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 RF PCB. Now even as Multiband Rig with switchable Low-Pass Filters.

Software to make the Microcontroller work can be found 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 Design is devided into 3 PCB´s in Size of a Credit Card (85x55mm) so it´s easier to change things in future.
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 3 switchable LPF´s
4. (Optional: RF PCB Monoband without Relay switching)

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 PCB´s are mechanically stabelized by Spacers in M3 Threading. If you use metal spacers, they will provide additional GND Connection between PCB´s and are very handy to connect for example Oszilloscope GND for measurement.

For Construction, I start with Main PCB, as it is the most complicated one.
Some building advice and Project Demonstration can be found in this Video: https://www.youtube.com/watch?v=qQI9Y6VXzl8&t=3s

At least for Main PCB, I would suggest to order stencil and use it with Reflow Oven. It can be Hand soldered, but at least with Small Si5351 IC, you will have a very hard time ;)
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.

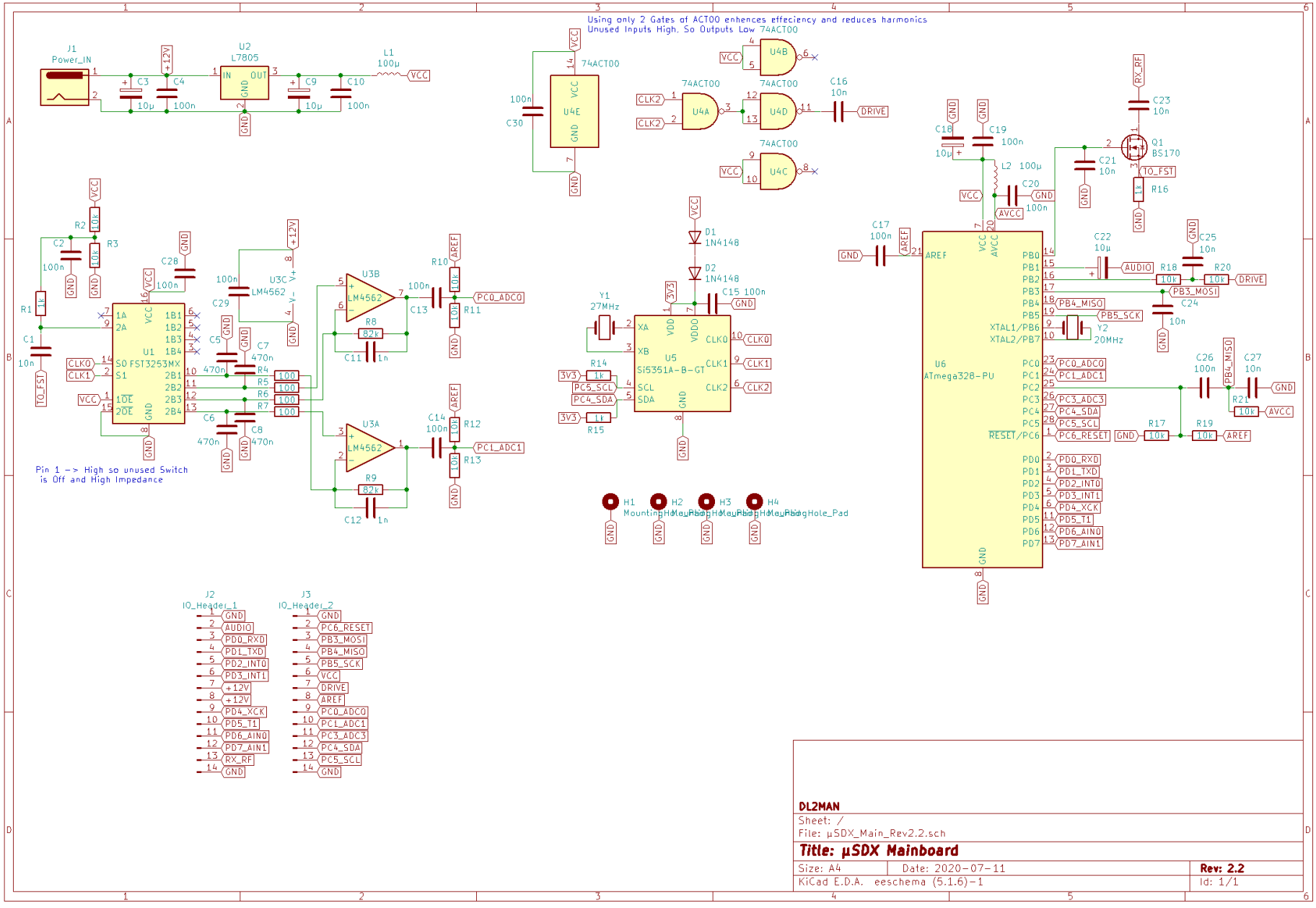
From the Stackable Connector Kit, use the 6Pin and 8Pin Connectors, to create 2x 14 Pin (IO Header 1 and 2).

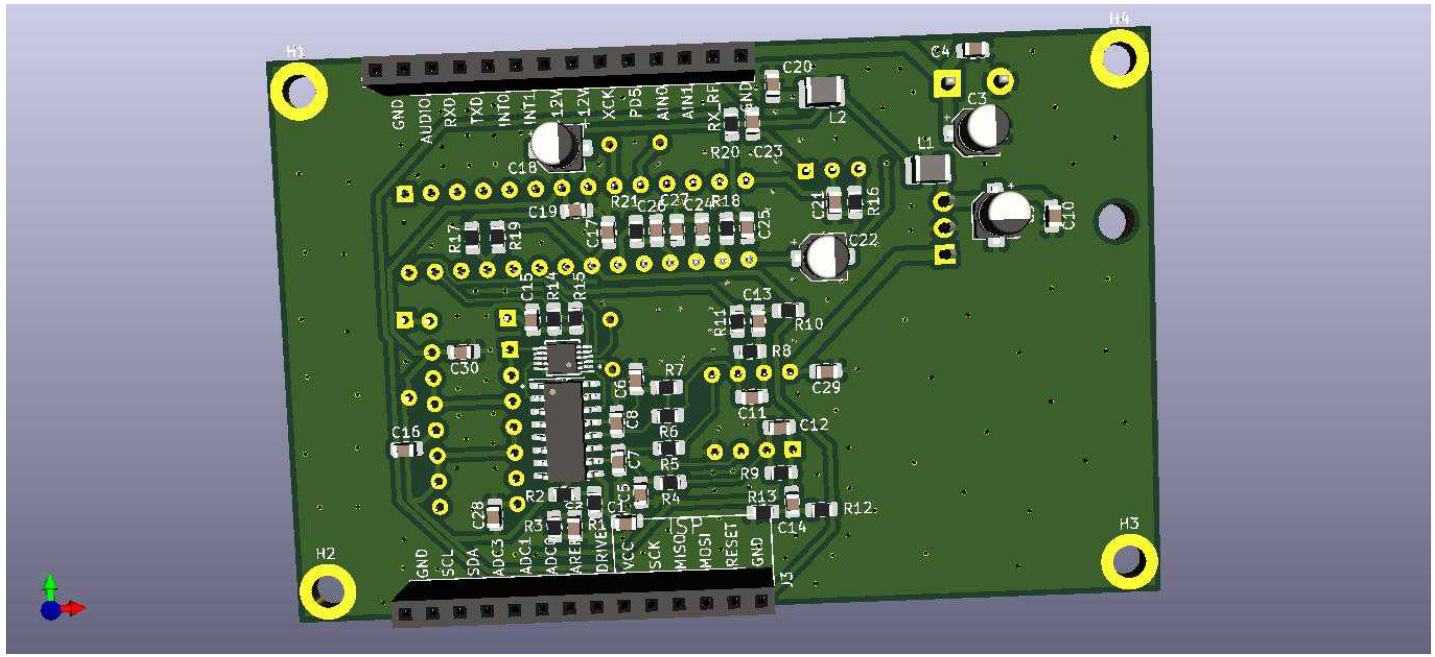
Male Connectors should be on the side, where the THT IC´s are and where the Power Jack is.

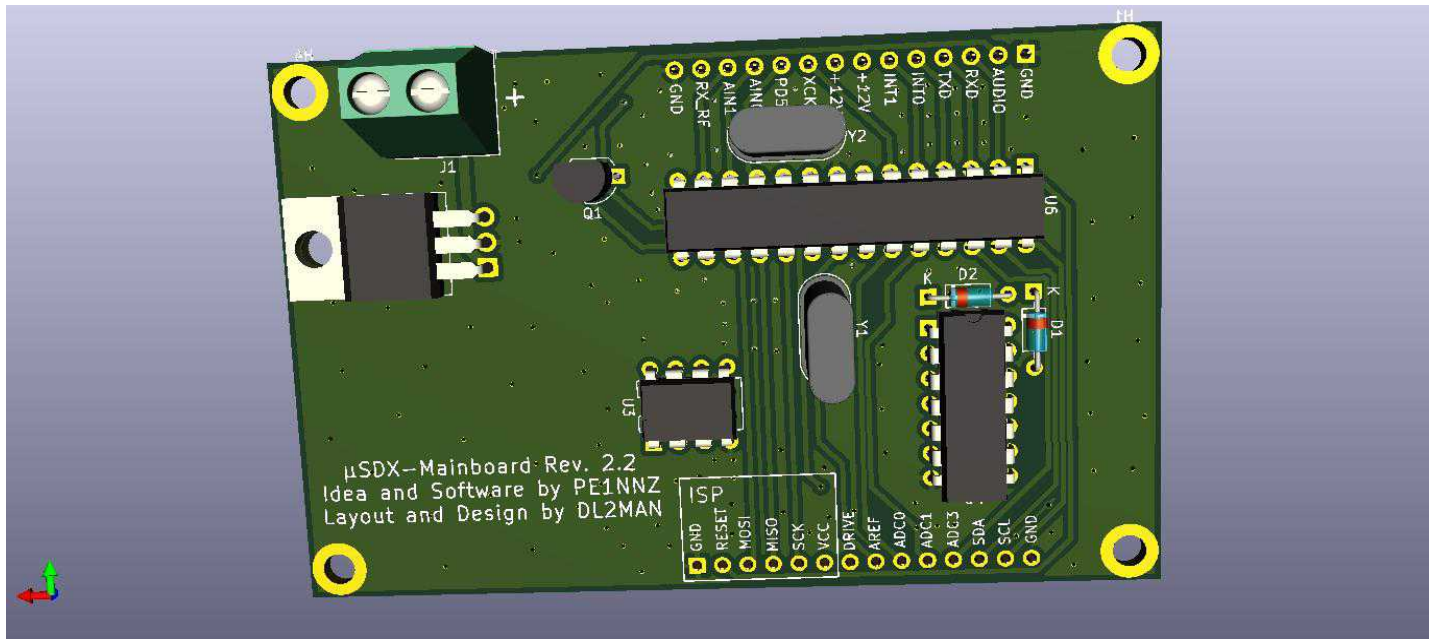
Use a file, or Sand Paper, to remove a little bit of the Plastic, where the 6 and 8 Pin Connectors have contact to each other, so they will fit in the 14 Pin Header Holes without force.

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.

Mainboard			
Part	Amount	Value	Description
C1,C16,C21,C25,C27,C24,C23	7	10n	Capacitor, SMT, 0805
C2,C4,C10,C13,C14,C15,C17,C19,C20,C26,C28,C29,C30	13	100n	Capacitor, SMT, 0805
C5,C6,C7,C8	4	470n	Capacitor, SMT, 0805
C11,C12	2	1n	Capacitor, SMT, 0805
D1,D2	2	1N4148	Diode 1N4148 THT
L1,L2	2	100µ	Fixed Inductance 100µH, Size 1610
R13,R2,R3,R10,R11,R12,R18,R17,R20,R19,R21	11	10k	Resistor, SMT, 0805
R1,R14,R15,R16	4	1k	Resistor, SMT, 0805
R4,R5,R6,R7	4	100	Resistor, SMT, 0805
R8,R9	2	82k	Resistor, SMT, 0805 0,1%
U1	1	FST3253MX	SOIC-16 Package
U4	1	74ACT00	DIP-14 Package
U5	1	Si5351A-B-GT	MSOP-10 Package
U6	1	Atmega328P-PU	DIP-28 Package, Use Socket !!!
Y1	1	27MHz	Crystal, THT, Package HC49U-S
Y2	1	20MHz	Crystal, THT, Package HC49U-S
U3	1	LM4562	DIP-8 Package
U2	1	L7805	TO-220 Package
J2	1	IO_Header_1	PinSocket_1x14, use Adafruit Stackable Pinheader MPN #85
J3	1	IO_Header_2	PinSocket_1x14, use Adafruit Stackable Pinheader MPN #85
J1	1	Power_IN	TerminalBlock_Phoenix_MKDS-1,5-2-5.08_1x02_P5.08mm_Horizontal
C3,C9,C18,C22	4	10µ	Electrolytic Capacitor 10µH 35V
Q1	1	BS170	TO-92 Housing
H1,H2,H3,H4	4	MountingHole_Pad	Spacer 10mm, M3 Threading, conductive


Using only 2 Gates of ACT00 enhences effeciency and reduces harmonics
Unused Inputs High, So Outputs Low 74ACT00
Pin 1 -> High so unused Switch is Off and High Impedance
DL2MAN
Sheet: /
File: µSDX_Main_Rev2.2.sch
Title: µSDX Mainboard
Size: A4
Date: 2020-07-11
Rev: 2.2
KiCad E.D.A. eeschema (5.1.6)-1
Id: 1/1

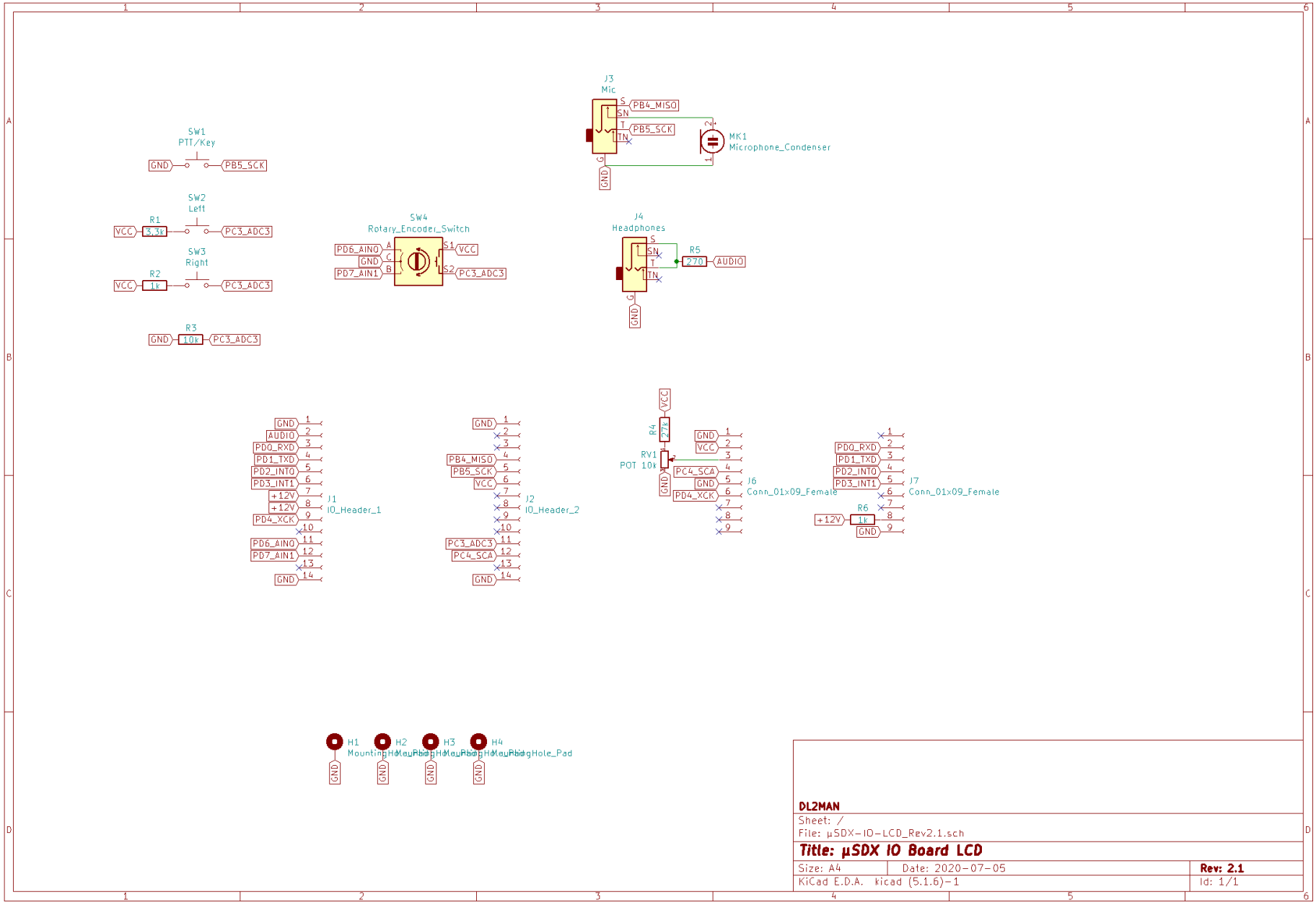

H1
H4
H2
H3
J3
GND
AUDIO
RXD
TXD
INT0
INT1
+12V
XCK
PD5
AIN0
AIN1
RX_RF
GND
C20
L2
C4
C3
L1
C10
C18
C19
R20
C23
R21
C26
C27
C24
R18
C25
C21
R16
C22
R17
R19
C17
C15
R14
R15
C13
R11
R10
R8
C30
C6
R7
C29
C8
C11
C12
C16
R6
C7
R5
R9
R2
C5
R4
R13
R12
C28
R3
C14
ADC3
ADC1
ADC0
AREF
DRIVE
VCC
SCK
MISO
MOSI
RESET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

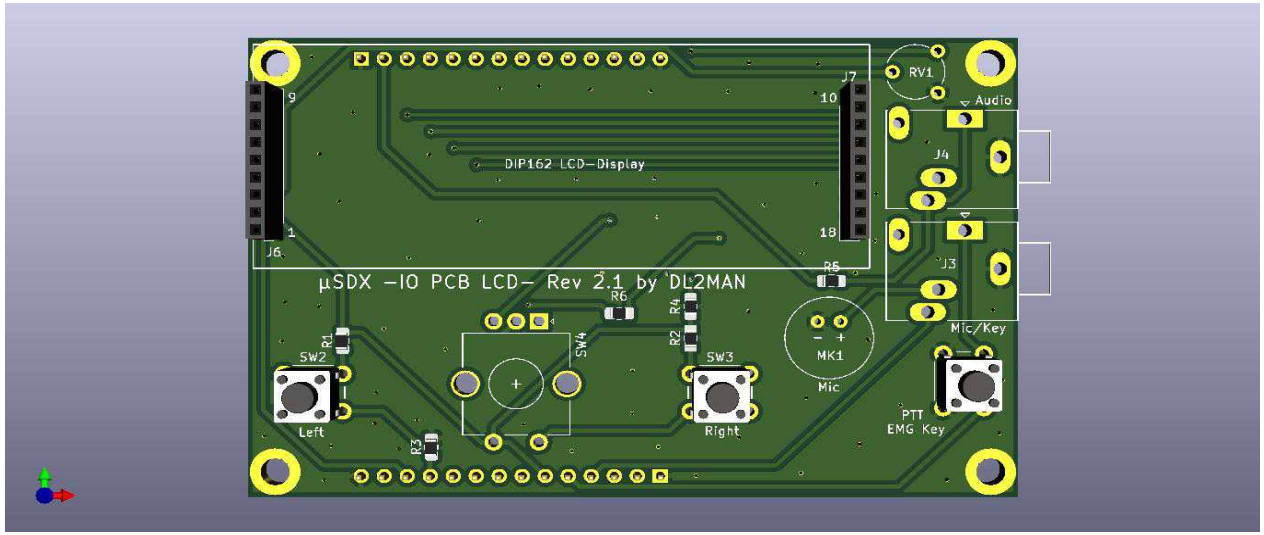

H1
H4
J2
J1
Q1
GND
RXD_RF
AIN1
AIN0
PD5
XCK
+12V
INT1
INT0
TXD
RXD
AUDIO
Y2
U6
D2
K
D1
Y1
U3
µSDX–Mainboard Rev. 2.2
Idea and Software by PE1NNZ
Layout and Design by DL2MAN
ISP
GND
RESET
MOSI
MISO
SCK
VCC
DRIVE
AREF
ADC0
ADC1
ADC3
SDA
SCL
GND

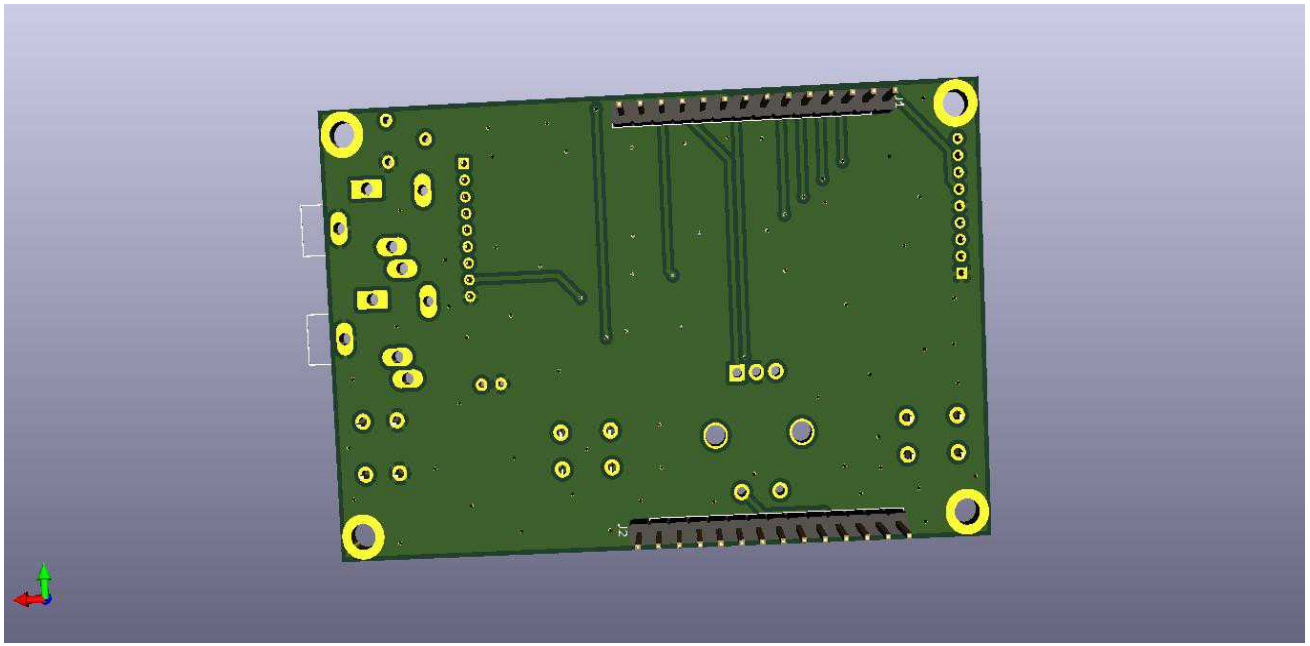
Continue with IO Board:
This built is actually pretty straight forward. Start with hand soldering the smaller parts (Resistors), Leave out R6 to save another 10mA of RX Current. This resistor is for LCD Background illumination, but it´s not needed if you have green LCD. It can be read without.
Continue 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.
You should bend the metal clamps of the frame on the backside of the LCD a little, to avoid contact with the IO Header. If you put it in to try fit, you will se, which ones to bend.
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.

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

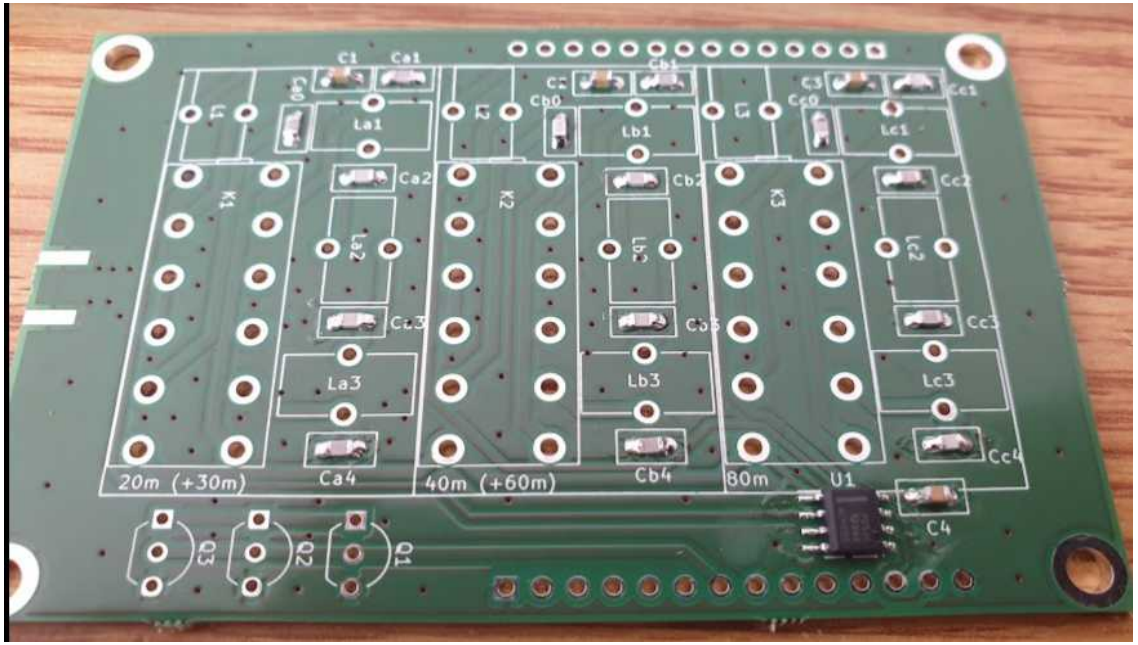

SW1
PTT/Key
GND
PB5_SCK
SW2
Left
R1
VCC
3.3k
PC3_ADC3
SW3
Right
R2
VCC
1k
PC3_ADC3
R3
GND
10k
PC3_ADC3
SW4
Rotary_Encoder_Switch
PD6_AIN0
GND
PD7_AIN1
A
C
B
S1
S2
VCC
PC3_ADC3
J3
Mic
S
SN
T
TN
G
PB4_MISO
PB5_SCK
GND
MK1
Microphone_Condenser
J4
Headphones
S
SN
T
TN
G
R5
270
AUDIO
GND
J1
IO_Header_1
GND
AUDIO
PD0_RXD
PD1_TXD
PD2_INT0
PD3_INT1
+12V
+12V
PD4_XCK
PD6_AIN0
PD7_AIN1
GND
J2
IO_Header_2
GND
PB4_MISO
PB5_SCK
VCC
PC3_ADC3
PC4_SCA
GND
R4
27k
VCC
RV1
POT 10k
GND
J6
Conn_01x09_Female
GND
VCC
PC4_SCA
GND
PD4_XCK
J7
Conn_01x09_Female
PD0_RXD
PD1_TXD
PD2_INT0
PD3_INT1
R6
+12V
1k
GND
H1
H2
H3
H4
MountingHole_Pad
GND
GND
GND
GND
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


DIP162 LCD-Display
µSDX -IO PCB LCD- Rev 2.1 by DL2MAN
J7
J6
RV1
Audio
J4
J3
Mic/Key
R5
R6
R4
R2
R1
SW2
SW4
SW3
MK1
Mic
Left
Right
R3
PTT
EMG Key

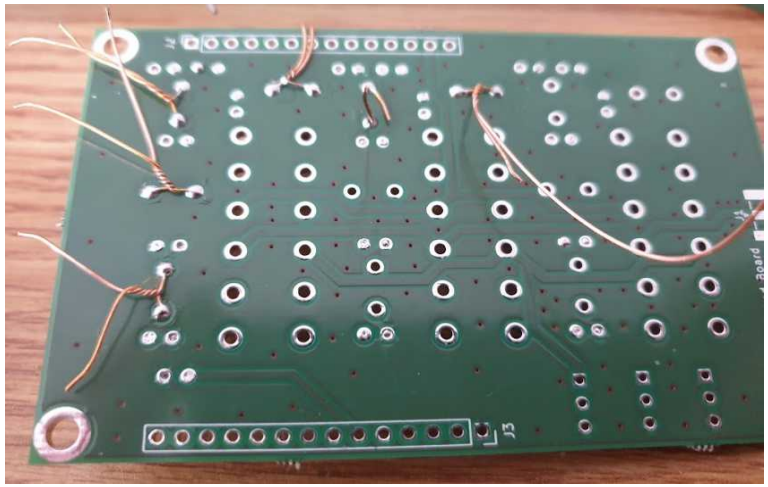


And finally we take care about RF PCB – **NEW Multi-Band PCB !**

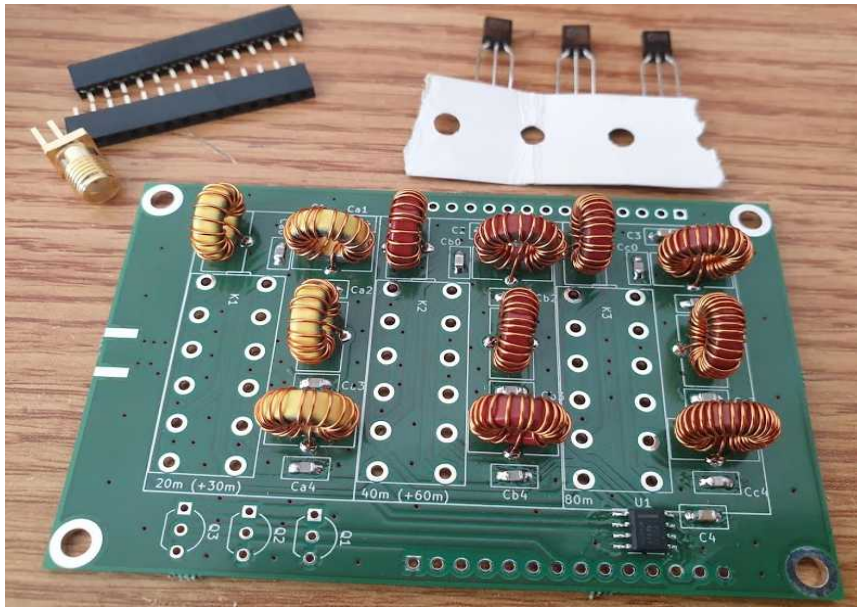
First place the Capacitors and the only SMT IC on the Board (U1) according to the Schematic and BOM Table below.
The footfrint allows to use THT Capacitors with 2,5mm Spacing and also 0805 SMT Type. I use what I have, but usually prefer SMT.
Use NP0 Capacitors with 100V.



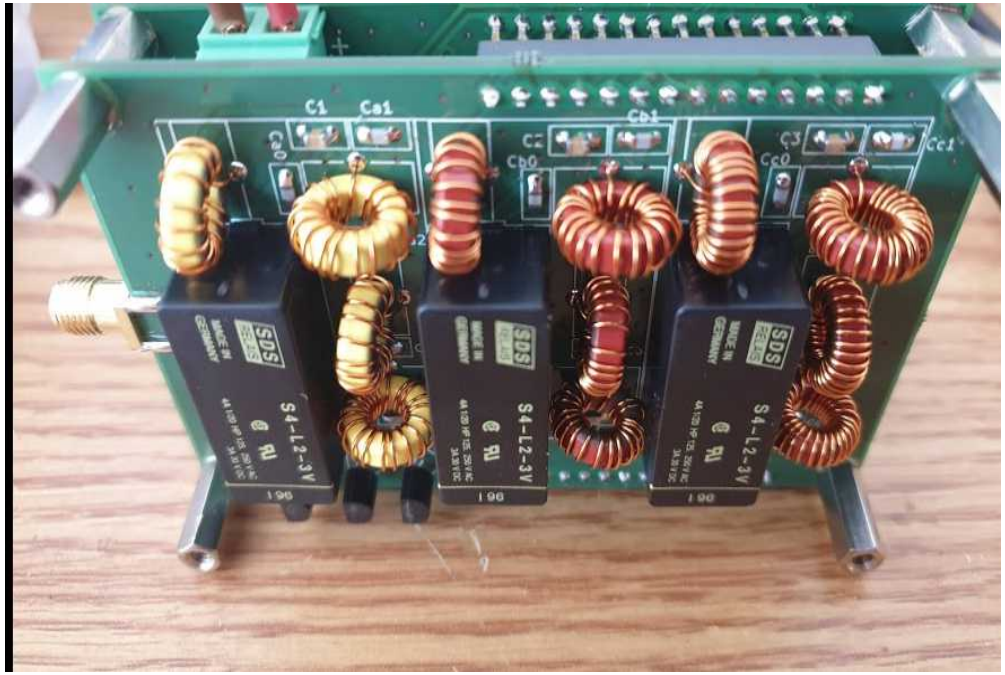
Then select Torids according to Schematic and wind them with the amount of turns according to Schematic. For Example: 0,11uH/n16/T37-6 means: Use Toroid Core T37-6 (= yellow one), put on 16 Turns, and if you can, measure Inductance and tweak until you have 0,77µH. The footprint on the PCB almost forces 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)



After all Toroids are Placed and soldered, it should look like next Photo. You can also guess what´s next.... Place and solder Jumper Connectors, SMA Jack and PA FET´s.



Last items that go on the RF Board are the Relays. Then Board is Ready for assembly and Test:



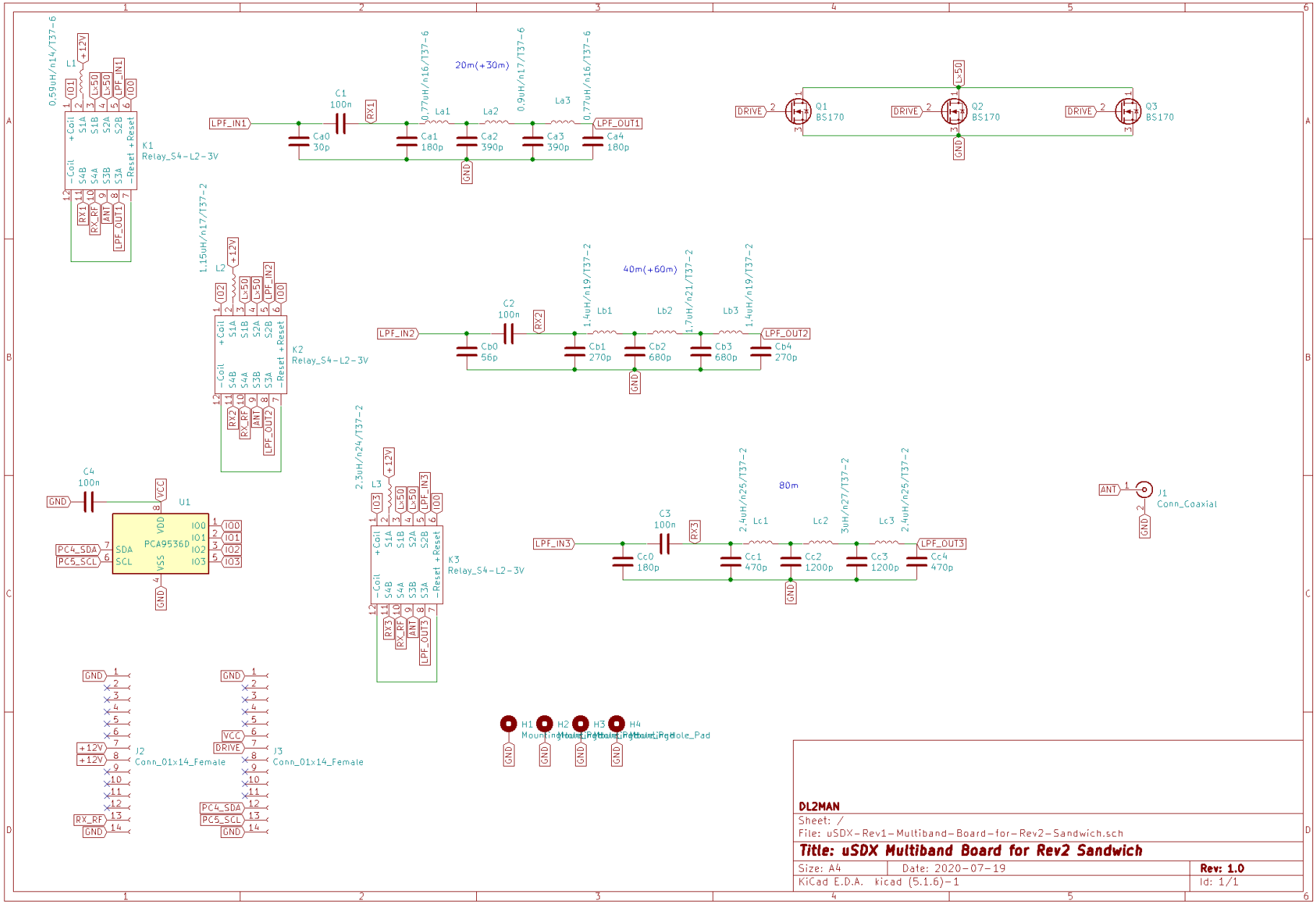
If you want to use the Relays, at this time (July 2020) you need the Software from Experimantal Branch (Search for „Latching“ in the Code).

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 60-80% 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:

Reference	Quantity	Value	Footprint
C1 C2 C3 C4	4	100n	Capacitor, X7R, 100V. THT 2,5mm or SMT 0805
Ca0	1	30p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
Ca1 Ca4 Cc0	3	180p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
Ca2 Ca3	2	390p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
Cb0	1	56p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
Cb1 Cb4	2	270p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
Cb2 Cb3	2	680p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
Cc1 Cc4	2	470p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
Cc2 Cc3	2	1200p	Capacitor, NP0, 100V. THT 2,5mm or SMT 0805
J1	1	Conn_Coaxial	SMA Connector, Edge Mount (stolen from Si5351 Breakout Boards)
J2 J3	2	Conn_01x14_Female	Connector_PinSocket_2.54mm:PinSocket_1x14_P2.54mm_Vertical
K1 K2 K3	3	Relay_S4-L2-3V	Relay, Latching 3V, 2 Coils, 4 Switches, (eg Panasonic S4-L2-3V or equal)
L1	1	0,59uH/n14/T37-6	14 Turns on T37-6 (Yellow) Core
L2	1	1,15uH/n17/T37-2	17 Turns on T37-2 (Red)Core
L3	1	2,3uH/n24/T37-2	24 Turns on T37-2 (Red)Core
La1 La3	2	0,77uH/n16/T37-6	16 Turns on T37-6 (Yellow) Core
La2	1	0,9uH/n17/T37-6	17 Turns on T37-6 (Yellow) Core
Lb1 Lb3	2	1,4uH/n19/T37-2	19 Turns on T37-2 (Red)Core
Lb2	1	1,7uH/n21/T37-2	21 Turns on T37-2 (Red)Core
Lc1 Lc3	2	2,4uH/n25/T37-2	25 Turns on T37-2 (Red)Core
Lc2	1	3uH/n27/T37-2	27 Turns on T37-2 (Red)Core
Q1 Q2 Q3	3	BS170	Package_TO_SOT_THT:TO-92_Inline_Wide
U1	1	PCA9536D	Package_SO:SOIC-8_3.9x4.9mm_P1.27mm


0.59uH/n14/T37-6
L1
+12V
IO1
LPF_IN1
Lx50
IO0
+Coil
S1A
S1B
S2A
S2B
-Coil
S4B
S4A
S3B
S3A
-Reset
+Reset
K1
Relay_S4-L2-3V
RX1
RX_RF
ANT
LPF_OUT1
C1
100n
RX1
20m(+30m)
0.77uH/n16/T37-6
La1
La2
0.9uH/n17/T37-6
La3
0.77uH/n16/T37-6
LPF_IN1
LPF_OUT1
Ca0
30p
Ca1
180p
Ca2
390p
Ca3
390p
Ca4
180p
GND
DRIVE
Q1
BS170
Q2
BS170
Q3
BS170
1.15uH/n17/T37-2
L2
IO2
K2
Relay_S4-L2-3V
RX2
LPF_IN2
LPF_OUT2
C2
100n
40m(+60m)
1.4uH/n19/T37-2
Lb1
1.7uH/n21/T37-2
Lb2
1.4uH/n19/T37-2
Lb3
Cb0
56p
Cb1
270p
Cb2
680p
Cb3
680p
Cb4
270p
2.3uH/n24/T37-2
L3
IO3
K3
Relay_S4-L2-3V
RX3
LPF_IN3
LPF_OUT3
C3
100n
80m
2.4uH/n25/T37-2
Lc1
3uH/n27/T37-2
Lc2
2.4uH/n25/T37-2
Lc3
Cc0
180p
Cc1
470p
Cc2
1200p
Cc3
1200p
Cc4
470p
C4
100n
U1
PCA9536D
VCC
VDD
VSS
SDA
SCL
IO0
IO1
IO2
IO3
PC4_SDA
PC5_SCL
ANT
J1
Conn_Coaxial
J2
Conn_01x14_Female
J3
Conn_01x14_Female
H1
H2
H3
H4
MountingHole_Pad
DL2MAN
Sheet: /
File: uSDX-Rev1-Multiband-Board-for-Rev2-Sandwich.sch
Title: uSDX Multiband Board for Rev2 Sandwich
Size: A4
Date: 2020-07-19
Rev: 1.0
KiCad E.D.A. kicad (5.1.6)-1
Id: 1/1

Alternatively you can built a Monoband PCB:

First place the Capacitors, matching the Band you want to build (see Table below).
The footfrint allows to use THT Capacitors with 2,5mm Scacing and also 0805 SMT Type.
Use NP0 Capacitors with minimum 50V. 100V Types are better and less prone to failure !

Then select and wind Torids according to your needs. 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 !

However, I found experimentally, that L1 and C1 Combination less critical then expected. More critical for output Power and efficieny are the values of LPF (C3,C4,C5,C6 and L2/L3/L4)

Ghetto Class E				
Band	**L1 (X=50 Ohm)**	**Toroid**	**n**	**C1**
80m	2,3µH	T37-2 (red)	24	150pF
60m	1,45µH	T37-2 (red)	19	100pF
40m	1,15µH	T37-2 (red)	17	82pF
30m	0,77µH	T37-6 (yellow)	16	56pF
20m	0,59µH	T37-6 (yellow)	14	39pF

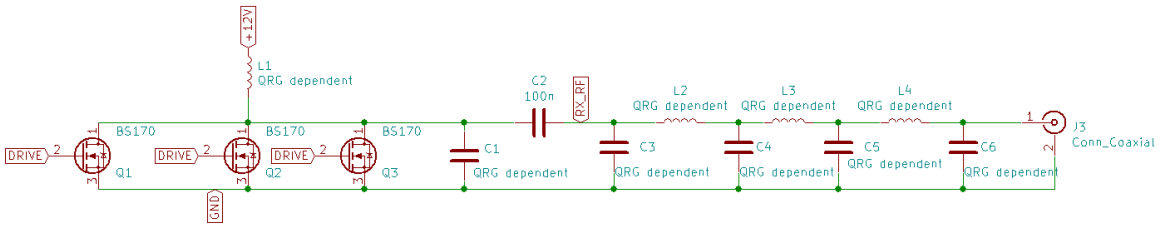
Tschebycheff LPF (50 Ohms in / 50 Ohms Out)

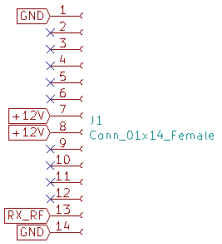
Band	**C3/C6**	**C4/C5**	**L3**	**L3 Toroid**	**L3 (n)**	**L2/L4**	**L2/L4 Toroid**	**L2 / L4 (n)**
80m	470pF	1200pF	3,0µH	T37-2 (red)	27	2,4µH	T37-2 (red)	25
60m	680pF	1200pF	2,3µH	T37-2 (red)	24	2,1µH	T37-2 (red)	23
40m	270pF	680pF	1,7µH	T37-2 (red)	24	1,4µH	T37-2 (red)	21
30m	270pF	560pF	1,3µH	T37-6 (yellow)	20	1,1µH	T37-6 (yellow)	19
20m	180pF	390pF	0,9µH	T37-6 (yellow)	17	0,77µH	T37-6 (yellow)	16

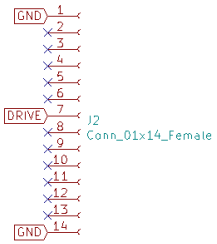
You finish with the 3x BS170´s, the SMA Connector and the Female Jumper Connectors

RF PCB			
L1,L2,L3,L4	4	QRG dependent	See Table above
J1,J2	2	Conn_01x14_Female	PinSocket_1x14_P2.54mm_Vertical Female
J3	1	Conn_Coaxial	SMA Connector from Si5351 Breakout Board (Edge Mountable)
C3,C4,C5,C6,C1	5	QRG dependent	See Table above
C2	1	100n	Either SMT 0805 or 2,54mm THT
H1,H2,H3,H4	4	MountingHole_Pad	Mount 15mm Spacers, Male Female as feet
Q1,Q2,Q3	3	BS170	TO-92 Package

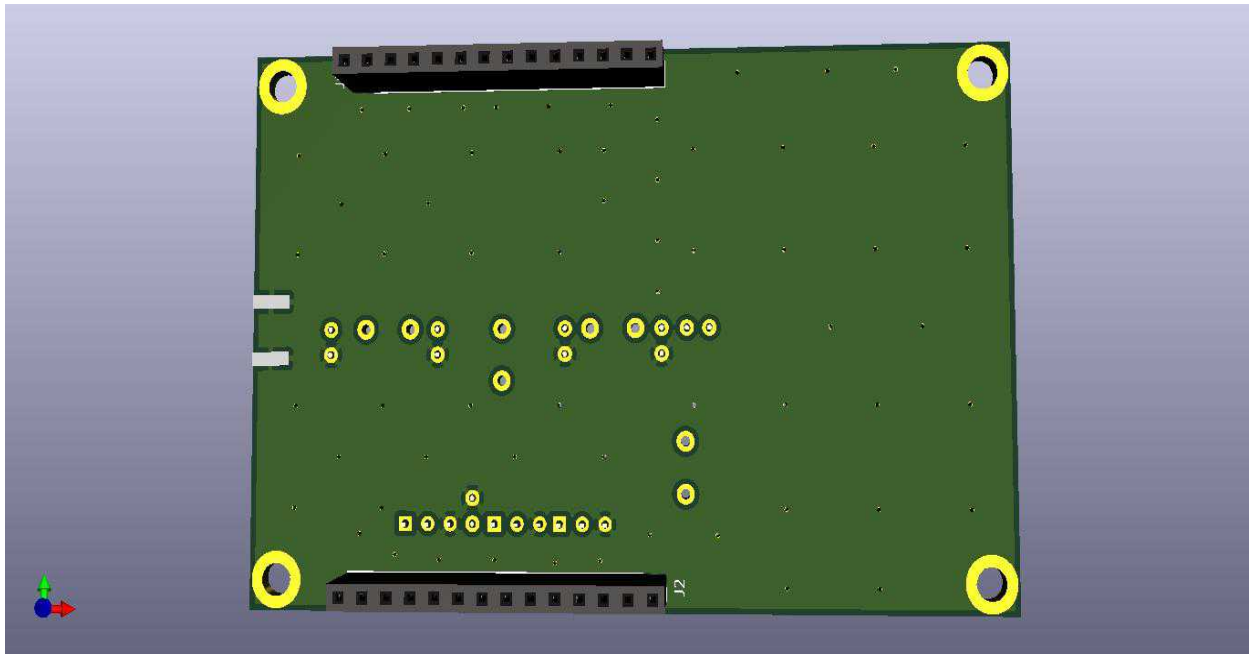
Have fun while building, and please report any bugs and errors while building to groups.io
or directly to me:
DL2MAN@gmx.de

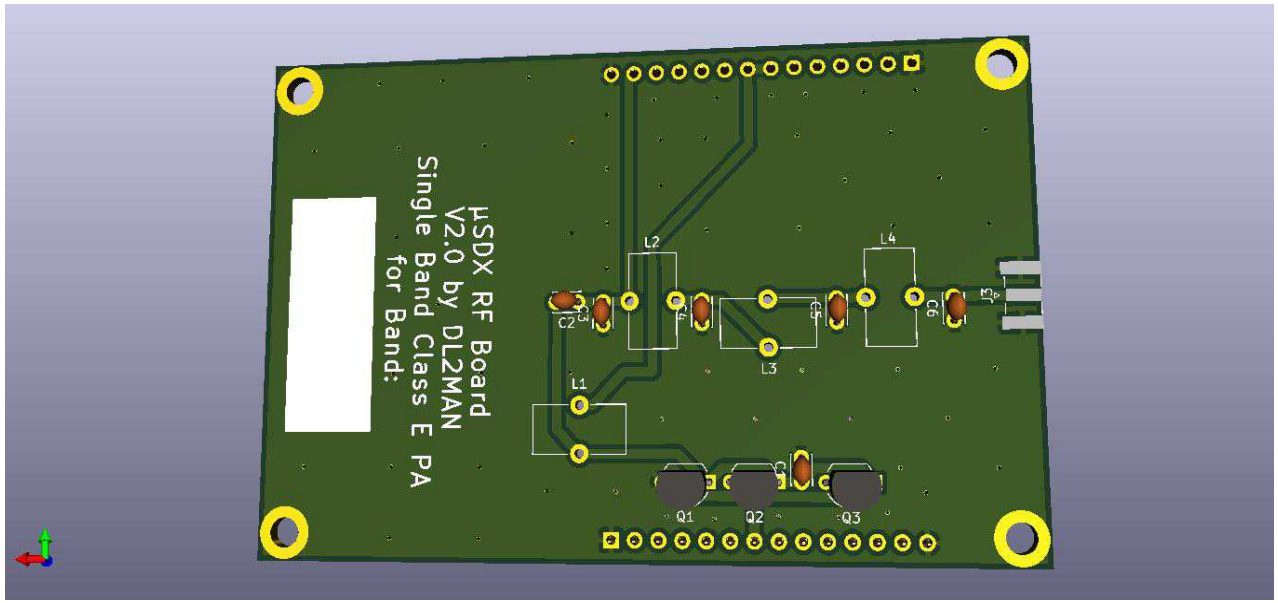






DL2MAN		
Sheet: /		
File: µSDX_RF_Rev2.sch		
Title: µSDX Monoband PA Board		
Size: A4	Date: 2020-07-05	**Rev: 2.0**
KiCad E.D.A. eeschema (5.1.6)-1		Id: 1/1


J2


µSDX RF Board
V2.0 by DL2MAN
Single Band Class E PA
for Band:
L2
L4
L3
L1
C2
C5
C6
Q1
Q2
Q3