Assembly Manual for the µSDX „Sandwich" Design by DL2MAN (Software by PE1NNZ)

The µSDX is a simple but very efficient QRP Multimode Shortwave Transceiver, that is able to transmit and receive SSB, CW, AM and FM. It can be configured for any Shortwave Band by exchanging/modifying RF PCB. Now even as Multiband Rig with switchable Low-Pass Filters.

**LICENSE:**
The uSDX (Basic Schematic and Software by Guido; PE1NNZ – PCB Design, Layout and Class E Muliband Circuit by Manuel; DL2MAN) is free to use, alter and built
**FOR NON COMMERCIAL USE ONLY** !
If you alter the PCB Layout or the Source Code, the Source must be mentioned if re-published. The same applies for re-publishing it un-altered. Name the source !
R&D took a lot of our free time, and a lot of our hard earned money. We´re happy to share the results for free with you, but won´t accept, if people make money out of other peoples work.
People interested in commercial use of this Product (or Parts of it), need to ask for permission (DL2MAN@gmx.de for Hardware, pe1nnz@amsat.org for Software) .

New Software will be released here: https://github.com/threeme3/QCX-SSB
Discussion about this Project here: https://groups.io/g/ucx/topic/sdx*breakout*board/75316472?p=,,,20,0,0,0::recentpostdate%2Fsticky,,,20,2,0,75316472

The DL2MAN „Sandwich" Design was developed to be able to do changes and add on´s without re-inventing the wheel every time. Kind of a modular concept, which proofed to be working, when you look at the development history. From Monoband „Ghetto" over Multiband „Ghetto" to Serial Resonance Class E Multiband with 80 PLUS % Efficiency. The circuit is devided into 3 PCB´s in Size of a Credit Card (85x55mm) so it´s easier to change things in future, and the size was chosen to proof how small it could be built. The uSDX could be even built smaller, but for now I want to stay at that form-factor.

We have
1. Main PCB (with AtMega, PLL, Multiplexer, OpAmp and NAND Gates, Basically the complete RX/TX Core)
2. IO-PCB (With all Knobs, Rotary Encoder, Audio and Mic/Key Jacks)
3. RF PCB with PA and 5 switchable highly efficient, tuned serial resonance, Class E circuits (80m/60m/40m/30m/20m)

IO PCB is Top of the Sandwich, Mainboard is middle and RF PCB is bottom in the final assembly.

PCB´s are beeing connected via standard Jumper Headers and Sockets in 2,54mm/0,1" Spacing.
The Main PCB requires special „stackable" Headers, that have „male" and „female" connector, so we can connect the other PCB´s from both sides.
The Main-PCB was re-designed into 4-Layers, so there is a complete GND and VCC Layer between the visible Layers. This is not neccessary, but helps in avoiding Ground Loops and reduces Noise in RX of our digital TRX.

The PCB´s are mechanically stabelized by Spacers in M3 Threading. Use metal spacers (no Nylon), so they will provide additional GND Connection between PCB´s and are very handy to connect for example Oszilloscope GND for measurement.

**Where to get PCB´s ?**

The PCB´s can be ordered from https://jlcpcb.com/ by uploading the zipped Gerber Files from the „PCB-Files" Folder.
The Process starts with the PCB Dimensions (55x85mm), choosing 2 Layers in case of IO PCB and 4 Layers in case of Main PCB and RF PCB.
Thickness 1,6mm is standard and will do the job.
They  Gerber Files do not need to be unpacked. Just upload the Zip File in the next step.
Each zip File is one PCB. The Gerber Files were all tested, and the resulting PCB´s from JLCPCB were perfect.
From my own observation, it made most sense to order 10 PCB´s each, because this way it was cheapest.
Usually there will be other HAM´s around, asking if anybody has a set of spare PCB, so you probably won´t end up keeping them.
Otherwise you can just ask in groups.io, if somebody wants to buy your spare PCB´s.
Please keep in mind, this is an open source project (other peoples development work, which you can use for free), and I´d appreciate if you just charge your expenses (Non Profit). Keep the HAM Spirit High !

**Where to get the Parts ?**

I´ve tried to make ordering of the Parts as easy as possible by providing complete „shopping baskets" at mouser.com
Mouser has almost every component, most of them in stock and ships virtually anywhere in the world.
Everything exept the toroids can be bought at mouser.

This is the complete Mouser „Shopping Cart" for uSDX **Mainboard** (including Designators):
**https://www.mouser.com/ProjectManager/ProjectDetail.aspx?AccessID=8690f61c05**

This is the complete Mouser „Shopping Cart" for uSDX **IO-Board** (including Designators):
**https://www.mouser.com/ProjectManager/ProjectDetail.aspx?AccessID=85157a5769**

This is the complete Mouser „Shopping Cart" for uSDX **Multi Band serial resonance RF Board** (including Designators):
**https://www.mouser.com/ProjectManager/ProjectDetail.aspx?AccessID=e1ba9350d9**

Those 3 shopping carts contain every part you need to built 1 complete DL2MAN Sandwich.
**EXEPT for the Toroids....**
**Additional to those carts you need 10x Micrometals or Amidon T37-2 (Red) and 1x FT37-43 (Black)**
Please do not order chinese Toroids from ebay or aliexpress, otherwise you will probably see problems with efficiency.
Like the capacitors on the RF Board, the use of High Quality Toroids (Original Micrometals or Amidon) is critical to success !
In America, they can be ordered from https://www.kitsandparts.com/
In Europe, they can be ordered from https://www.reichelt.de/

In case you have an alternative Source, or the Parts in Stock, it´s easy to remove parts after adding the cart to your order.
It´s also very easy to order each cart multiple times, if you make group buy or want to built more.

Tips:

- The Display is very expensive at mouser. You can find them cheaper at reichelt.de or on ebay sometimes
- The SMA Connector is also relatively expensive. I´ve used chinese Edge Mount SMA Connectors from ebay. The ones that are supplied with the Si5351 Breakout Boards
- You still need copper enamel wire (I use 0,4mm Diameter for all of my coils)
- The relays come and go, so any IM43 J and G Type will work. I´ve adjusted the Footprint to accept both. (J = SMT J-Leg, G = SMT Gull Wing)
- In worst case you can even use IM42 J or G. Coil code 42 is for 4,5 V Type latching and should still work.
- Gull Wings are easier to Hand solder. Keep that in mind, when you can chose.

Tools required:

- Good Soldering Iron with not too big Tip, but also no needle Tip. My preferred Tip is 2,6mm wide.
- Side cutter
- Stable Tweezers with very small tip
- Flux !
- Good eyes, or alternatively magnification Glass / Microsope
- calm hands ;)

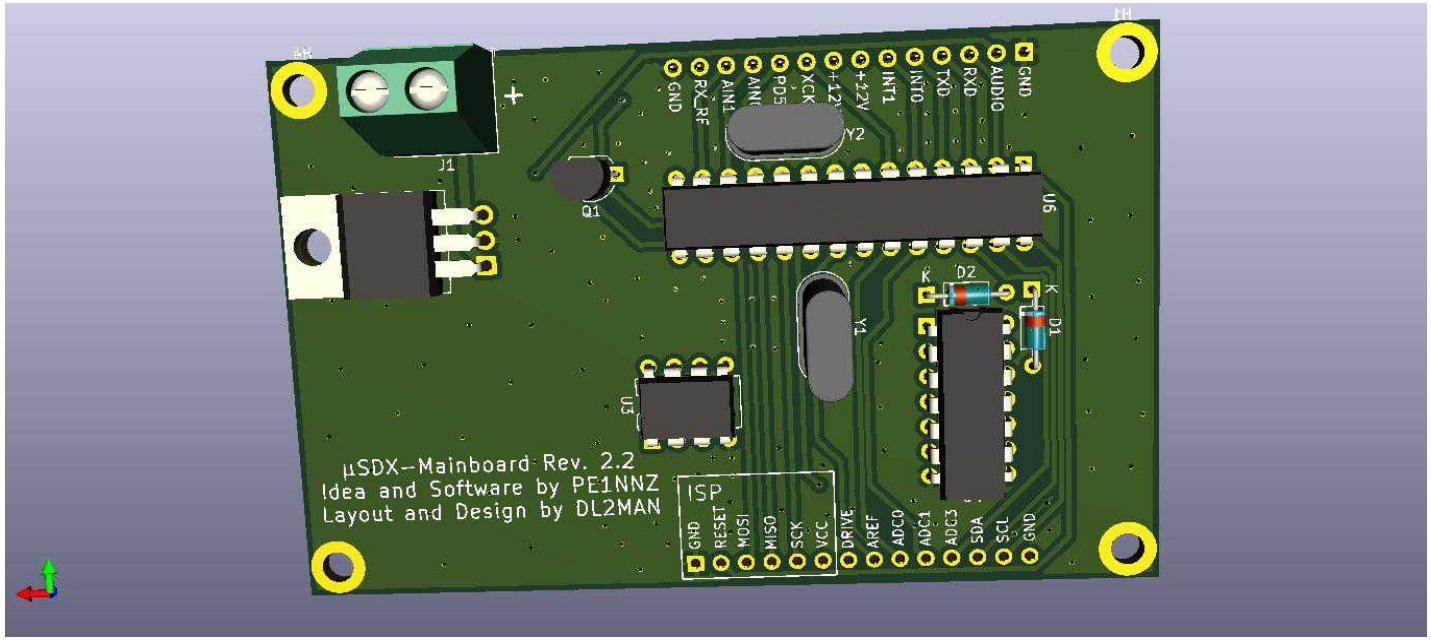
Optional (but not required):

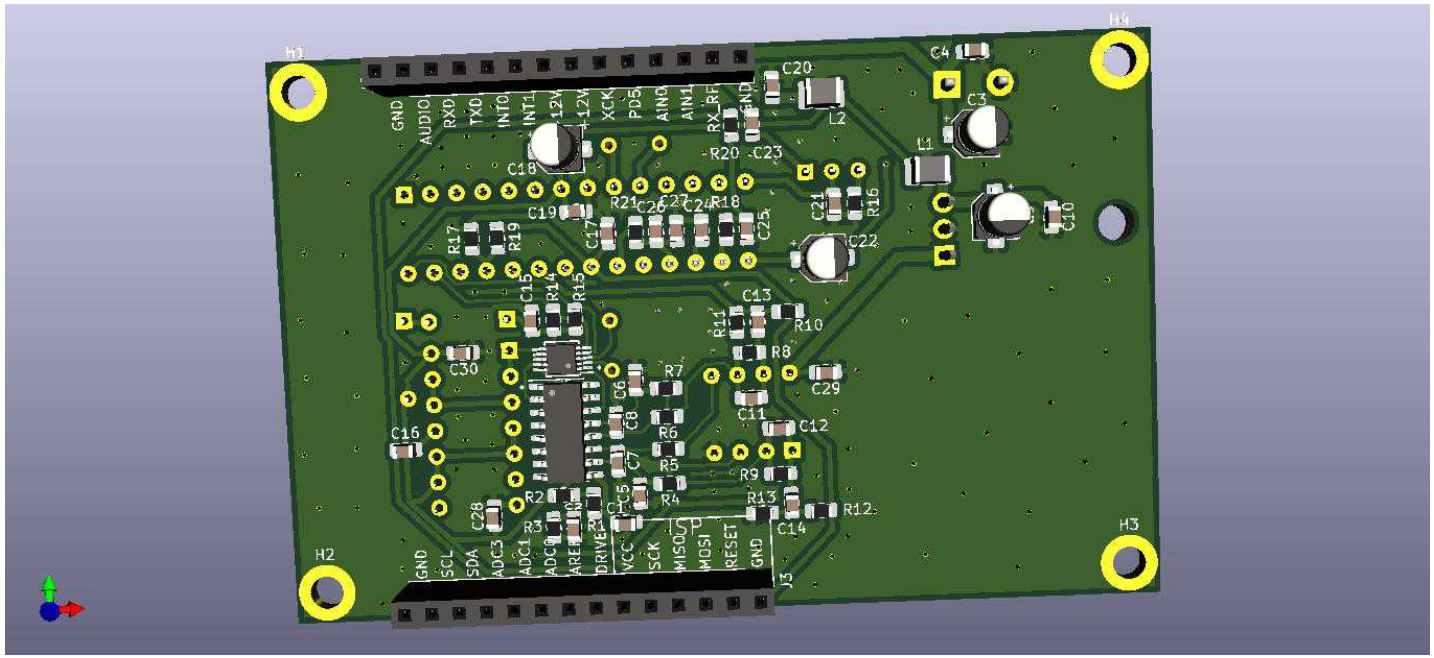
- Stencil for Mainboard and Multiband Board
- Soldering Paste and Reflow Oven

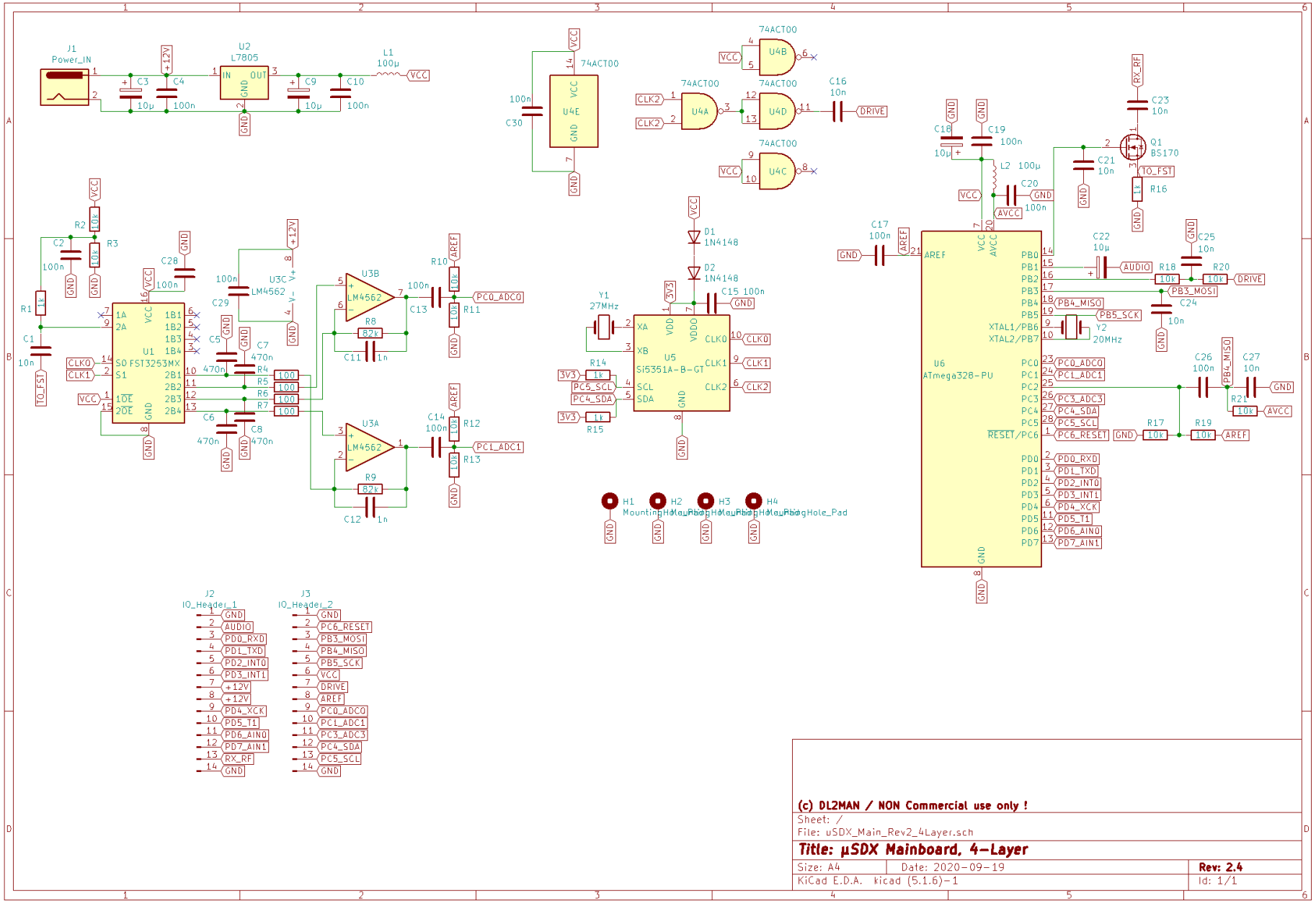
With every new built, I start with Main PCB, as it has the most parts on it.
Some building advice and Project Demonstration can be found in this Video: https://www.youtube.com/watch?v=qQI9Y6VXzl8&t=3s

Ordering a stencil (together with PCB) and using solder paste and reflow oven can accelerate built time significantly, but it is not neccessary. All the Parts can be soldered by Hand if you just apply the right technique.
An assembly tutorial with tips will be released soon.

Old Manual Text: (Will be reworked soon)
Apply solder paste with stencil, and place all Capacitors (without electrolytic Caps), resistors and Inductances. Place SMT IC´s.
Solder them in reflow Oven.
Hand Solder electrolytic Caps, Crystals and THT IC´s (including 7805),  I suggest use of IC Socket at Least for the AT Mega µ-Controller.
Hand Solder BF170 and Power Jack.
From the Stackable Connector Kit, use the 6Pin and 8Pin Connectors, to create 2x 14 Pin (IO Header 1 and 2).
Male Connectors should be on the side, where the THT IC´s are and where the Power Jack is.
Use a file, or Sand Paper, to remove a little bit of the Plastic, where the 6 and 8 Pin Connectors have contact to each other, so they will fit in the 14 Pin Header Holes without force.
Try to solder them in as straight as possible, and try to solder only close to PCB.  After placing the ATMega into Socket, you can Power it up and Program it with an Arduino UNO as ISP.




H1
H2
H3
H4
J1
J2
J3
GND
AUDIO
RXD
TXD
INT0
INT1
12V
12V
XCK
PD5
AIN0
AIN1
RX_RF
GND
SCL
SDA
ADC3
ADC2
ADC1
ADC0
AREF
DRIVE
VCC
SCK
MISO
MOSI
RESET
GND
C1
C2
C3
C4
C5
C6
C7
C8
C10
C11
C12
C13
C14
C15
C16
C17
C18
C19
C20
C21
C22
C23
C24
C25
C26
C27
C28
C29
C30
L1
L2
R1
R2
R3
R4
R5
R6
R7
R8
R9
R10
R11
R12
R13
R14
R15
R16
R17
R18
R19
R20
R21






Sheet: /
File: uSDX_Main_Rev2_4Layer.sch

**Title: µSDX Mainboard, 4-Layer**

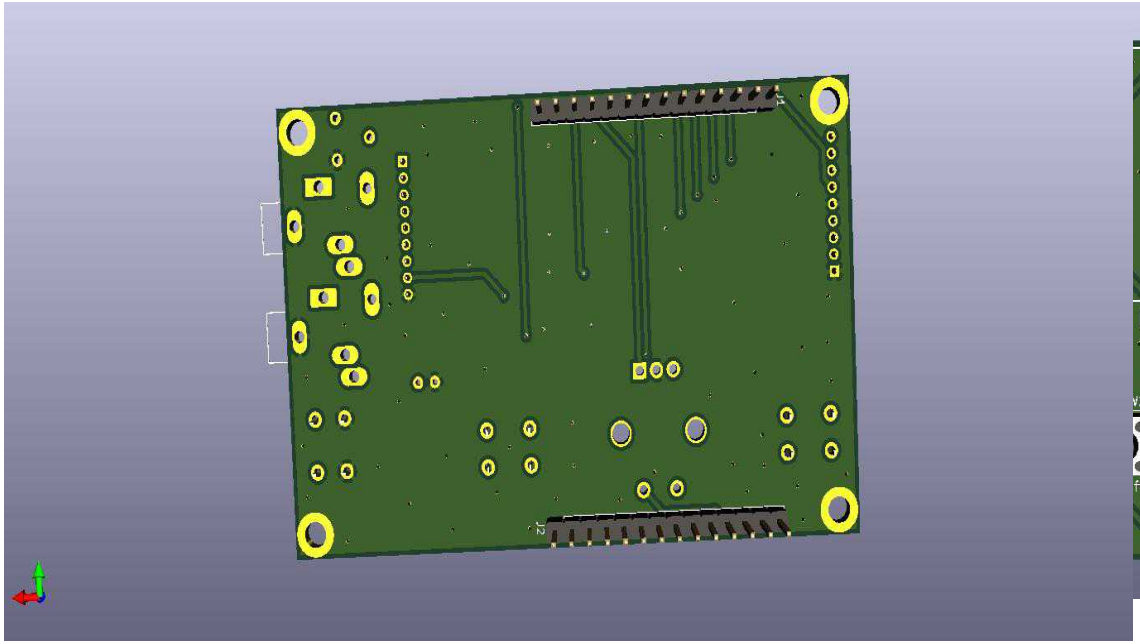
| Size: A4 | Date: 2020-09-19 | Rev: 2.4 |
|---|---|---|
| KiCad E.D.A. kicad (5.1.6)-1 | | Id: 1/1 |

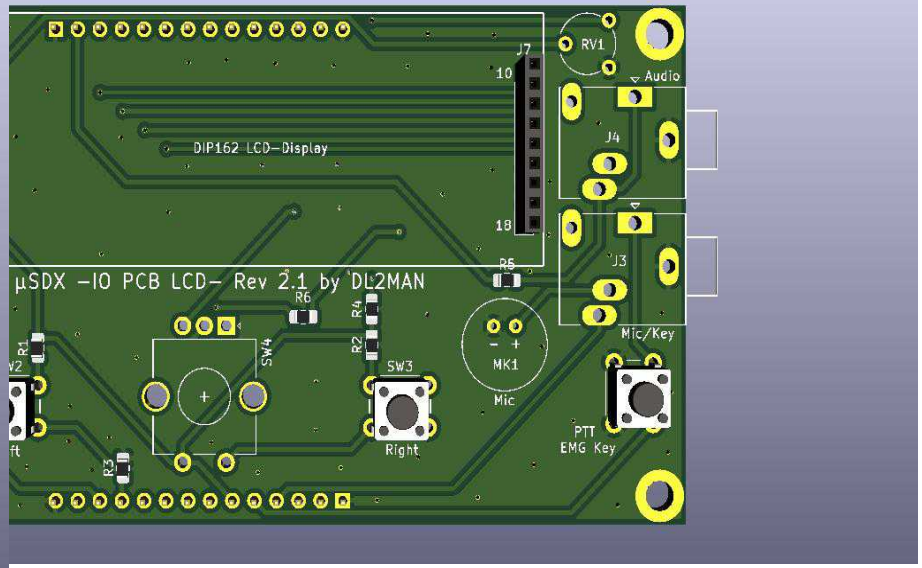
| Mainboard: Component Count: | 71 | | |
|---|---|---|---|
| Ref | Qnty | Value | Description |
| C1, C16, C21, C23, C24, C25, C27, | 7 | 10n | Capacitor SMT 0805 X7R |
| C2, C4, C10, C13, C14, C15, C17, C19, C20, C26, C28, C29, C30, | 13 | 100n | Capacitor SMT 0805 X7R |
| C3, C9, C18, C22, | 4 | 10µ | Capacitor electrolytic SMT |
| C5, C6, C7, C8, | 4 | 470n | Capacitor SMT 0805 X7R |
| C11, C12, | 2 | 1n | Capacitor SMT 0805 X7R |
| D1, D2, | 2 | 1N4148 | 1N4148 |
| H1, H2, H3, H4, | 4 | MountingHole_Pad | MountingHole_Pad |
| J1, | 1 | Power_IN | Terminal Block_Phoenix |
| J2, | 1 | IO_Header_1 | Cut from Adafruit Accessories 36pin |
| J3, | 1 | IO_Header_2 | Stacking Header 5 pack |
| L1, L2, | 2 | 100µ | Fixed Inductor SMT Size 1210 |
| Q1, | 1 | BS170 | BS170 |
| R1, R14, R15, R16, | 4 | 1k | Resistor SMT 0805 |
| R2, R3, R10, R11, R12, R13, R17, R18, R19, R20, R21, | 11 | 10k | Resistor SMT 0805 |
| R4, R5, R6, R7, | 4 | 100 | Resistor SMT 0805 |
| R8, R9, | 2 | 82k | Resistor SMT 0805 0,1% |
| U1, | 1 | FST3253MX | FST3253 TSSOP-16 Package |
| U2, | 1 | L7805 | 7805 in TO220 Package |
| U3, | 1 | LM4562 | LM4562 in DIP-8 Package |
| U4, | 1 | 74ACT00 | 74ACT00 ins DIP-14 Package |
| U5, | 1 | Si5351A-B-GT | Si5351A-B-GT in MSOP-10 Package |
| U6, | 1 | ATmega328-PU | Atmega328-PU in DIP-28 Package |
| Y1, | 1 | 27MHz | Crystal in HC49-US Package |
| Y2, | 1 | 20MHz | Crystal in HC49-US Package |

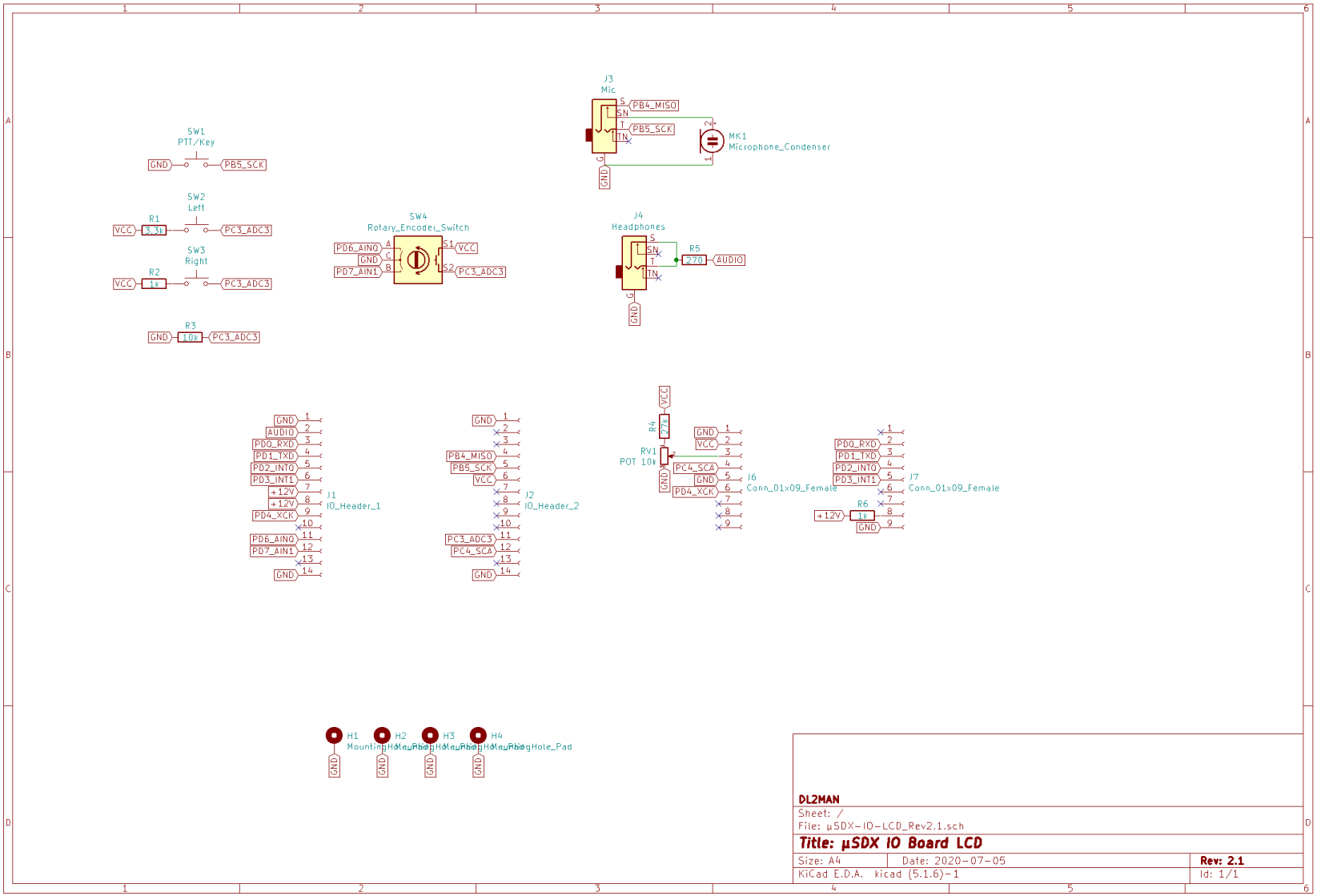
Continue with IO Board:
This built is actually pretty straight forward. Start with hand soldering the smaller parts (Resistors), Leave out R6 to save another 10mA of RX Current.
This resistor is for LCD Background illumination, but it´s not needed if you have green LCD. It can be read without.
Continue with Poti, Mic, Buttons.
Then you need to solder in J1 and J2 (You need to do this before soldering Display, otherwise solder joints cannot be reached anymore).
Then solder in Dispaly and rotary encoder.
The on-Board Mic (MK1) is a standard electret type capsule. It is beeing disconnected, when a plug is inserted in Key/Mic Jack.

Note: One of the 4 Spacers is below display and cannot be fit with M3 Screw. So this one is only a spacer. The other 3 also aid in mechanical stabilisation and GND Connection.
Note 2:You should bend the metal clamps of the frame on the backside of the LCD a little, to avoid contact with the IO Header.

| IO PCB | | | |
|---|---|---|---|
| SW4 | 1 | Rotary_Encoder_SW | RotaryEncoder_Alps_EC11E-Switch |
| H3,H4,H2,H1 | 4 | MountingHole_Pad | Spacer 10mm, M3 Threading Male/Female |
| MK1 | 1 | EMY-9765P | Condenser Microphone |
| J1 | 1 | IO_Header_1 | Use „breakable“ male Pin headers. |
| J2 | 1 | IO_Header_2 | Use „breakable“ male Pin headers. |
| R1 | 1 | 3,3k | Resistor, SMT, 0805 |
| R2,R6 | 2 | 1k | Resistor, SMT, 0805 (Leave Out R6 for no Background illumination) |
| R3 | 1 | 10k | Resistor, SMT, 0805 |
| R5 | 1 | 270 | Resistor, SMT, 0805 |
| SW2 | 1 | Left | Print Push Button_6mm_H5mm |
| SW3 | 1 | Right | Print Push Button_6mm_H5mm |
| J3 | 1 | Mic | Jack_3.5mm_with 2x internal Switch (eg LUM 1503-09) |
| J4 | 1 | Headphones | Jack_3.5mm_with 2x internal Switch (eg LUM 1503-09) |
| SW1 | 1 | PTT/Key | Print Push Button_6mm_H5mm |
| J6,J7 | 1 | LCD 162 DIP | LCD Display „162 DIP“ Green |
| R4 | 1 | 27k | Resistor, SMT, 0805 |
| RV1 | 1 | POT 10k | Potentiometer_Piher_PT-6-V_Vertical |



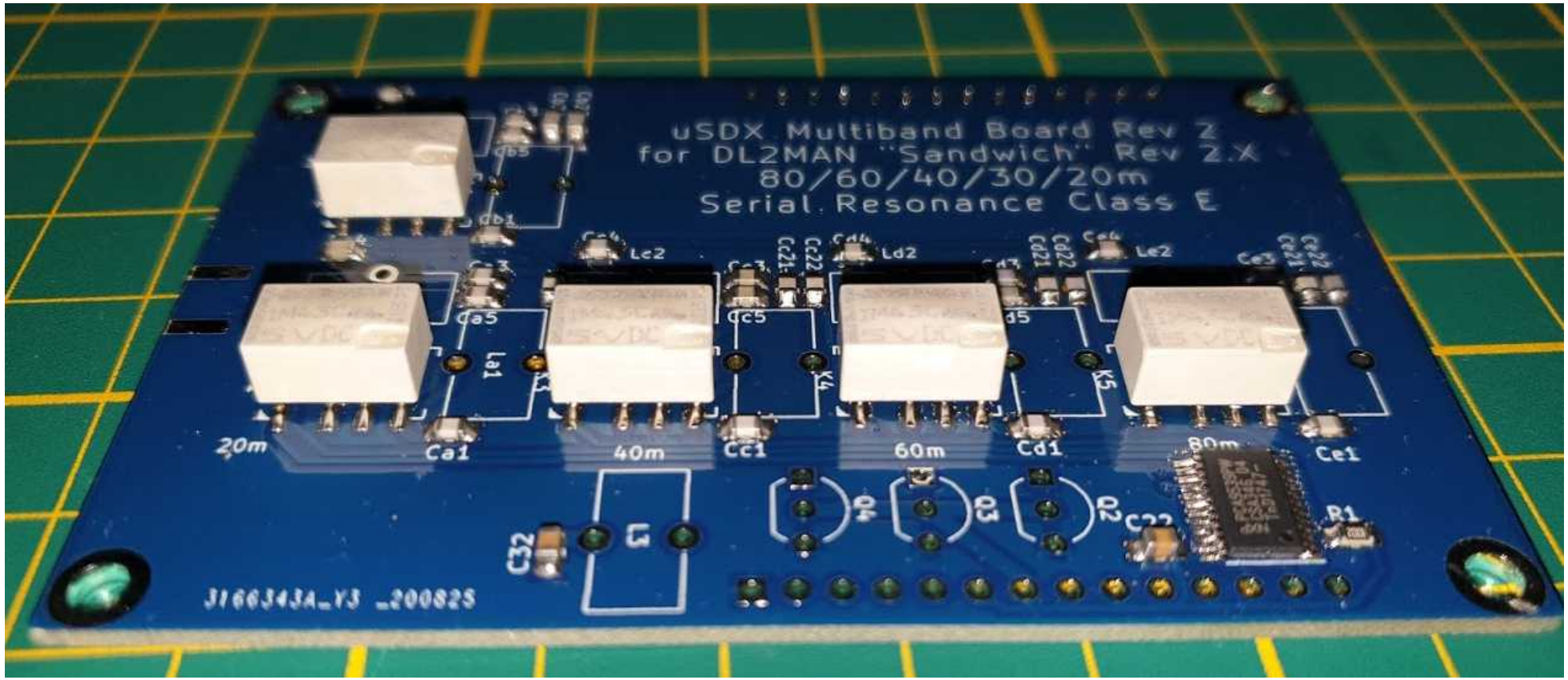





| | | |
|---|---|---|
| DL2MAN | | |
| Sheet: / | | |
| File: µSDX-IO-LCD_Rev2.1.sch | | |
| **Title: µSDX IO Board LCD** | | |
| Size: A4 | Date: 2020-07-05 | **Rev: 2.1** |
| KiCad E.D.A. kicad (5.1.6)-1 | | Id: 1/1 |

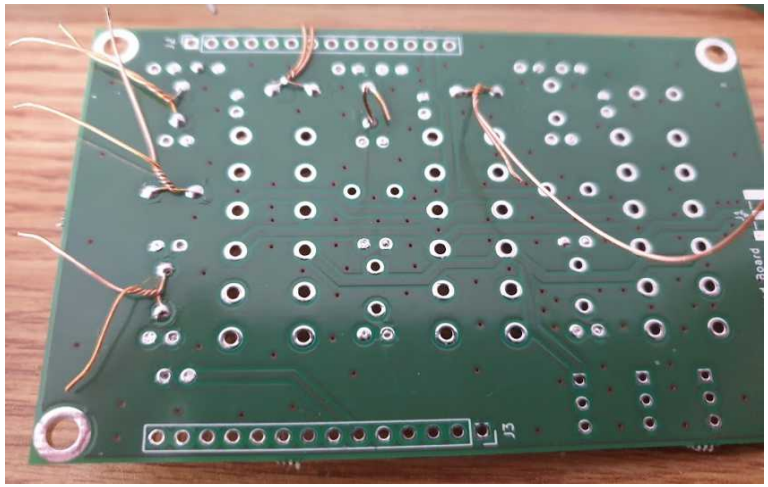
And finally we take care about RF PCB – **NEW serial resonance Multi-Band PCB !**

First place the Capacitors, the Resistor and the GPIO Expander IC on the Board (U1) according to the Schematic and BOM Table below.
The capacitors on this board need to be 0805 NP0/C0G Type, rated for 100V !  Leave out Ca1 for best Efficiency !
Then continue with the Relays:



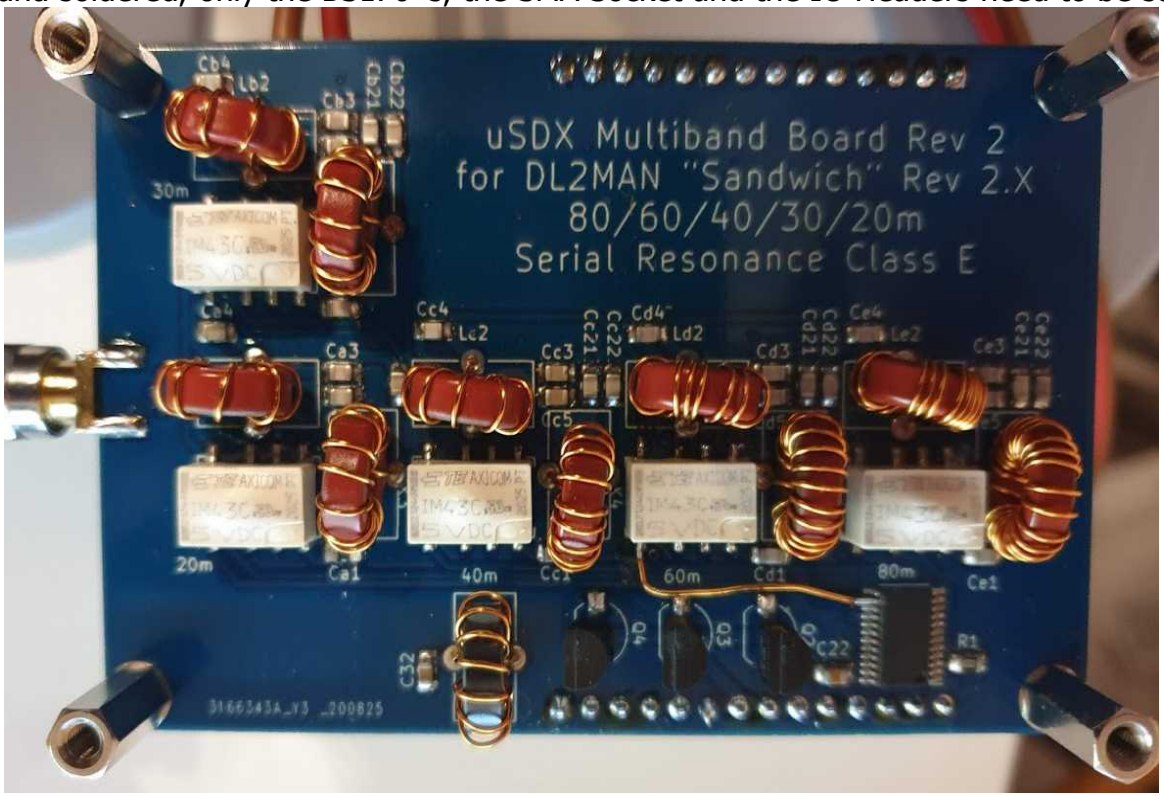
Then select Torids according to Schematic and wind them with the amount of turns according to Schematic. For Example: 10T/T37-2 means: Use Toroid Core T37-2 (= red one), put on 10 Turns. The given Values were tested with at least 3 Builts and produced always between 4-6W Output (@13,8V) and Efficiency over 80%.  The footprint on the PCB almost forces you to devide the windings evenly and I found this gives me much more repeatable results. Remember: When you insert the Copper enamel wire in the core, this is already winding Nr. 1 ! Wind one Toroid Coil after another, scratch off the enamel coating of their ends with a knife, stick the lose ends through PCB and twist them like shown below. Then they won´t fall out, and you can solder them after having placed all of them (or a few of them like in my Photo)

This Photo was taken from the first Multiband Version, but the technique is stll the same.



All my coils were wound with 0,4mm Diameter copper enamel wire.

After all Toroids are Placed and soldered, only the BS170´s, the SMA Socket and the IO Headers need to be soldered on:




The Software File in „Firmware" Folder is optimized for use with Relay Switching.

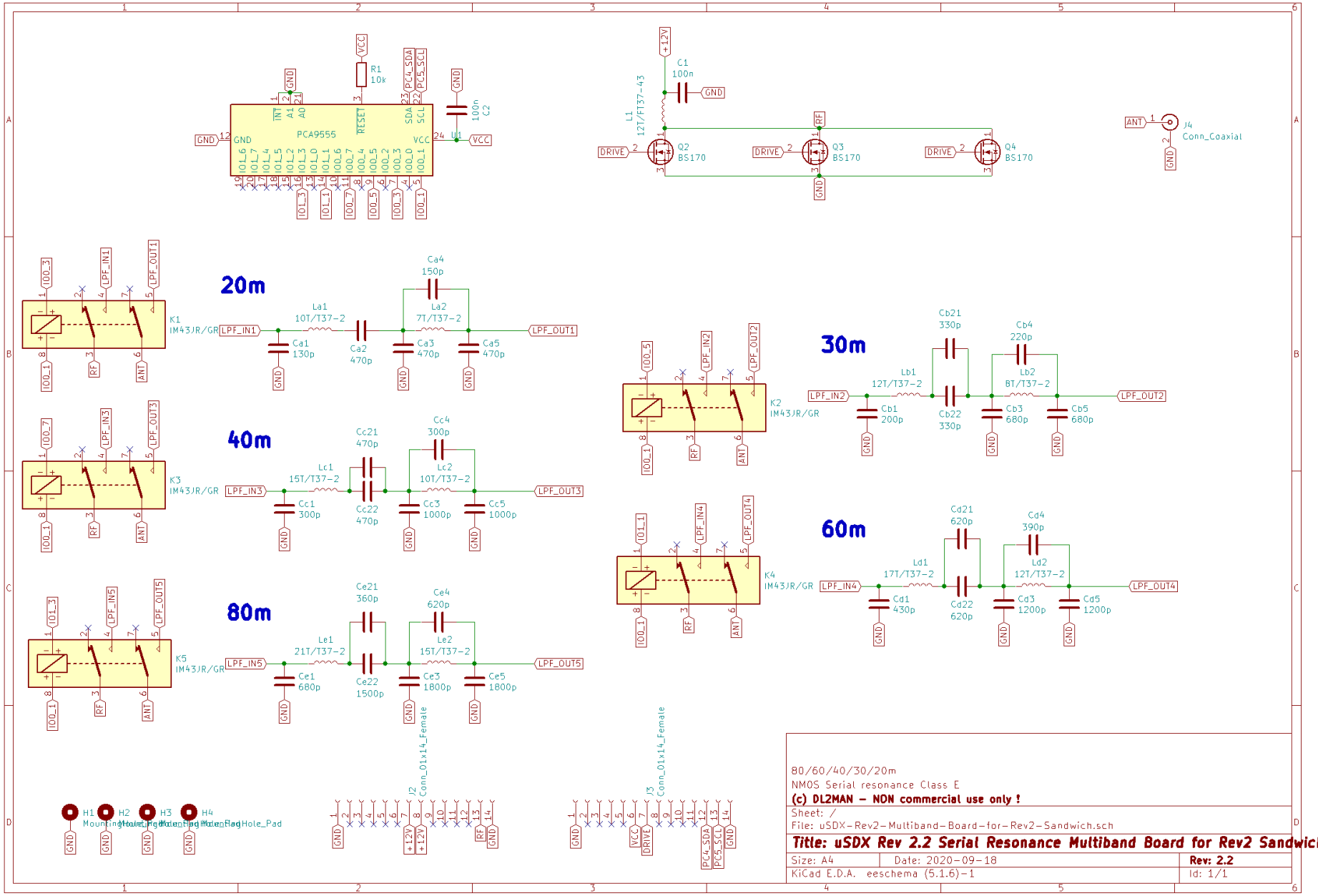
When testing  RF Boards, I always measure Input Current and Output Power into Dummy Load.
You can calculate efficieny:
Power In = Voltage in x (Current In while TX – Current while RX)
Effieciency in % = Power Out / Power In x 100

With this built and the above Values in Schematic I get beween 82-93% of Efficiency.

You can affect efficiency and output Power by Squeezing windings more together or seperate them more evenly. Usually no more tweaking then that is required.

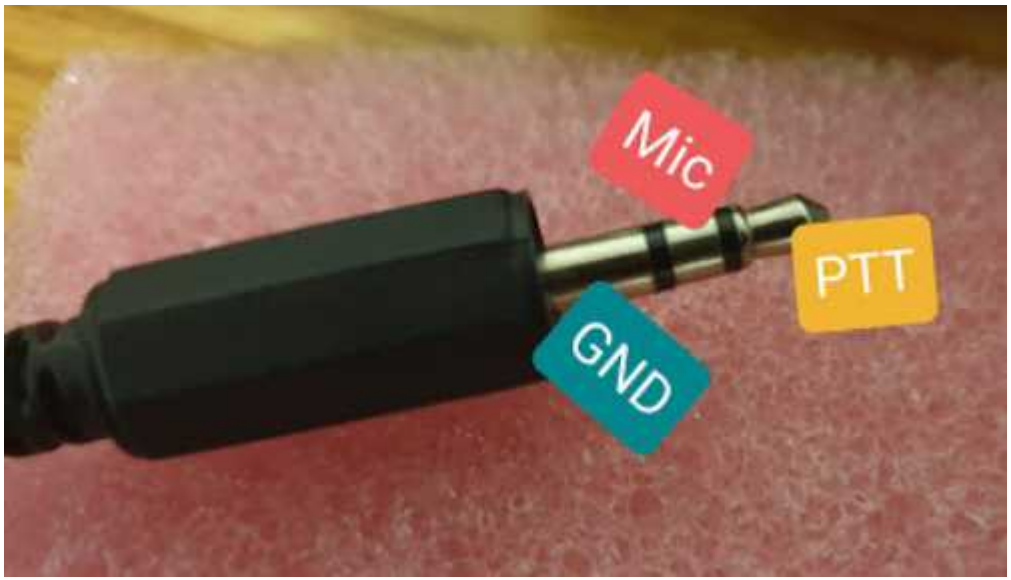
Multiband RF Board BOM:

| Component Count: | 59 | | |
|---|---|---|---|
| Ref | Qnty | Value | Footprint |
| C1, C2, | 2 | 100n | Capacitor_SMD, Size 0805, X7R Type |
| Ca1, | 1 | (130p) leave out ! | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Ca2, Ca3, Ca5, Cc21, Cc22, | 5 | 470p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Ca4, | 1 | 150p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cb1, | 1 | 200p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cb3, Cb5, Ce1, | 3 | 680p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cb4, | 1 | 220p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cb21, Cb22, | 2 | 330p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cc1, Cc4, | 2 | 300p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cc3, Cc5, | 2 | 1000p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cd1, | 1 | 430p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cd3, Cd5, | 2 | 1200p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cd4, | 1 | 390p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Cd21, Cd22, Ce4, | 3 | 620p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Ce3, Ce5, | 2 | 1800p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Ce21, | 1 | 360p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| Ce22, | 1 | 1500p | Capacitor_SMD: Size 0805, NP0/C0G Type, rated for 100V |
| H1, H2, H3, H4, | 4 | MountingHole_Pad | MountingHole:MountingHole_3.2mm_M3_DIN965_Pad |
| J2, J3, | 2 | Conn_01x14_Female | Connector_PinSocket_2.54mm:PinSocket_1x14_P2.54mm_Vertical |
| J4, | 1 | Conn_Coaxial | Connector_Coaxial:SMA_Samtec_SMA-J-P-X-ST-EM1_EdgeMount |
| K1, K2, K3, K4, K5, | 5 | IM43JR/GR | Relais:Relay_DPDT_AXICOM_IMSeries_GullWing |
| L1, | 1 | 12T/FT37-43 | 12 Turns on FT37-43 Core (Black) |
| La1, Lc2, | 2 | 10T/T37-2 | 10 Turns on T37-2 Core (red) |
| La2, | 1 | 7T/T37-2 | 7 Turns on T37-2 Core (red) |
| Lb1, Ld2, | 2 | 12T/T37-2 | 12 Turns on T37-2 Core (red) |
| Lb2, | 1 | 8T/T37-2 | 8 Turns on T37-2 Core (red) |
| Lc1, Le2, | 2 | 15T/T37-2 | 15 Turns on T37-2 Core (red) |
| Ld1, | 1 | 17T/T37-2 | 17 Turns on T37-2 Core (red) |
| Le1, | 1 | 21T/T37-2 | 21 Turns on T37-2 Core (red) |
| Q2, Q3, Q4, | 3 | BS170 | Package_TO_SOT_THT:TO-92_Inline_Wide |
| R1, | 1 | 10k | Resistor_SMD, Size 0805 |
| U1, | 1 | PCA9555 | Package_SO:TSSOP-24_4.4x7.8mm_P0.65mm |


PCA9555
R1
10k
C2
100n
U1
+12V
C1
100n
L1
12T/FT37-43
Q2
BS170
Q3
BS170
Q4
BS170
J4
Conn_Coaxial
20m
K1
IM43JR/GR
La1
10T/T37-2
La2
7T/T37-2
Ca1
130p
Ca2
470p
Ca3
470p
Ca4
150p
Ca5
470p
30m
K2
IM43JR/GR
Lb1
12T/T37-2
Lb2
8T/T37-2
Cb1
200p
Cb21
330p
Cb22
330p
Cb3
680p
Cb4
220p
Cb5
680p
40m
K3
IM43JR/GR
Lc1
15T/T37-2
Lc2
10T/T37-2
Cc1
300p
Cc21
470p
Cc22
470p
Cc3
1000p
Cc4
300p
Cc5
1000p
60m
K4
IM43JR/GR
Ld1
17T/T37-2
Ld2
12T/T37-2
Cd1
430p
Cd21
620p
Cd22
620p
Cd3
1200p
Cd4
390p
Cd5
1200p
80m
K5
IM43JR/GR
Le1
21T/T37-2
Le2
15T/T37-2
Ce1
680p
Ce21
360p
Ce22
1500p
Ce3
1800p
Ce4
620p
Ce5
1800p
J2
Conn_01x14_Female
J3
Conn_01x14_Female
H1
H2
H3
H4
MountingHole_Pad
80/60/40/30/20m
NMOS Serial resonance Class E
(c) DL2MAN – NON commercial use only !
Sheet: /
File: uSDX-Rev2-Multiband-Board-for-Rev2-Sandwich.sch
Title: uSDX Rev 2.2 Serial Resonance Multiband Board for Rev2 Sandwich
Size: A4
Date: 2020-09-18
Rev: 2.2
KiCad E.D.A. eeschema (5.1.6)-1
Id: 1/1

Final important notes:

– When connecting external microphone, mic wiring needs to be as following:




I have changed this in my Layout, so the wiring is compatible to the „CW straight key" standard.

– If you already have another mic, it is simple to make a short adapter cable, which switches wires for the sandwich.

– when programming via ISP Interface, you should to insert a 3,5mm Stereo Plug (with nothing connected) into Mic/PTT Jack. Otherwise the internal Diode of the On-Board Microphone **can** mess with the MISO Line.

– Be extremely careful, when putting the Sandwich together. It´s pretty easy to miss one pin, or even worse: Connect it to the wrong pin.

– Always remove it from Power Supply when Disassembling / Reassembling the boards.

– If you see problems with bad Sideband rejection: There´s 2 Solutions: **Chose one (not both !)**

– **1. Hardware:** Add 50 Ohms Resistor in Series with C1. This can be done by placing C1 on one Pad standing, and adding 2 parallel 100Ohms resistors standing on the other pad, then connect the top Connectors of the resistors and C1 with a solder blop . This was found to be helpful on 4 Layer Mainboard in some cases.

– **2. Software:** In recent Software (m-Version from RX_Improved Branch), there´s a Menu Setting (RX Phase Adj) that allows you to fix any misalignment. The most precise Way to do this, is hooking up an oscilloscope to ADC0 and ADC1 and adjust that value until both phases (I and Q) are equal in Amplitute, and 90° apart (while inserting a signal into the Receiver !!!). However, if you do not have an oscilloscope, in most cases, the best Value found was around 100.

Always adjust Phase on the HIGHEST Band of your Buillt.

Additional Info:

If you want to built it for other Bands, then I did, here are the theoretical Values to do so. However: Efficiency decreases with increasing frequency !
Always put your Highest Band closest to the Antenna Jack.
Keep the Order from Highest to lowest Band according to the existing scheme.
In software (attached to this building manual), you then gotta change the „switching points" of the relays, according to the bands you ´ve placed. (Line 1479):

```
uint8_t lpf_io = (f > 12) ? 3 : (f > 8) ? 5 : (f > 6) ? 7 : (f > 4) ? 9 : /*(f > 2)*/ 11; // cut-off freq in MHz to IO port of LPF relay
```

| 3x BS170 (theoretical) | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| Band | 80m | 60m | 40m | 30m | 20m | 17m | 15m | 12m | 10m | 6m |
| F (MHz) | 3,56 | 5,35 | 7,05 | 10,1 | 14,05 | 18,08 | 21,05 | 24,9 | 28,05 | 50,09 |
| Load (Ohm) | 12 | 12 | 12 | 12 | 12 | 12 | 12 | 12 | 12 | 12 |
| C1 (pF) | 657 | 420 | 307 | 199 | 128 | 88 | 69 | 50 | 39 | 0 (-1) |
| C2 (pF) | 1864 | 1240 | 941 | 657 | 472 | 367 | 315 | 266 | 237 | 132 |
| L2 (µH) | 2,04 | 1,36 | 1,03 | 0,72 | 0,52 | 0,4 | 0,35 | 0,29 | 0,26 | 0,15 |
| C3 (pF) | 1826 | 1215 | 922 | 644 | 463 | 360 | 309 | 261 | 232 | 130 |
| C5 (pF) | 1826 | 1215 | 922 | 644 | 463 | 360 | 309 | 261 | 232 | 130 |
| L3 (µH) | 0,82 | 0,55 | 0,41 | 0,29 | 0,21 | 0,16 | 0,14 | 0,12 | 0,1 | 0,06 |
| C4 (pF) | 609 | 405 | 307 | 215 | 154 | 120 | 103 | 87 | 77 | 43 |

The Designators in this Table does not match the designators on PCB, as you will change them. Look at the following schmatic and compare which one goes where.

L1 is in any case 50µH – around 12 Turns on FT37-43.

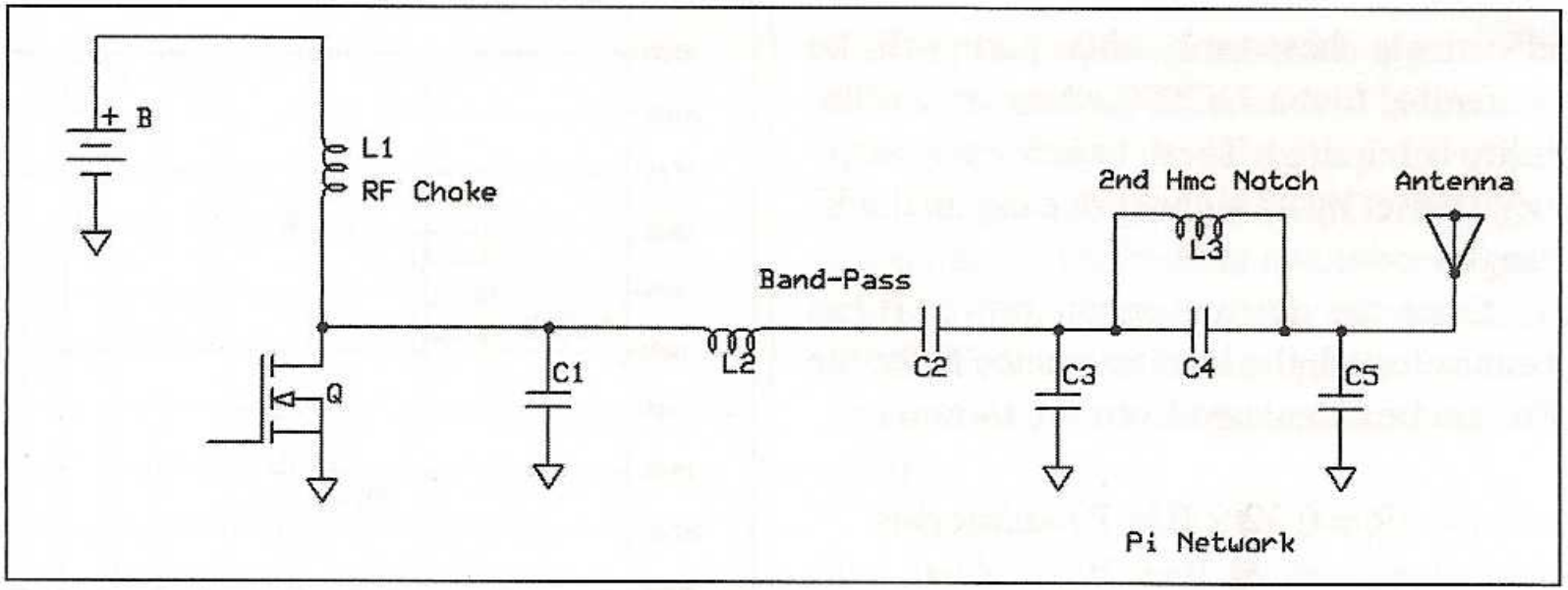



**Figure 3—Practical Class-E power amplifier.**

Have fun while building, and please report any bugs and errors while building to groups.io
or directly to me:
DL2MAN@gmx.de