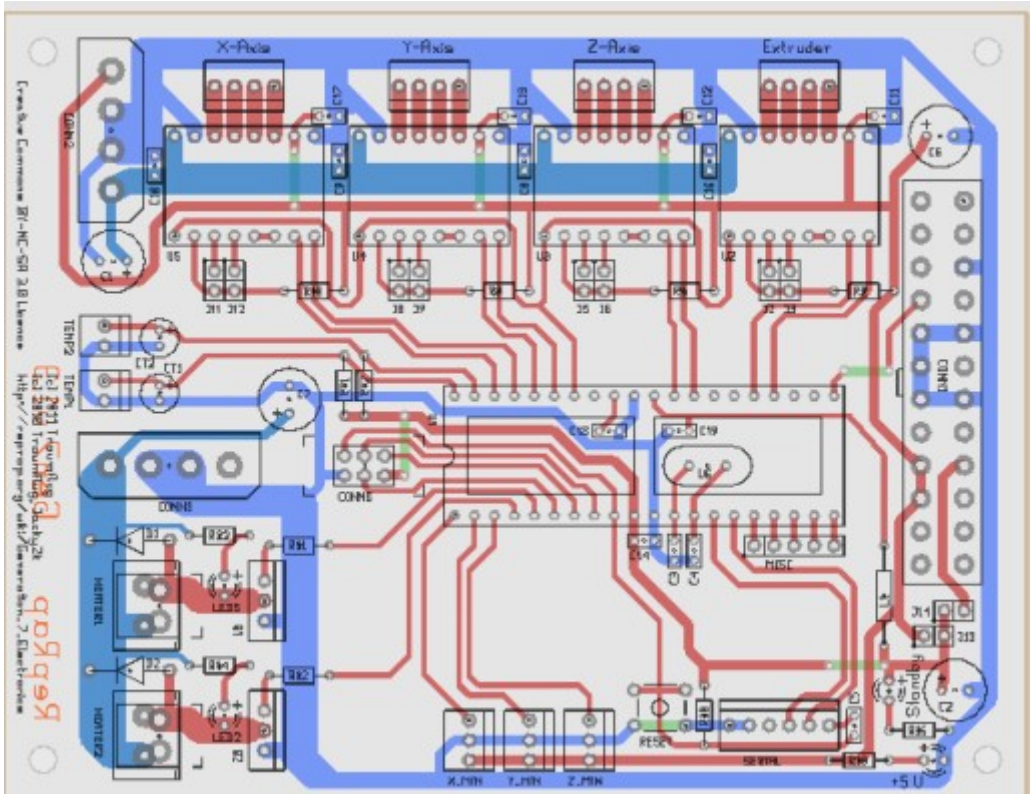
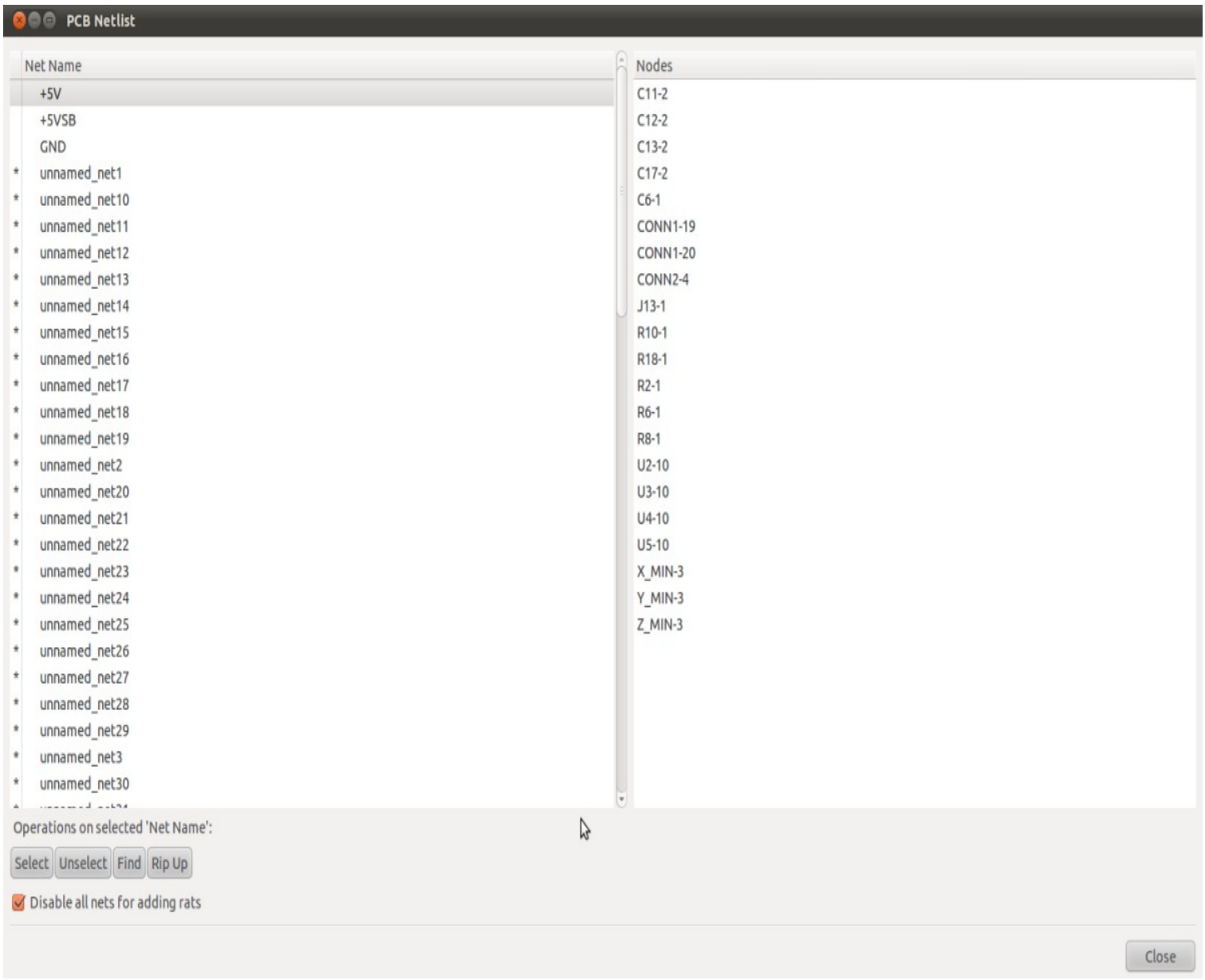
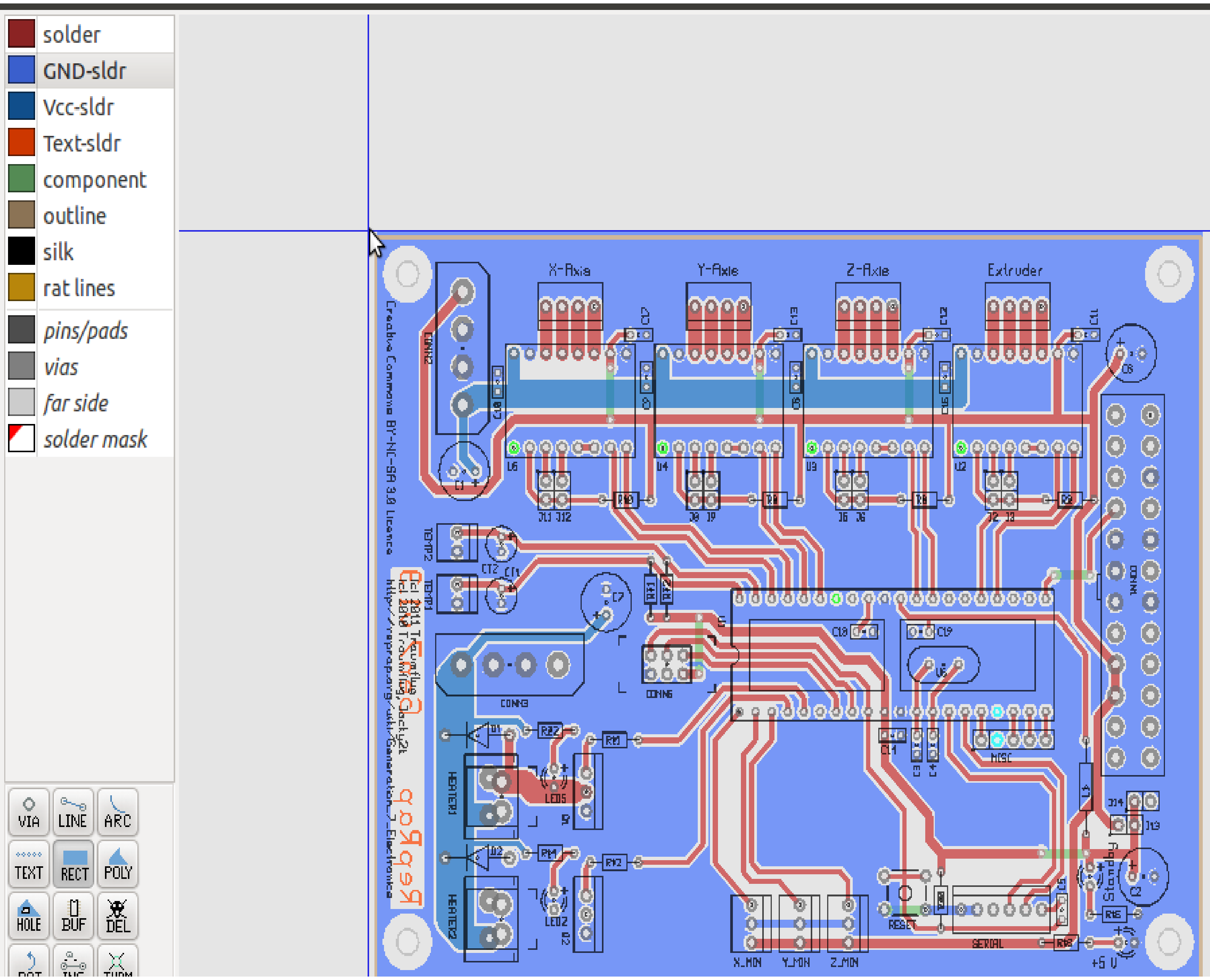
Open the layout in PCB.



- Switch to the "GND-sldr" layer.
- Remove all tracks on this layer (= all light blue ones = all of the GND net minus vias and bridges, find the net with Menu -> Window -> Netlist).
- Draw a RECT (find the tool in the left bar) as big as the entire board.
- Do an "optimize rats nest" (o-key).


PCB Netlist
Net Name
+5V
+5VSB
GND
unnamed_net1
unnamed_net10
unnamed_net11
unnamed_net12
unnamed_net13
unnamed_net14
unnamed_net15
unnamed_net16
unnamed_net17
unnamed_net18
unnamed_net19
unnamed_net2
unnamed_net20
unnamed_net21
unnamed_net22
unnamed_net23
unnamed_net24
unnamed_net25
unnamed_net26
unnamed_net27
unnamed_net28
unnamed_net29
unnamed_net3
unnamed_net30
Nodes
C11-2
C12-2
C13-2
C17-2
C6-1
CONN1-19
CONN1-20
CONN2-4
J13-1
R10-1
R18-1
R2-1
R6-1
R8-1
U2-10
U3-10
U4-10
U5-10
X_MIN-3
Y_MIN-3
Z_MIN-3
Operations on selected 'Net Name':
Select
Unselect
Find
Rip Up
Disable all nets for adding rats
Close


solder
GND-sldr
Vcc-sldr
Text-sldr
component
outline
silk
rat lines
pins/pads
vias
far side
solder mask
VIA
LINE
ARC
TEXT
RECT
POLY
HOLE
BUF
DEL
X-Axis
Y-Axis
Z-Axis
Extruder
CONN3
CONN6
RESET
SERIAL
X_MIN
Y_MIN
Z_MIN
+5 V