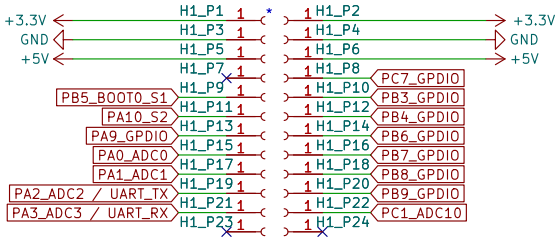
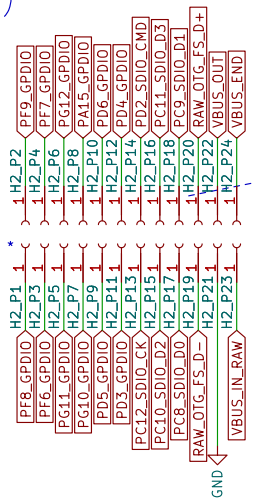
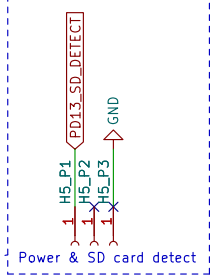
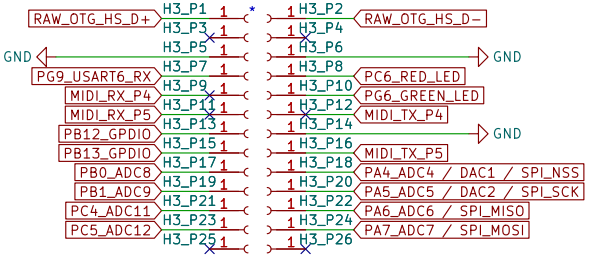
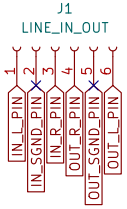
Ksoloti Core v0.4+ headers (top view)











SDIO

checked

File: sdio.kicad_sch

Audio IO

checked

File: audio_io.kicad_sch

Power

checked

File: power.kicad_sch

CV Inputs

checked

File: cv_in.kicad_sch

LEDs and Buttons

checked

File: led_out.kicad_sch

CV outputs

checked

File: cv_out.kicad_sch

digital mic

checked

File: digital_mic.kicad_sch

MIDI

checked

File: midi.kicad_sch

usb

checked

File: usb.kicad_sch

Gate Outputs

checked

File: gate_io.kicad_sch

expansion_header

checked

File: untitled.kicad_sch

2 audio inputs, 2 audio outputs, Eurorack level

8 pots
4 CV inputs (P1–P4) are summed with pots 1–4
4 independent CV inputs (A–D), non–trimmable (+/–5V)
2 independent CV inputs (X, Y), trimmable V/oct (+/–5V or 0..10V via jumper)

2 buttons (S3, S4)
2 encoders with switch (E1, E2)

4 fadable duo–color LEDs, 2 Axo status LEDs

2 gate outputs, ca. 10.3V
2 CV outputs, trimmable V/oct (+/–5V or 0..10V via jumper)

1 OLED display, 128x64px

FID1 Fiducial

FID2 Fiducial

FID3 Fiducial

changelog

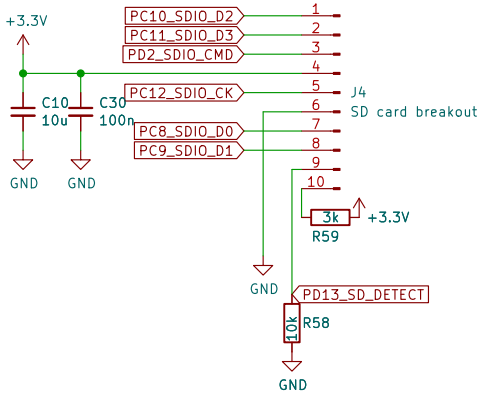
CHANGELOG

File: changelog.kicad_sch

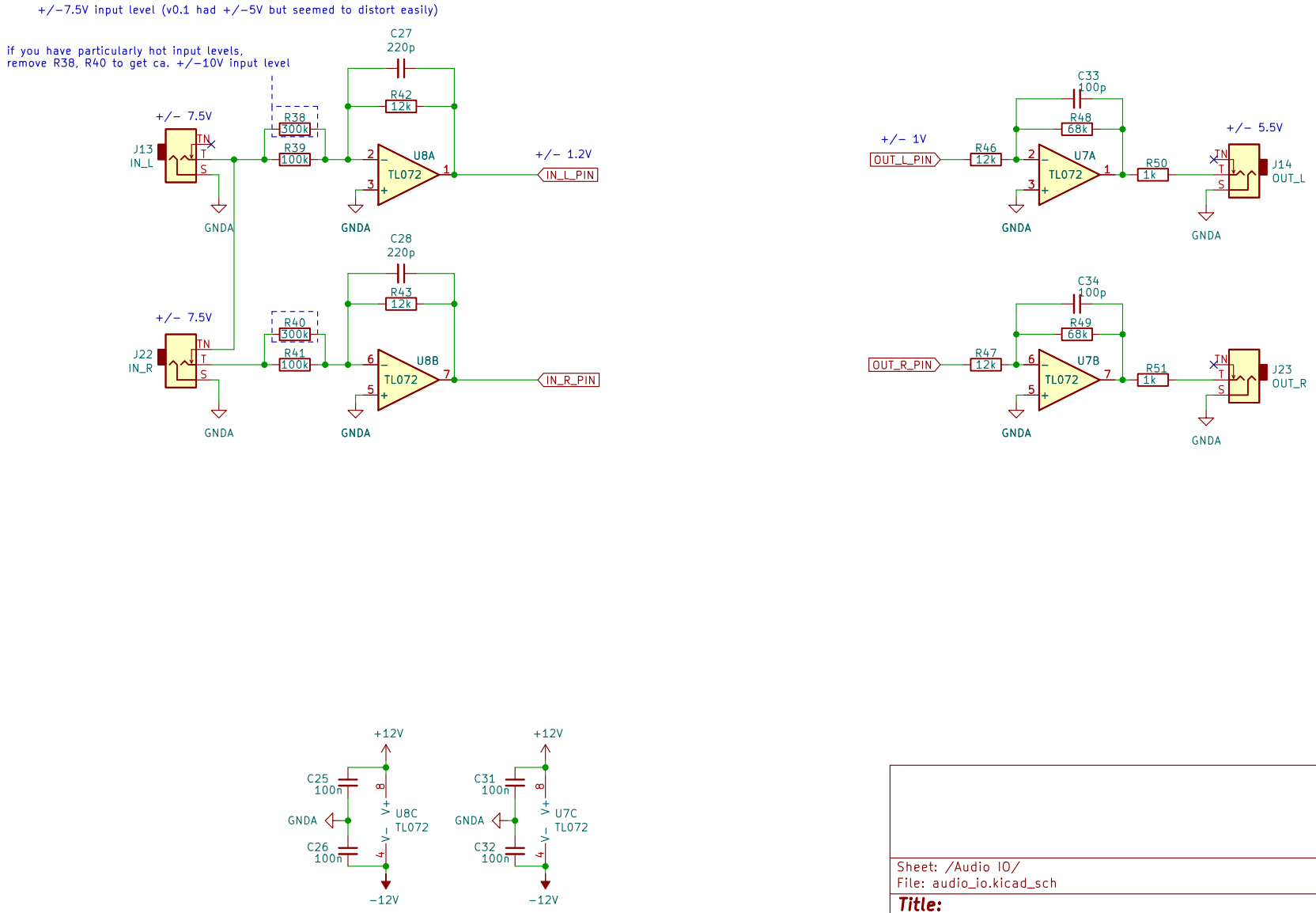
Sheet: /
File: ksoloti_big_genes.kicad_sch

Title:

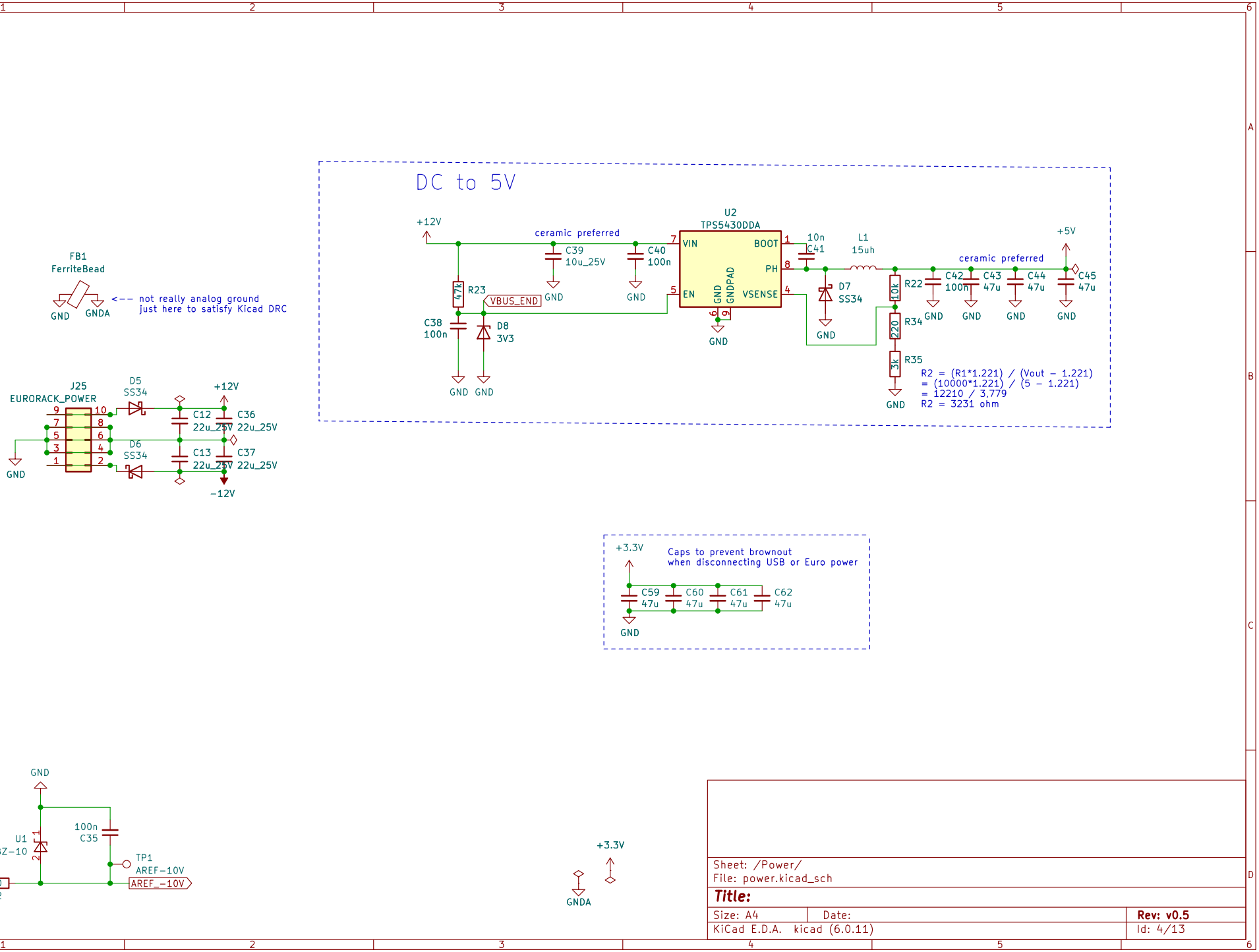
Size: A4	Date: 2024–02–06	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 1/13



Sheet: /SDIO/		
File: sdio.kicad_sch		
Title:		
Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 2/13

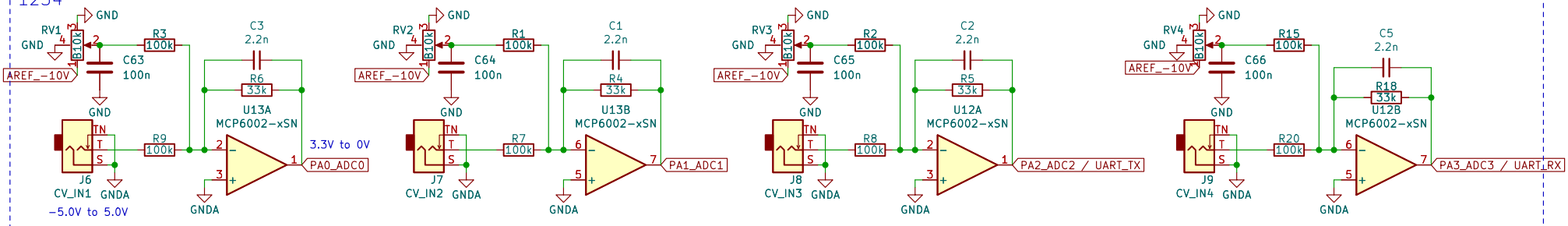


Sheet: /Audio IO/		
File: audio_io.kicad_sch		
Title:		
Size: A4	Date: 2024-03-05	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 3/13


DC to 5V
U2
TPS5430DDA
+12V
ceramic preferred
C39
10u_25V
C40
100n
VIN
BOOT
PH
EN
GND
GNDPAD
VSENSE
10n
C41
L1
15uh
+5V
ceramic preferred
R23
47k
VBUS_END
C38
100n
D8
3V3
D7
SS34
R22
10k
R34
220
R35
3k
C42
100n
C43
47u
C44
47u
C45
47u
R2 = (R1*1.221) / (Vout - 1.221)
= (10000*1.221) / (5 - 1.221)
= 12210 / 3,779
R2 = 3231 ohm
FB1
FerriteBead
GND
GNDA
<-- not really analog ground
just here to satisfy Kicad DRC
J25
EURORACK_POWER
D5
SS34
D6
SS34
+12V
-12V
C12
22u_25V
C36
22u_25V
C13
22u_25V
C37
22u_25V
+3.3V
Caps to prevent brownout
when disconnecting USB or Euro power
C59
47u
C60
47u
C61
47u
C62
47u
U1
LM4040DBZ-10
100n
C35
TP1
AREF_-10V
330
R72
Sheet: /Power/
File: power.kicad_sch
Title:
Size: A4
Date:
Rev: v0.5
KiCad E.D.A. kicad (6.0.11)
Id: 4/13

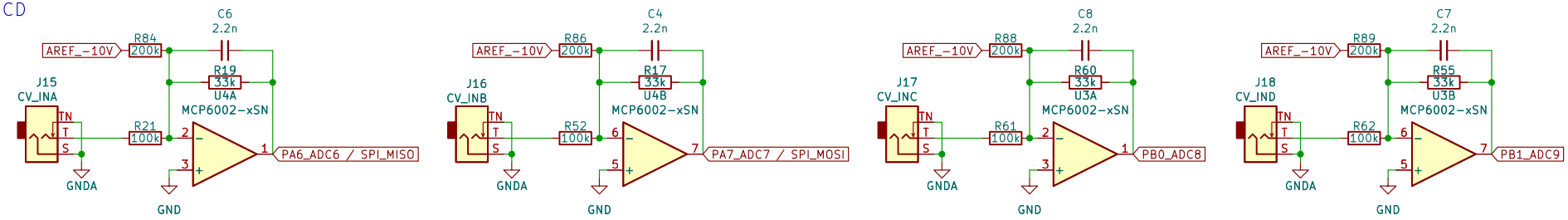
Pots / CV inputs

1234

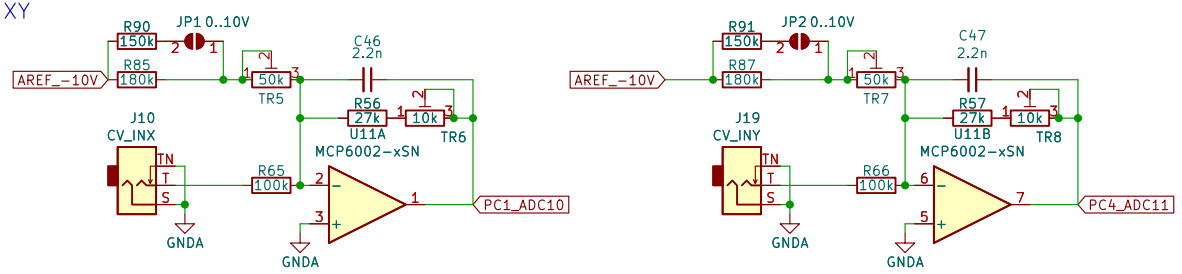


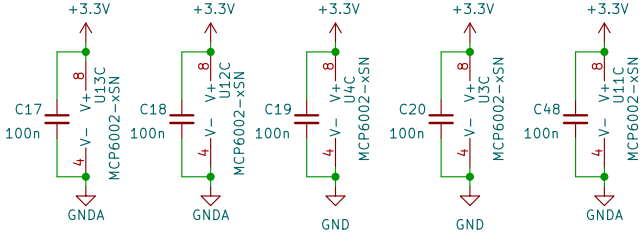
5678

ABCD



XY

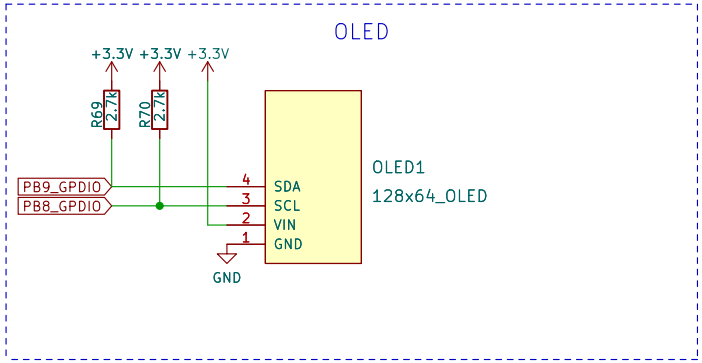


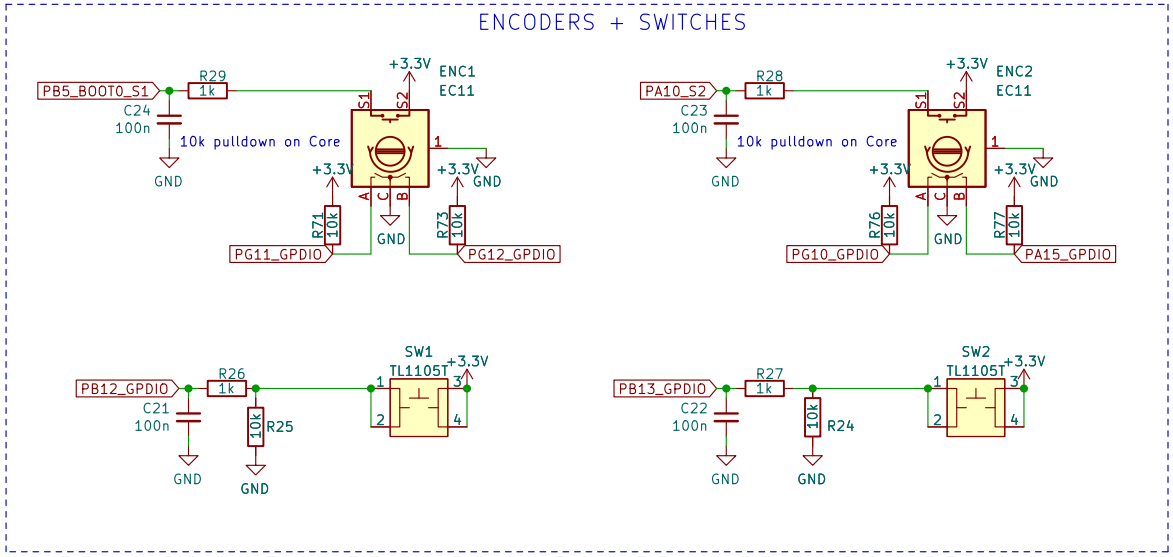


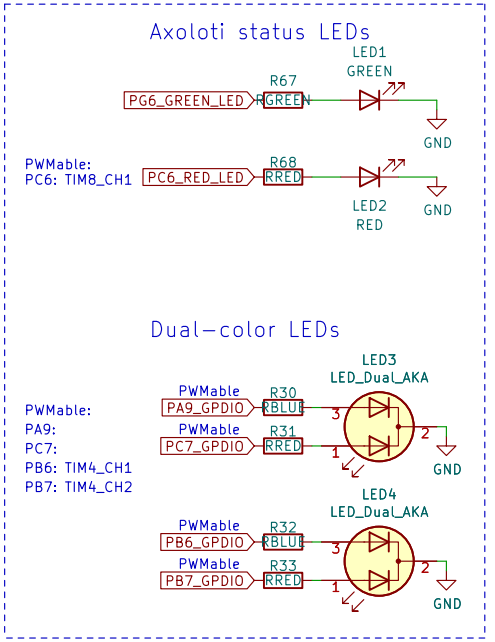
Sheet: /CV Inputs/
File: cv_in.kicad_sch

Title:

Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 5/13







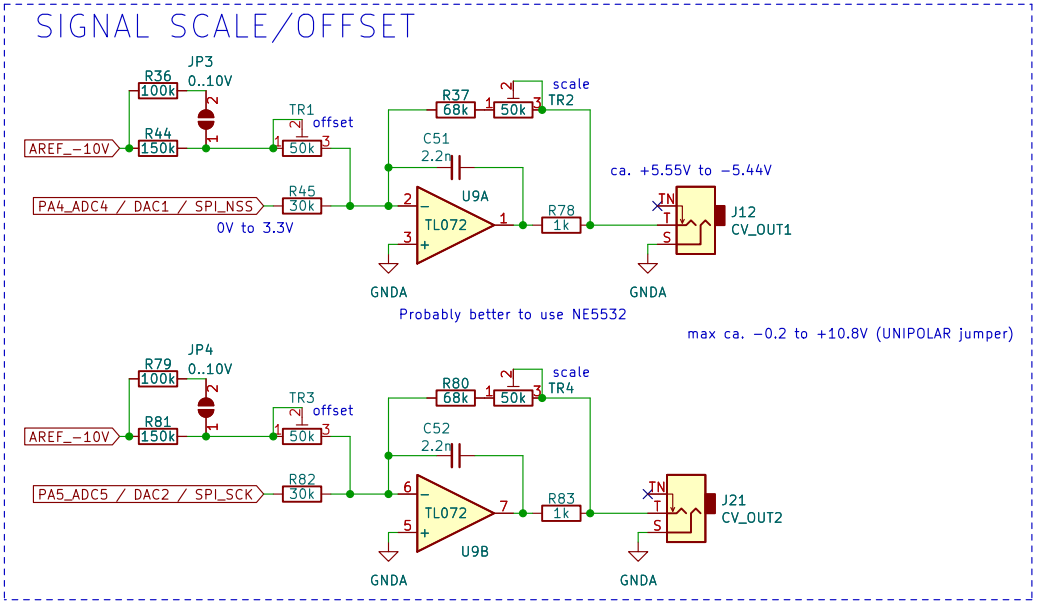
Sheet: /LEDs and Buttons/
File: led_out.kicad_sch

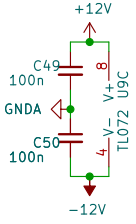
Title:

Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 6/13

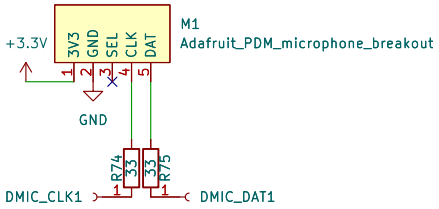
CV outputs

SIGNAL SCALE/OFFSET

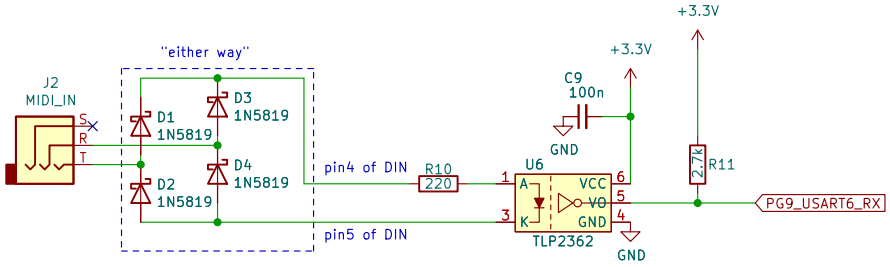




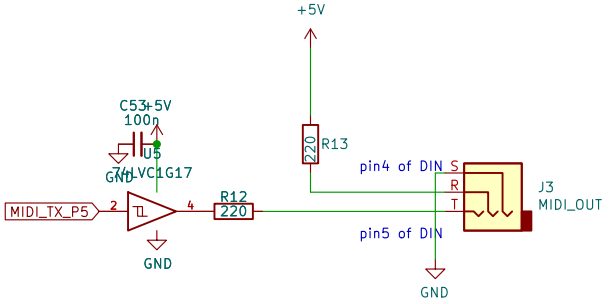
Sheet: /CV outputs/	
File: cv_out.kicad_sch	
Title:	
Size: A4	Date:
KiCad E.D.A. kicad (6.0.11)	**Rev: v0.5**
	Id: 7/13



Sheet: /digital mic/		
File: digital_mic.kicad_sch		
Title:		
Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 8/13

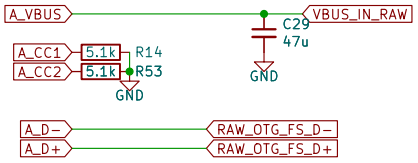


conforming to TRS MIDI specs
https://www.midi.org/midi-articles/trs-specification-adopted-and-released

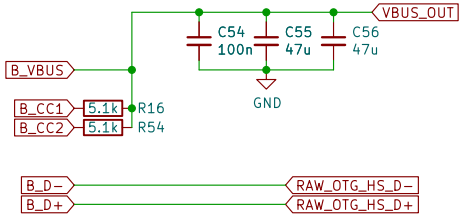


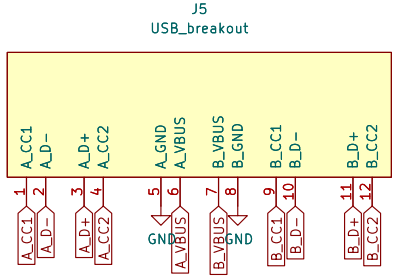
Sheet: /MIDI/		
File: midi.kicad_sch		
Title:		
Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 9/13

USB PROGRAMMER

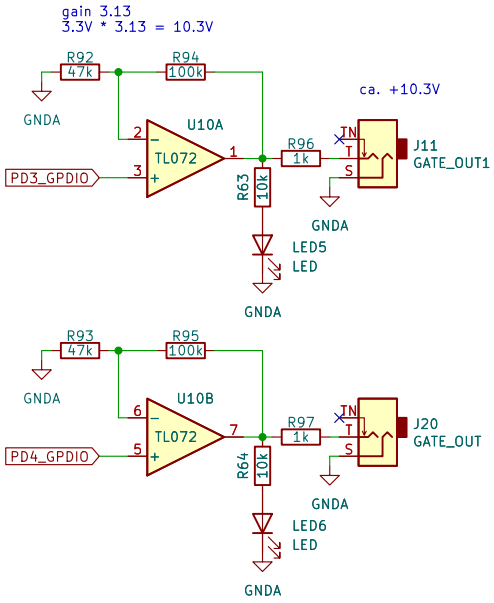


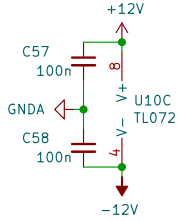
USB HOST





Sheet: /usb/		
File: usb.kicad_sch		
Title:		
Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 10/13

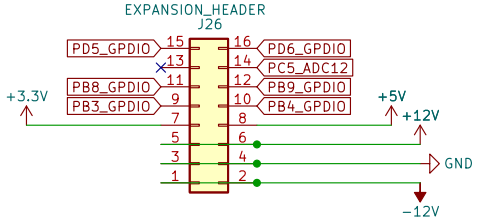




Sheet: /Gate Outputs/
File: gate_io.kicad_sch

Title:

Size: A4 | Date: | **Rev: v0.5**
KiCad E.D.A. kicad (6.0.11) | Id: 11/13



Sheet: /expansion_header/		
File: untitled.kicad_sch		
Title:		
Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 12/13

Changelog

v0.3 done – prototype ordered 2024-02-07

v0.4 done – prototype ordered 2024-03-06

- swap pins PD5 and PD6 on expansion header (consistency)
- rework USB connector and SD card connector footprints, panel holes
- Use pseudo-SMD pin socket for J1 line in/out to Core
- Use LM4040-10 instead of CJ431 for stabler VREF
- Add extra caps on 3.3V rail
- Use "thin" OLED footprint (slightly different dimensions)
- Adjust Thonkiconn stereo jacks footprint size, position
- Fix encoder pins not connecting to any MCU pin

v0.5 done – prototype ordered 2024-03-19

- Add filter caps to pots 1-4
- Rework USB connectors: design breakout board holding two horizontal connectors and pin headers

Sheet: /changelog/		
File: changelog.kicad_sch		
Title:		
Size: A4	Date:	**Rev: v0.5**
KiCad E.D.A. kicad (6.0.11)		Id: 13/13