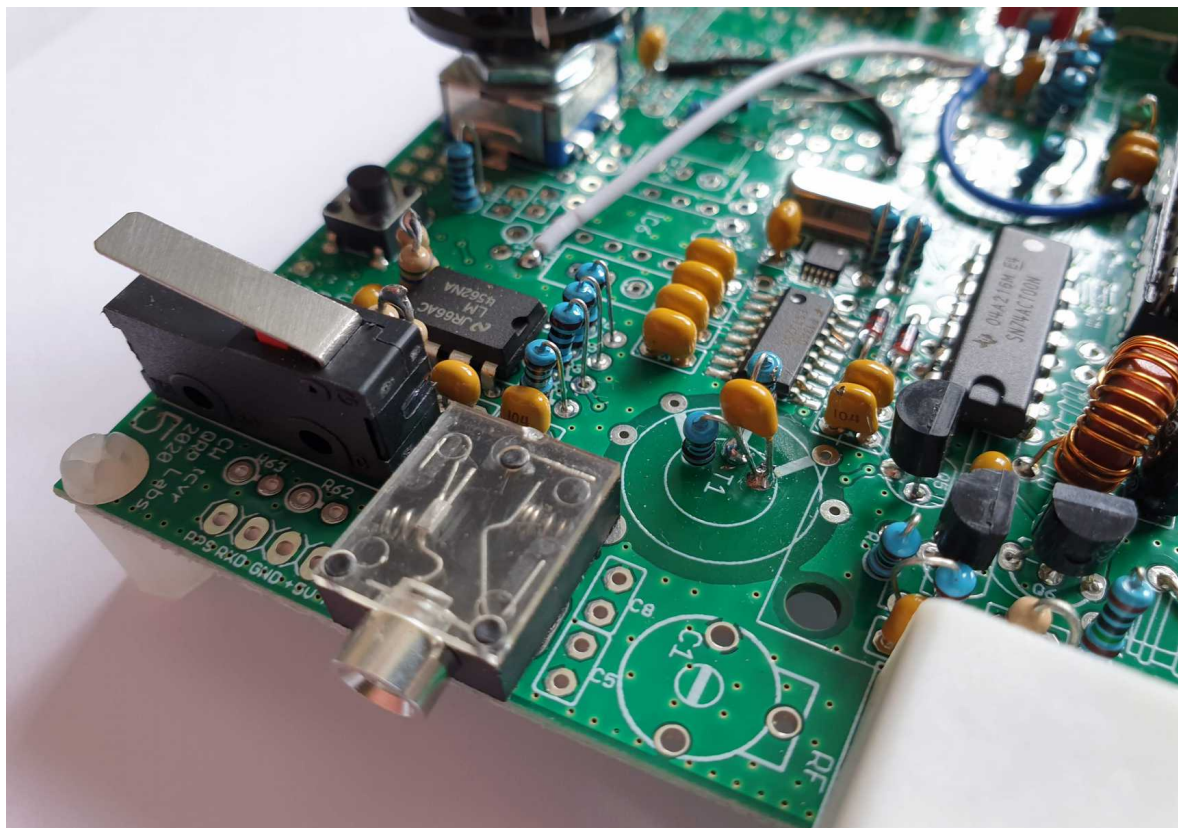
Place Resistor 1K between Pins 3 and 4 of PCB Pattern T1

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:

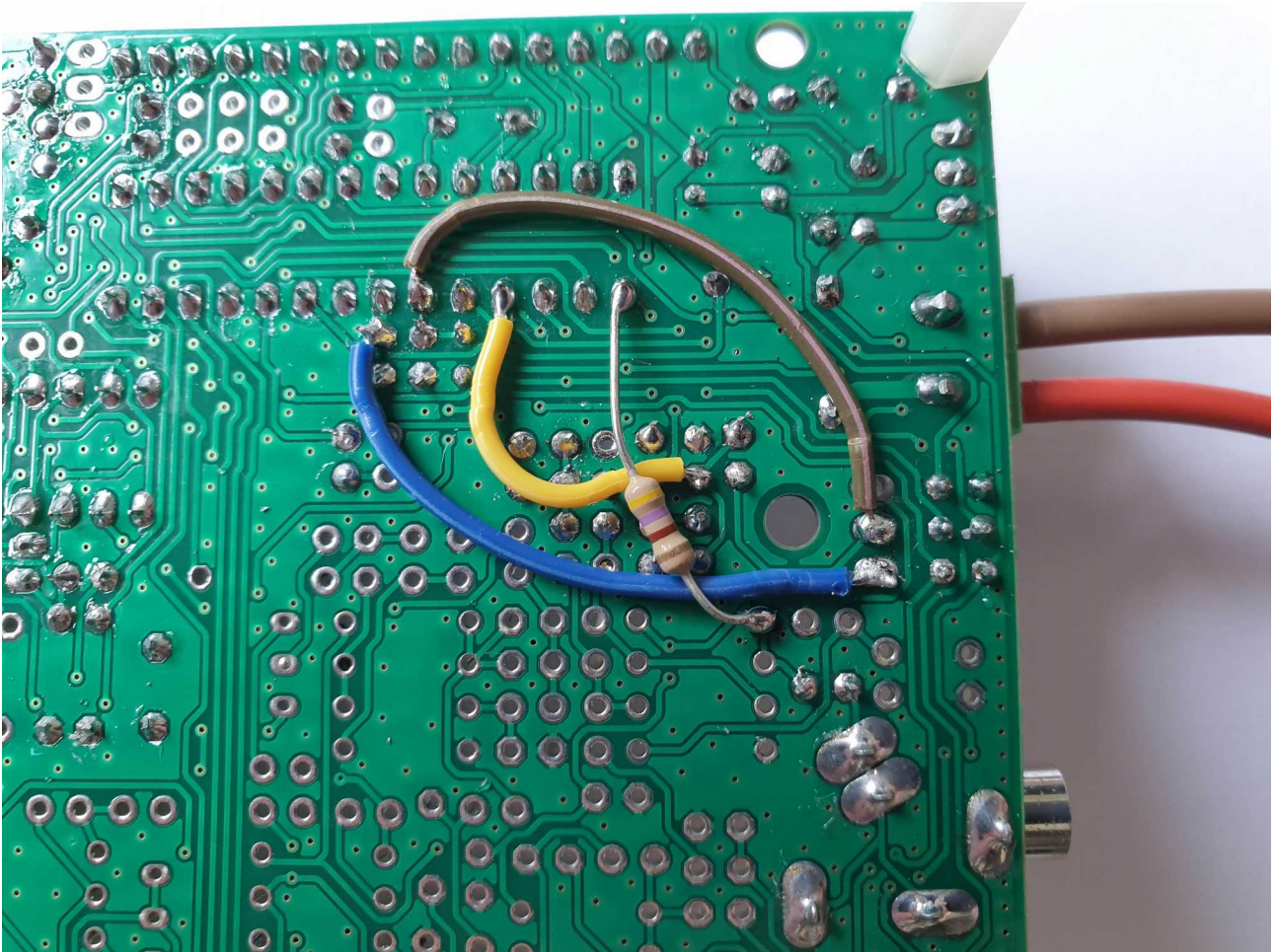
Colors are up to you, but for description I will refer to my chosen colors....

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. If you prefer passive speaker, place another wire instead.



Step 6:

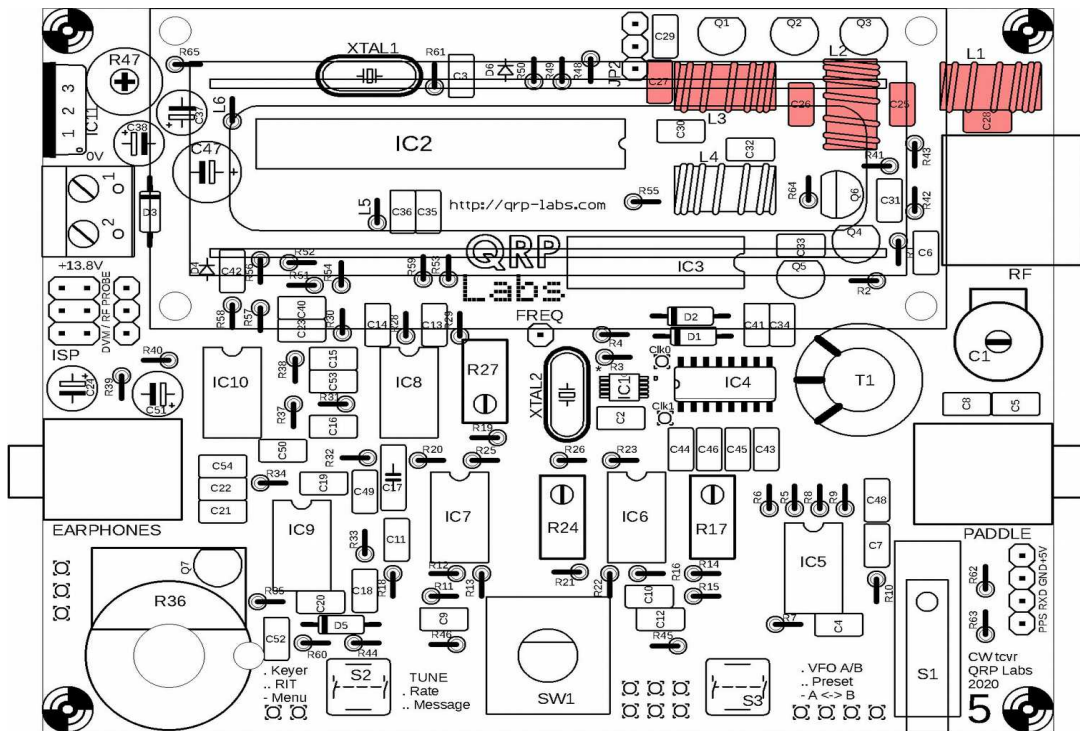
Now we need to take care about the Low Pass Filter.

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.

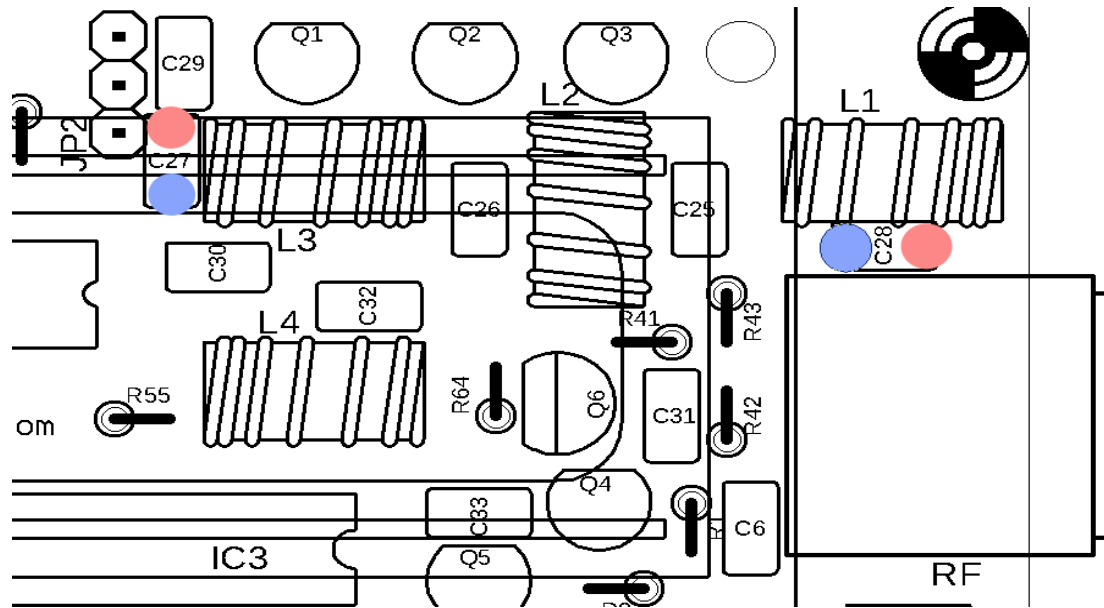


I've taken all those possible values for QCX LPF, and put them together in one Table for you:

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



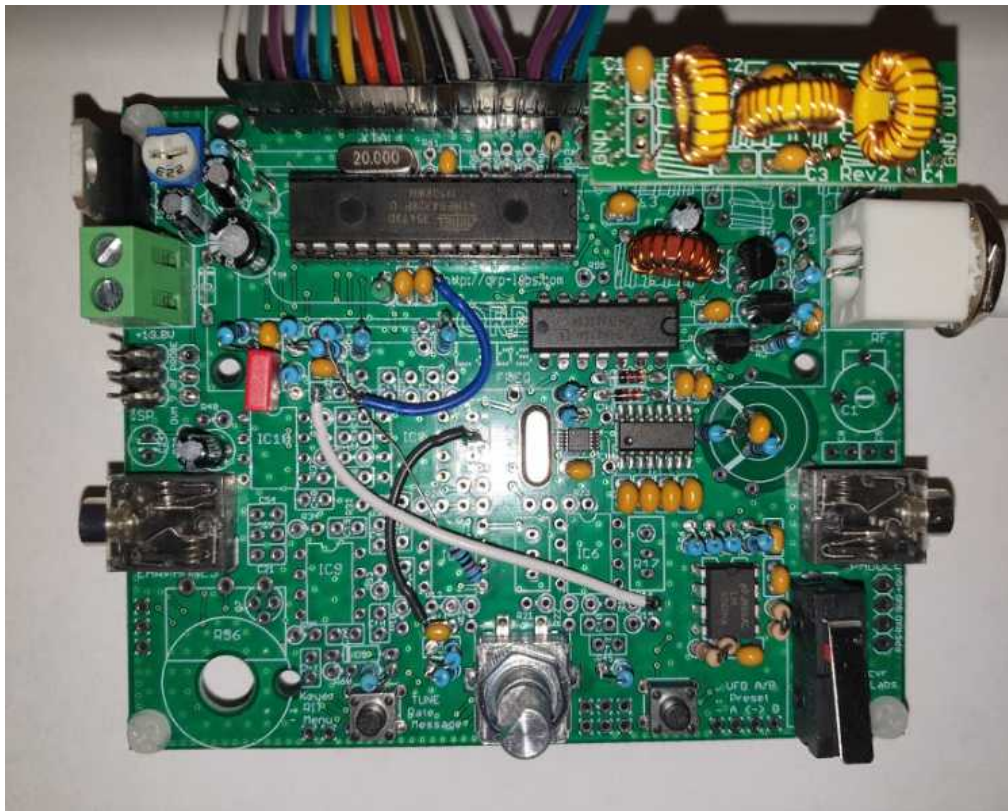
Use PCB C27 to feed Input of external LPF

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.



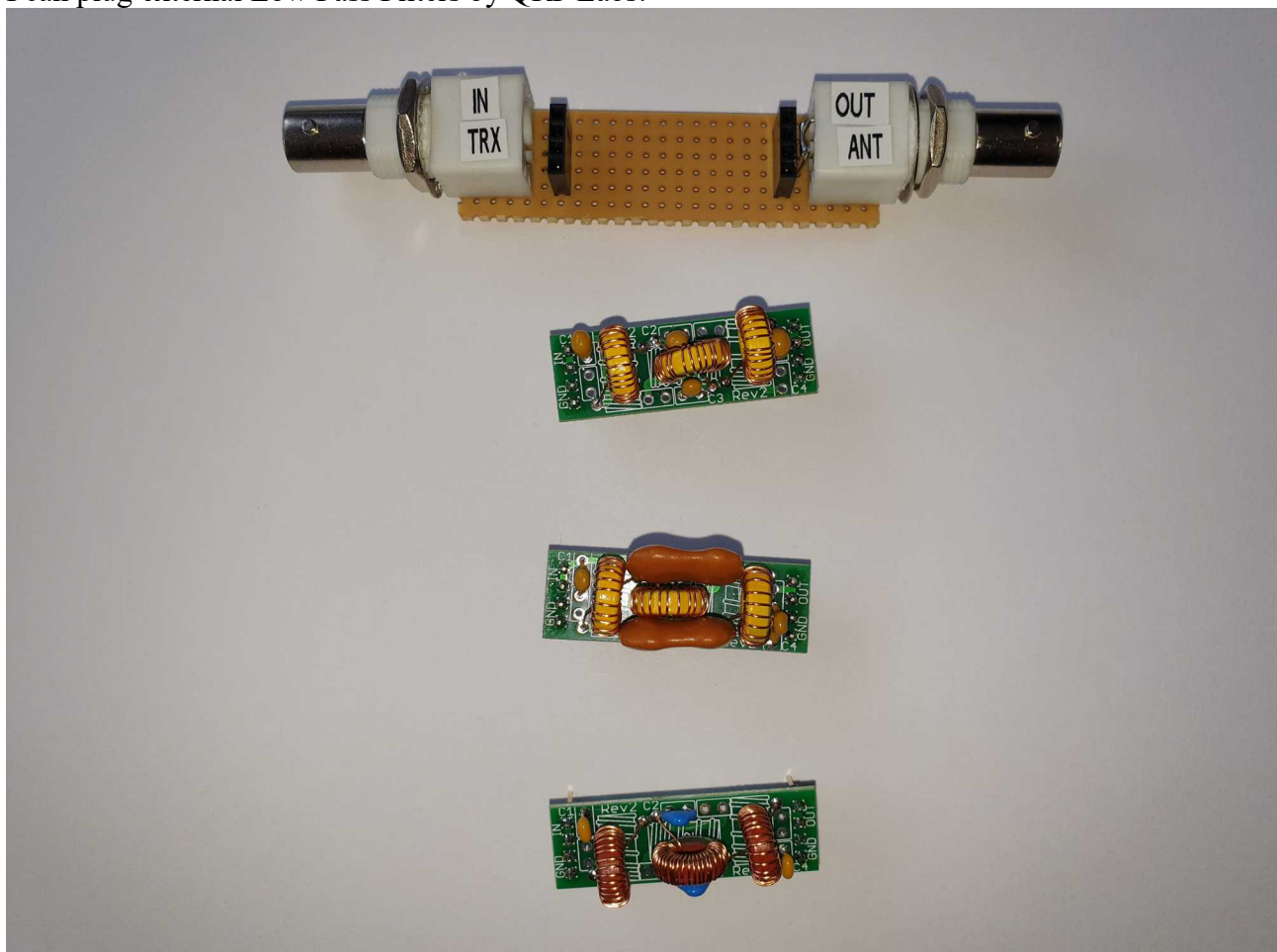
Option 6c)

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:



Optional Step 7: Guidos secret magic witchery mod..... ;)

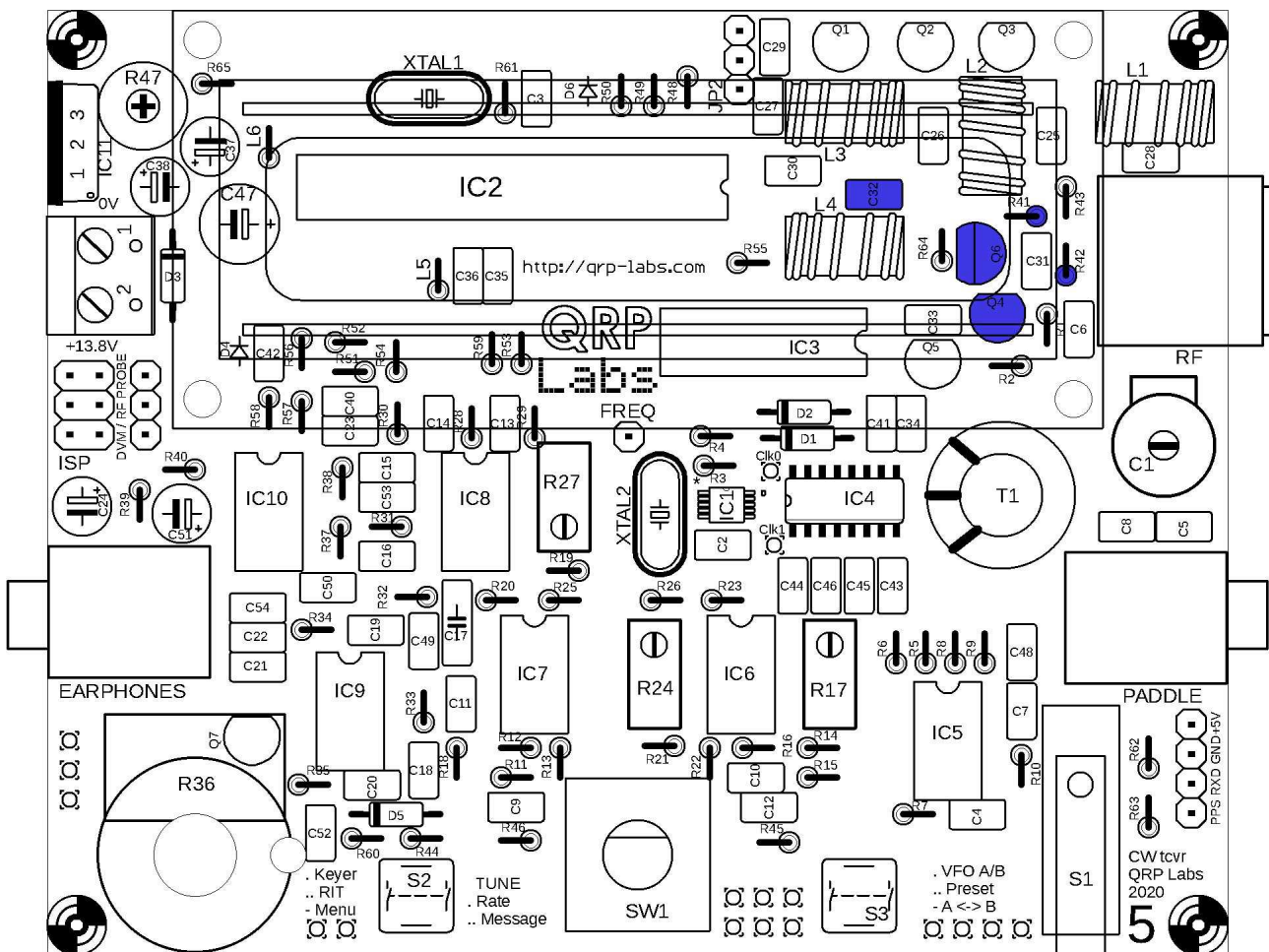
Sounds interesting, uh ? I was aware of this Version of the Schematic, but since it was only a footnote in Guido's original instructions, I ignored it. Bad Mistake..... (See Notes 3 on his Github Page)

It uses even less components, and allows software control over working point of PA. This way you can tweak for optimum efficiency of your PA by Menu Setting 8.3 "PA Bias Max". I was able to get more Output-Power with less Input Current. Technically it adjusts the gate voltage of you PA Fet's. **Recommended Maximum Value for this setting is 128.** Less is more. You have to measure DC input Current at the same time as Output Power, **to find your optimal working point.**

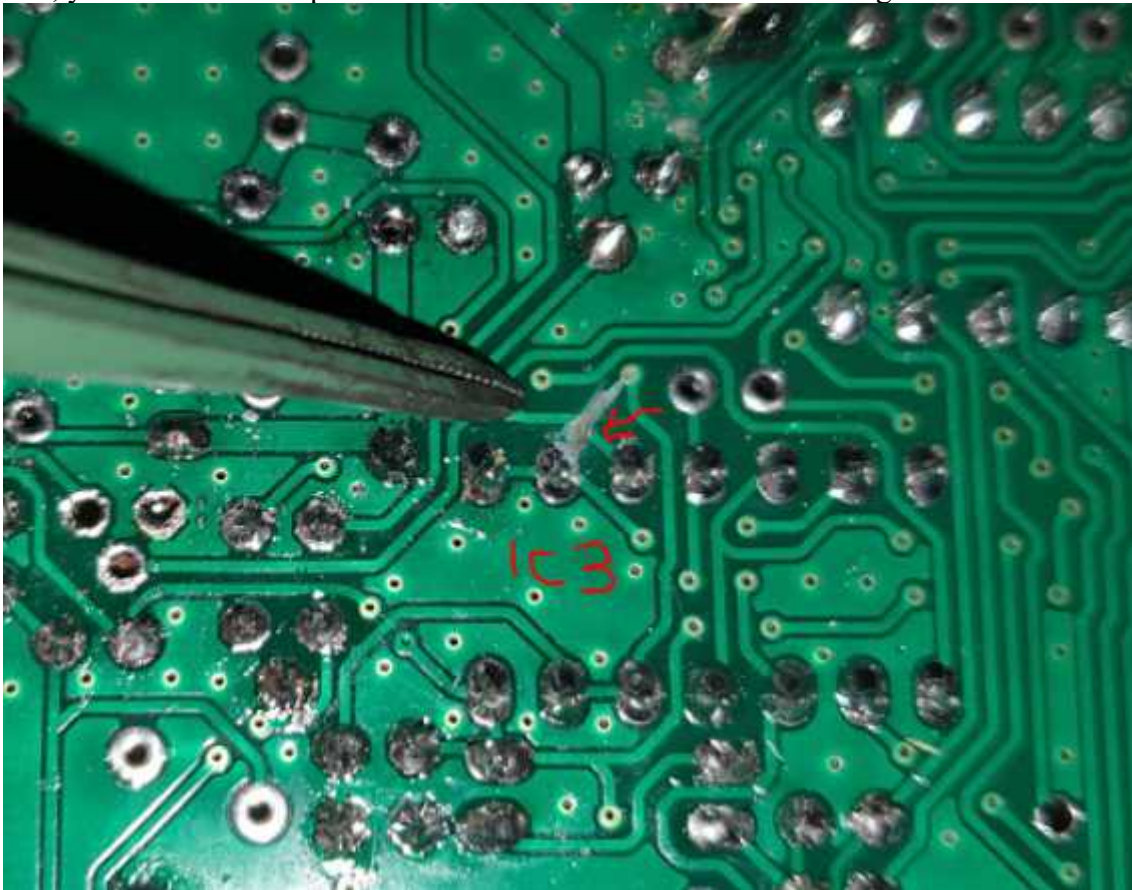
But unfortunately it requires destroying one PCB Trace. So no way back from here....

If you already built the QCX SDR/SSB from one of my earlier Versions of instructions, you need to remove a few Parts: Q4, Q6, C31, R41 and R42.

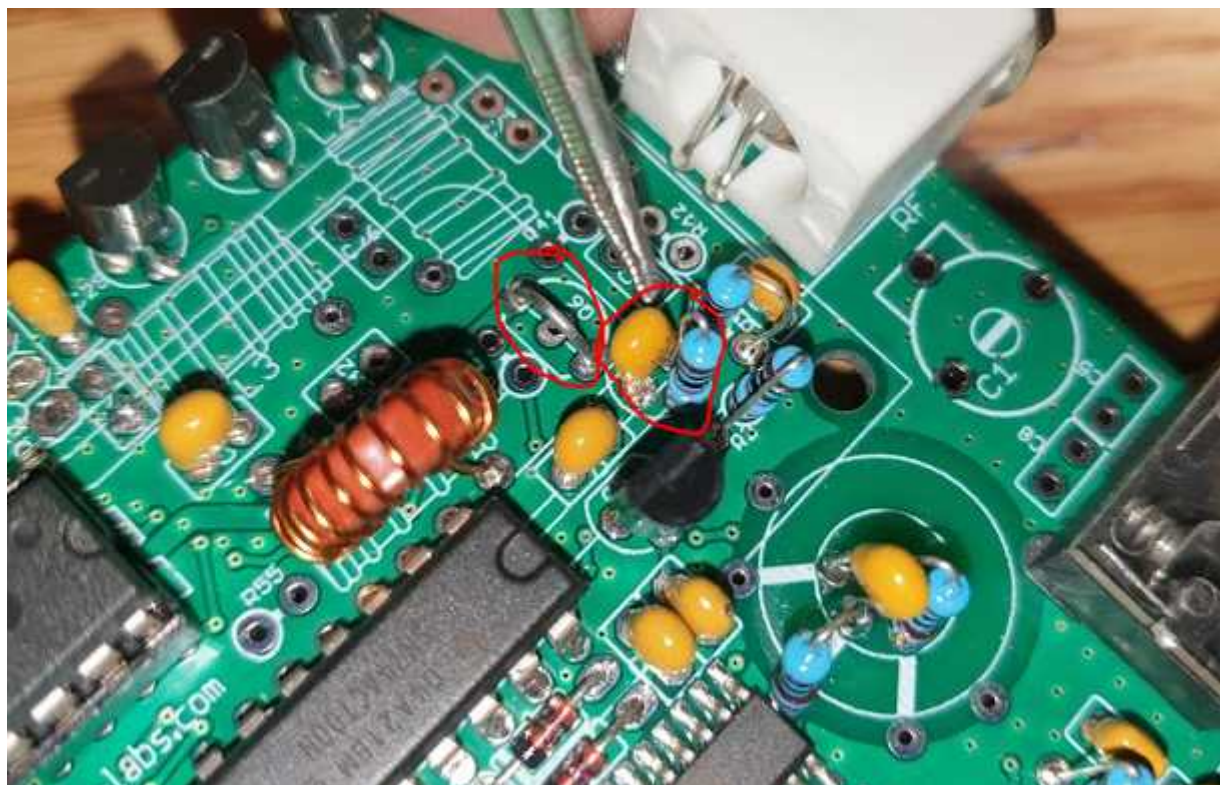
If you're building it from Zero, you probably saw the remarks (Leave out for Guido's magical witchery mod)



After that, you need to interrupt the Connection at Pin 3 of IC3 at soldering side of PCB:



Solder in Bridge between outer Pins of Q6 Pads. Place 10nF Capacitor into outer Pins of PCB Q4, Place Resistor 10K from Middle Pin of Q4 to Outer Pin of Q4 (BNC-Jack Side of PCB)



Next: Solder 10nF Capacitor between Pin 3 of IC 3 and Gate-Line of PA-FET's Q1-Q3,
Solder in 10K Resistor between Outer Pin of PCB Q4 (the Pin with already 2 wires through hole)
and Gate Line of PA-FET's Q1-Q3.



Ignore the Jumper Connectors, which might not be there at your PCB's. I epoxied them in Place as Socket for my external LPF's (See Step 6b).

And the Mod is finished.

Step 8:

Making it alive.... or putting firmware on it....

Before that, you should of course place the ATmega into the 28 Pin Socket.

Programming it is actually pretty simple, if you carefully follow step by step.

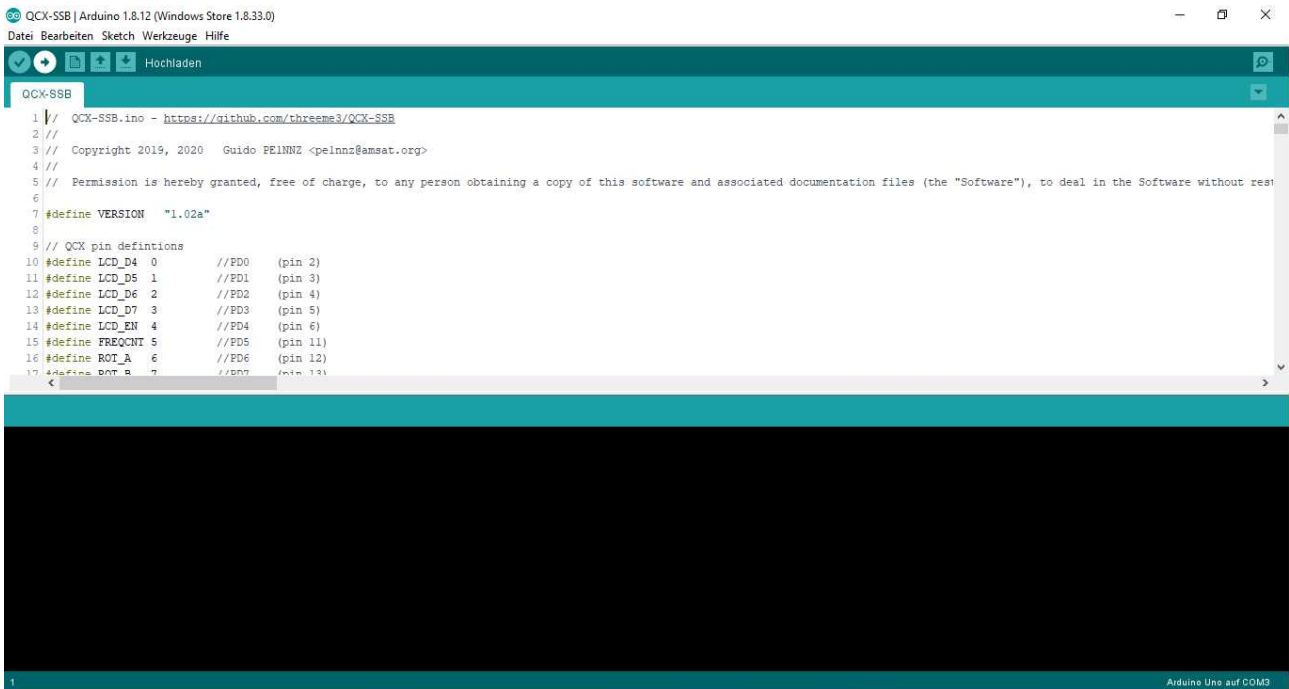
First you need to install Arduino IDE (Source: <https://www.arduino.cc/en/main/software>)

Then you need to Download the Firmware File „QCX SSB.ino“ from

<https://github.com/threeme3/QCX-SSB>

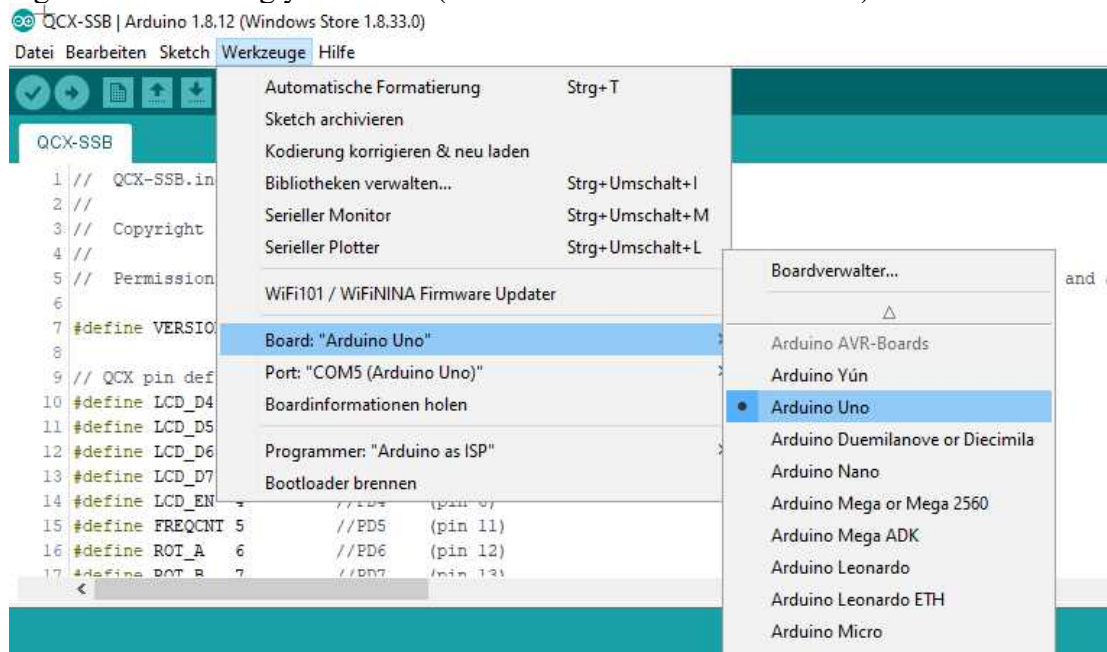
When you're using Guidos source code (The INO File, instead of the Hex File) there's no need for an additional software tool..

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 ;)

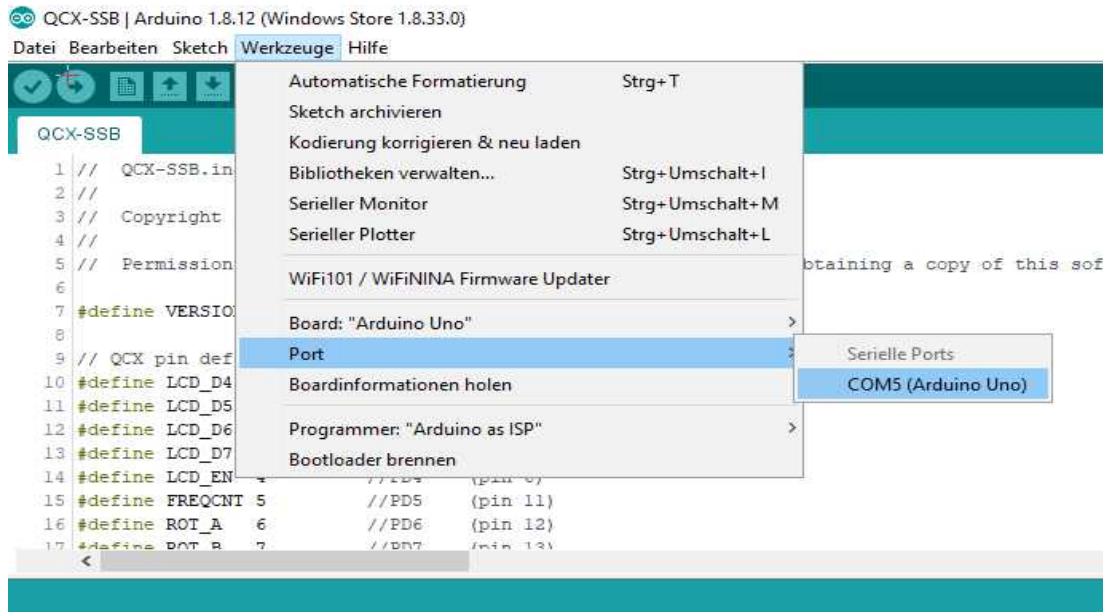


```
1 // QCX-SSB.ino - https://github.com/threeme3/QCX-SSB
2 //
3 // Copyright 2019, 2020 Guido FELINNZ <felinnz@amsat.org>
4 //
5 // Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without res
6 //
7 #define VERSION "1.02a"
8 //
9 // QCX pin definitions
10 #define LCD_D4 0 //PD0 (pin 2)
11 #define LCD_D5 1 //PD1 (pin 3)
12 #define LCD_D6 2 //PD2 (pin 4)
13 #define LCD_D7 3 //PD3 (pin 5)
14 #define LCD_EN 4 //PD4 (pin 6)
15 #define FREQCNT 5 //PD5 (pin 11)
16 #define ROT_A 6 //PD6 (pin 12)
17 #define ROT_B 7 //PD7 (pin 13)
```

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



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

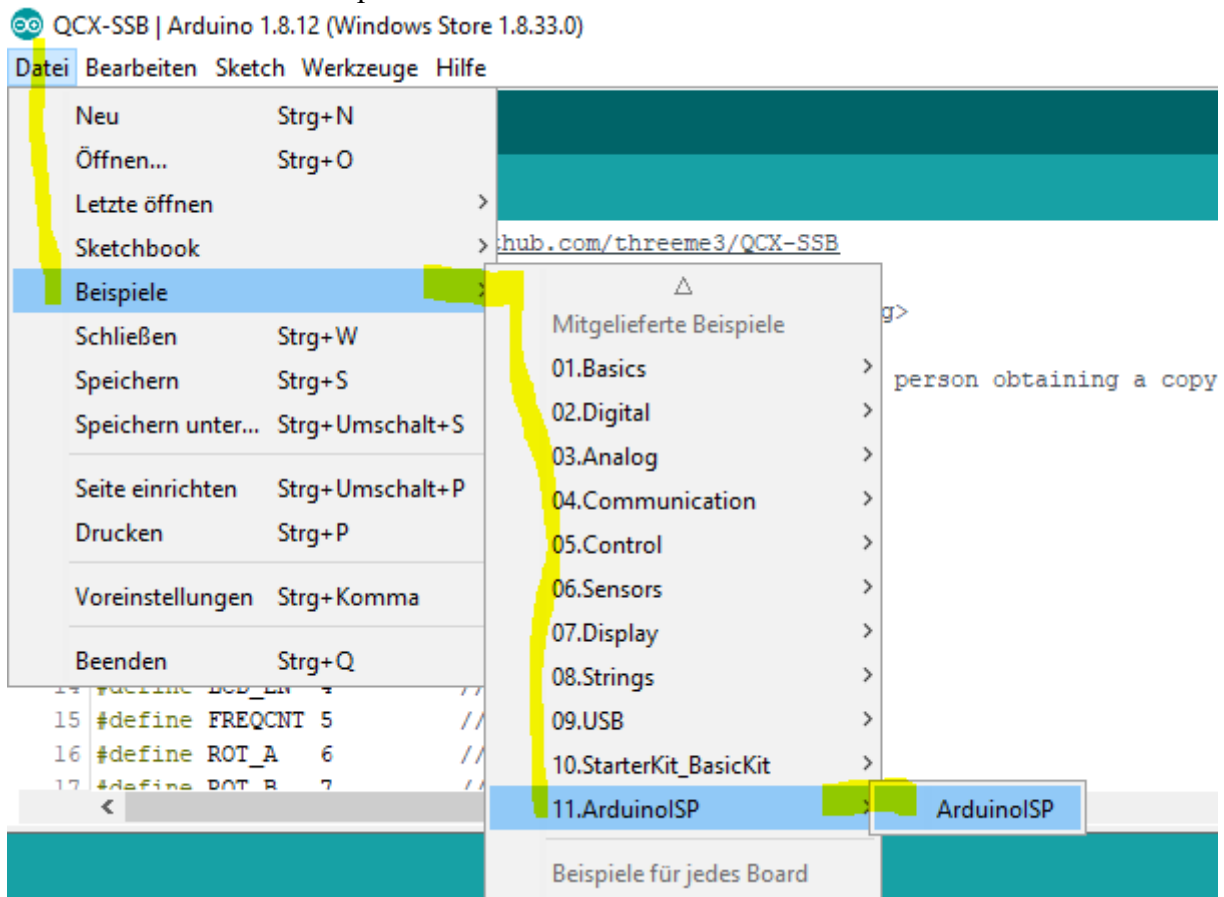


After that, you should see something like this in the lower right corner of the Arduino IDE Window:

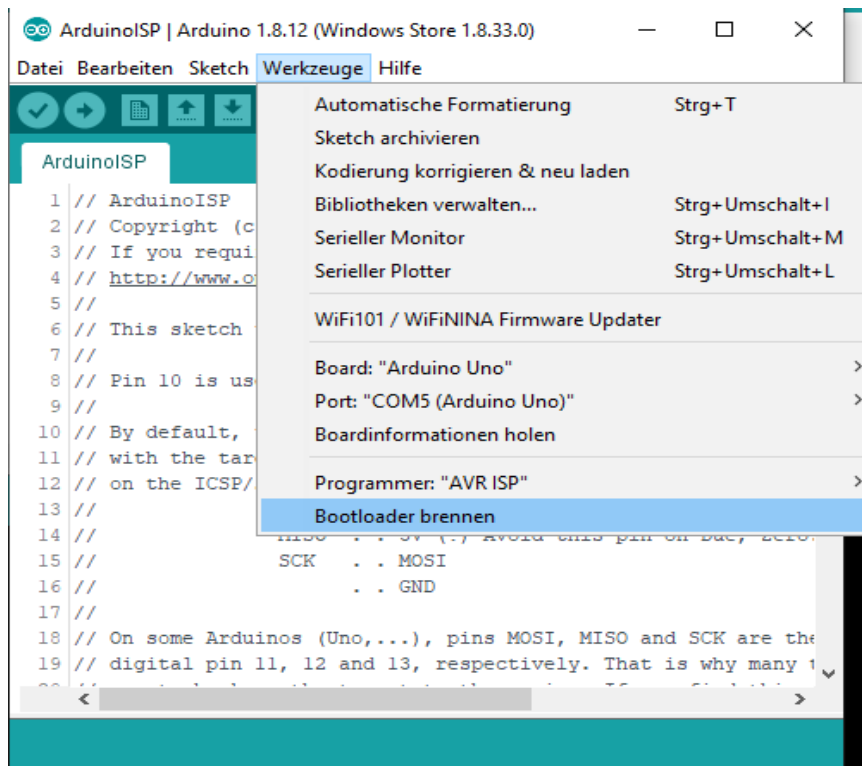


Next, you need to prepare the Arduino UNO, so it will serve as Programmer for the QCX.

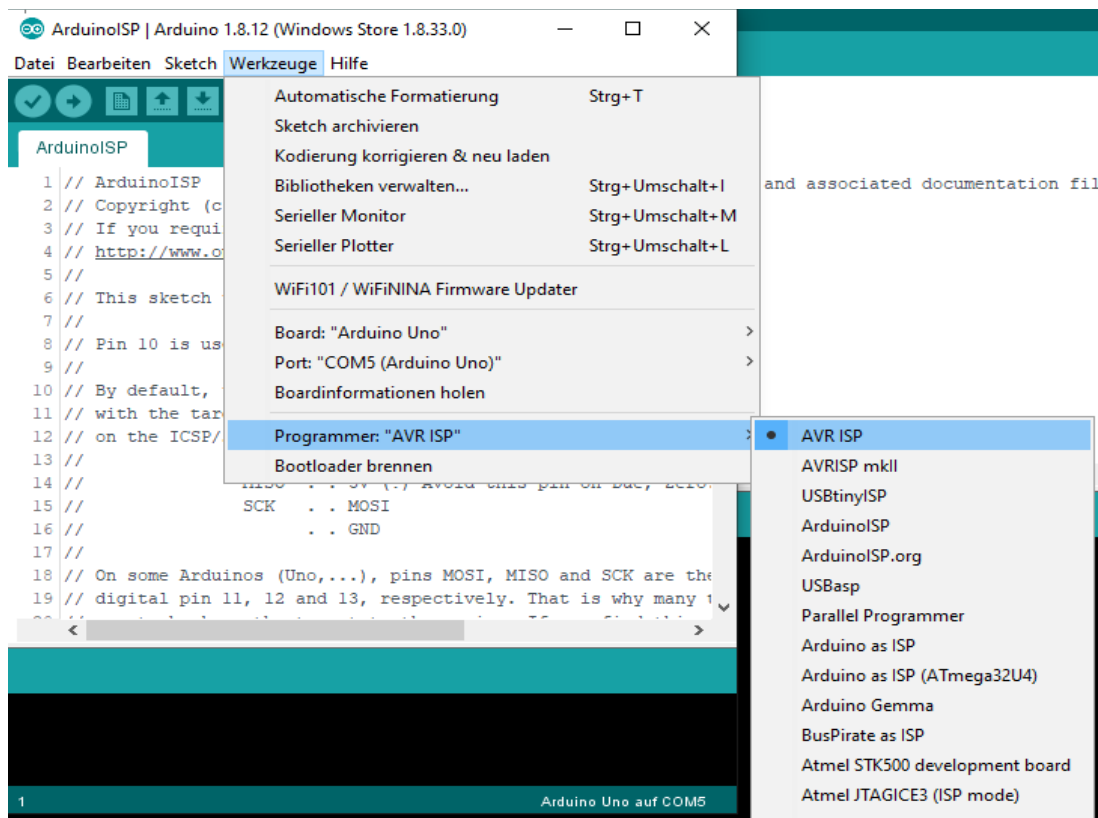
To do so choose File → Examples → ArduinoISP → Arduino ISP



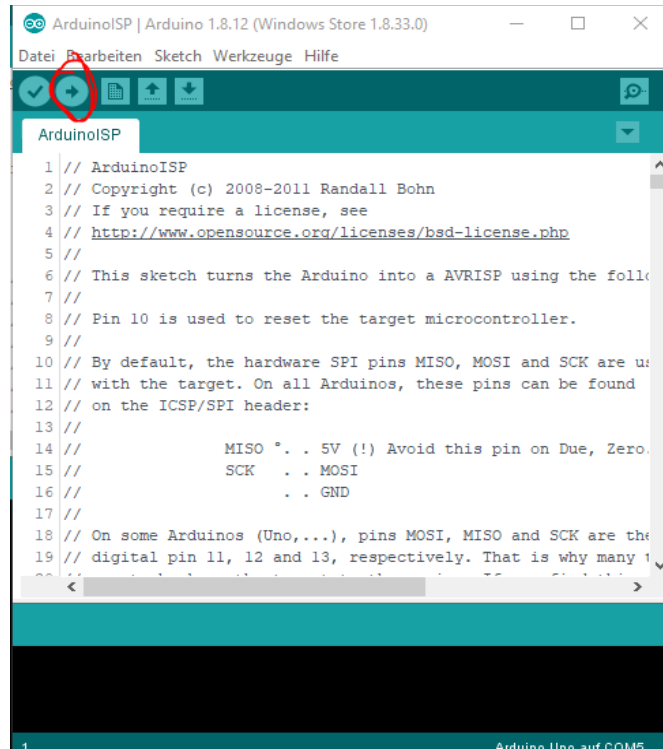
A new sketch window will open.
In this Window, first select Tools → Burn Bootloader.



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



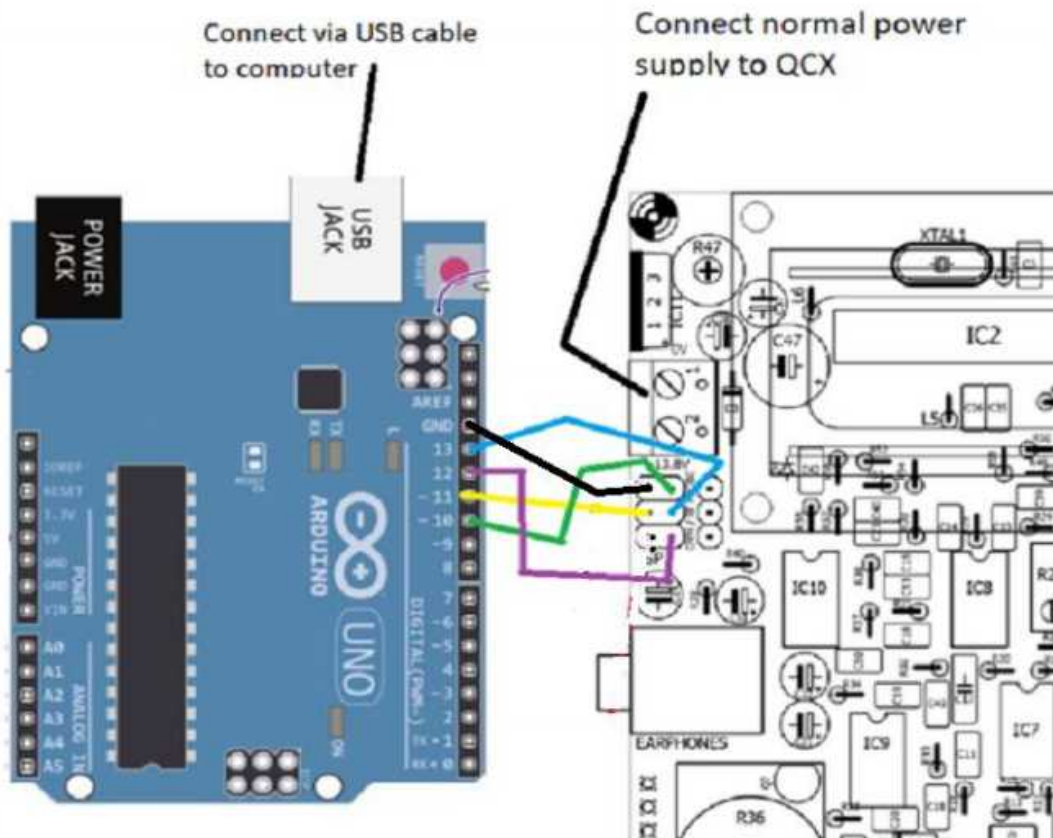
Then you need to push this button:



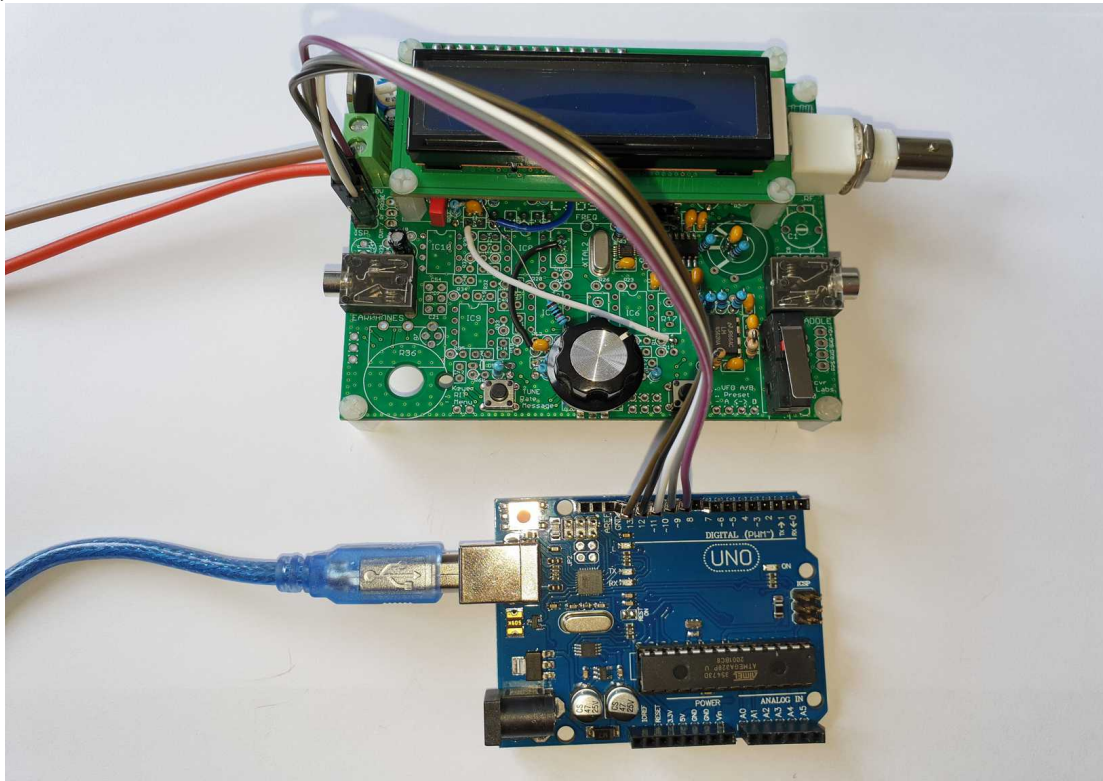
and after a few seconds, the Arduino UNO will be ready to serve as Programmer for our QCX. We can close our Arduino ISP Window now, as we don't need it anymore, and we now should have only QCX SSB Window remaining.

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:



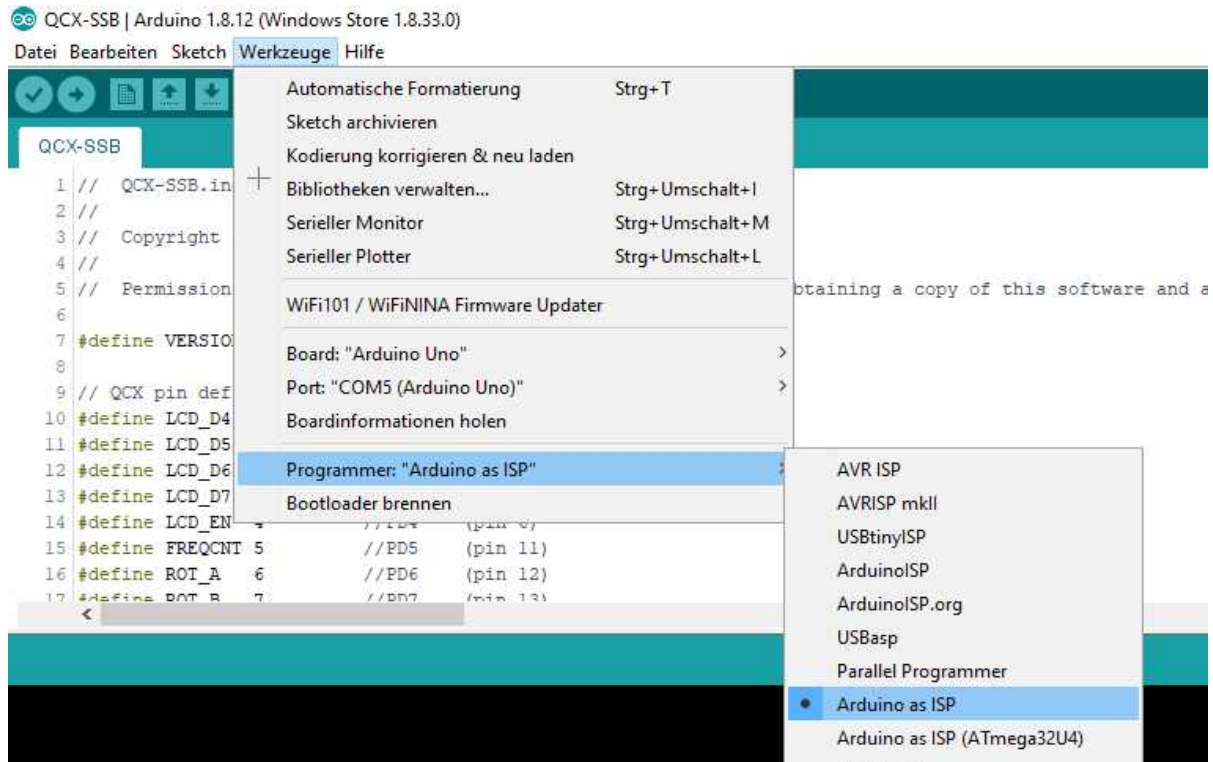
After you have double checked, that the wiring is right, you can connect Arduino's USB cable to your PC again, but do not yet Power up the QCX !

Tip:

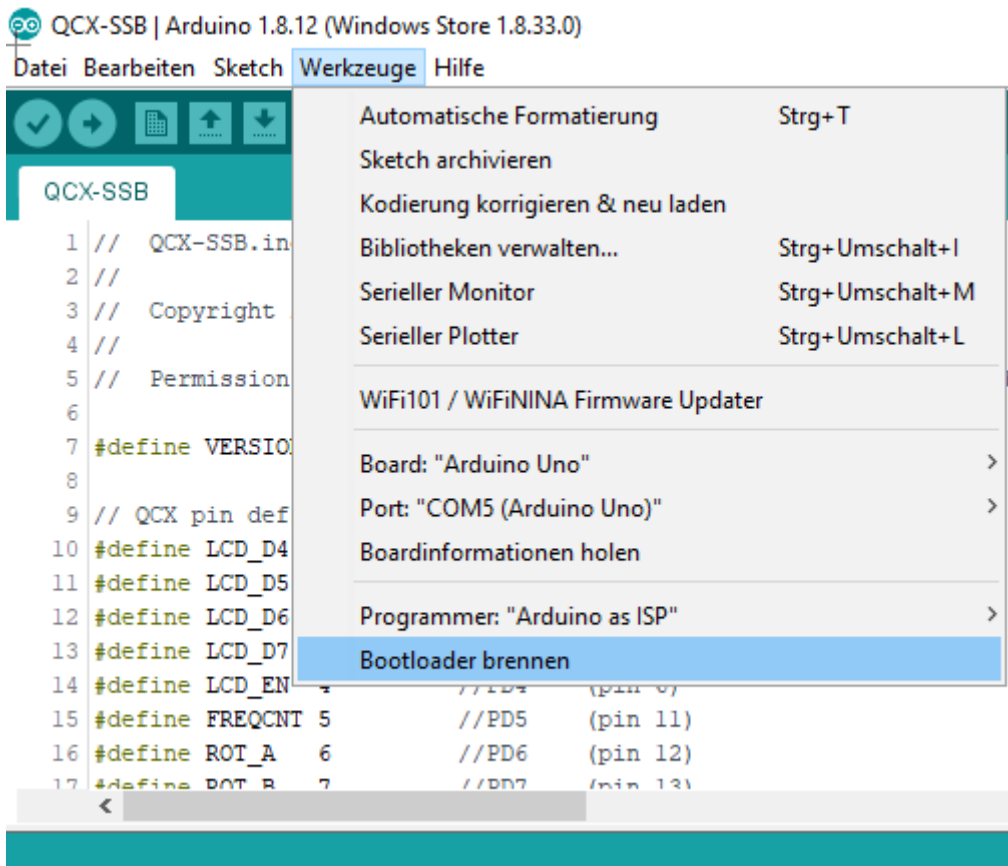
Once you have placed the Pins on the ISP interface, you can tape the pins together. This way you have a cheap "connector" and do not need to fiddle around for future Updates.
Just remember the orientation.....



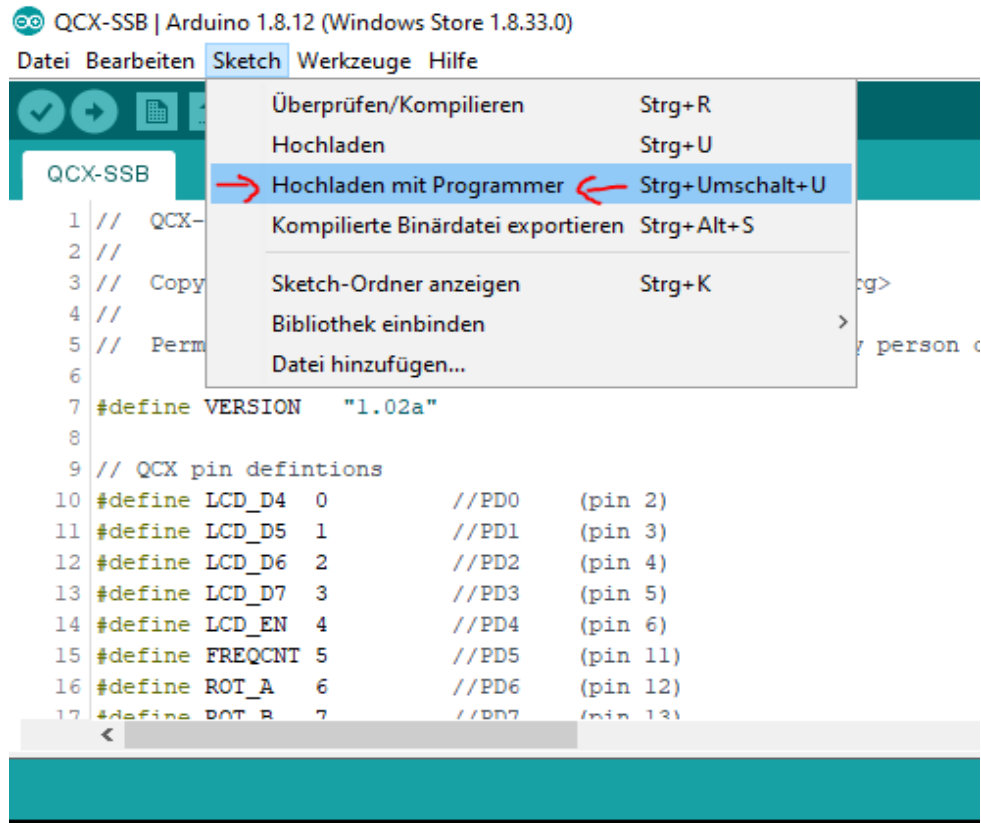
Board Settings and Com Port should still be the same as before, but we need to change the Programmer now, as we use the UNO to program the external QCX now. Choose Tools → Programmer → **Arduino as ISP**



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.... ;)



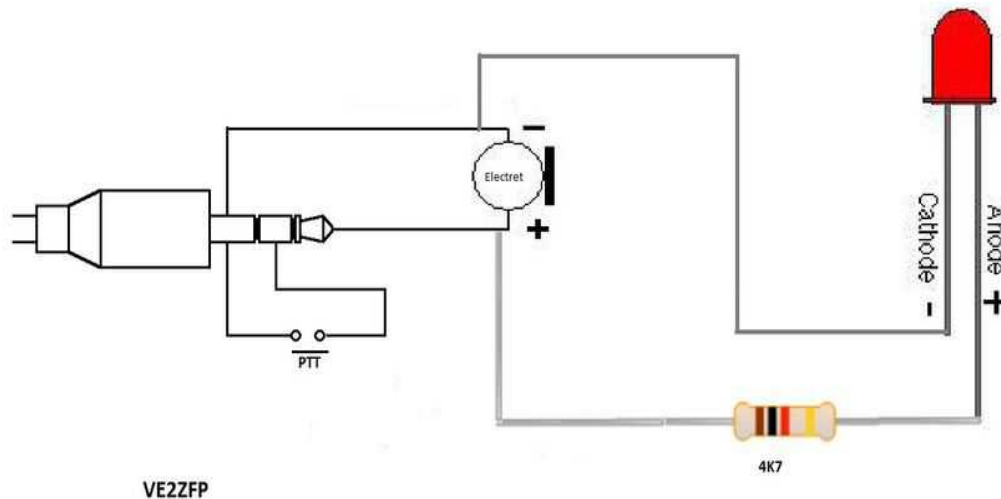
And after removing the ISP cable and a reboot, if you did all your soldering correct, and Adjusted your Display Brightness by R47, you should see the Menu of the QCX SSB, and your adventure starts.

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.

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.

The future of this Project is the μ CX (micro controlled Transceiver), which we want to build on with a designated PCB. Join the discussion at <https://ucx.groups.io/>

Versions:

Version 1.1, Added “Guido’s magic witchery mod

Version 1.02, regaining missing X2, added wire instead of Resistor for audio path, added Tape-Tip, changed L4 Toroid (tnx Ryan Flowers)

(DL2MAN, 20 May 2020)

for QCX PCB Board Revision 5 and QCX SSB Firmware 1.02d

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