

Sheet: /codec_adau1961/
File: codec_adau1961.kicad_sch

Title: Ksoloti Core

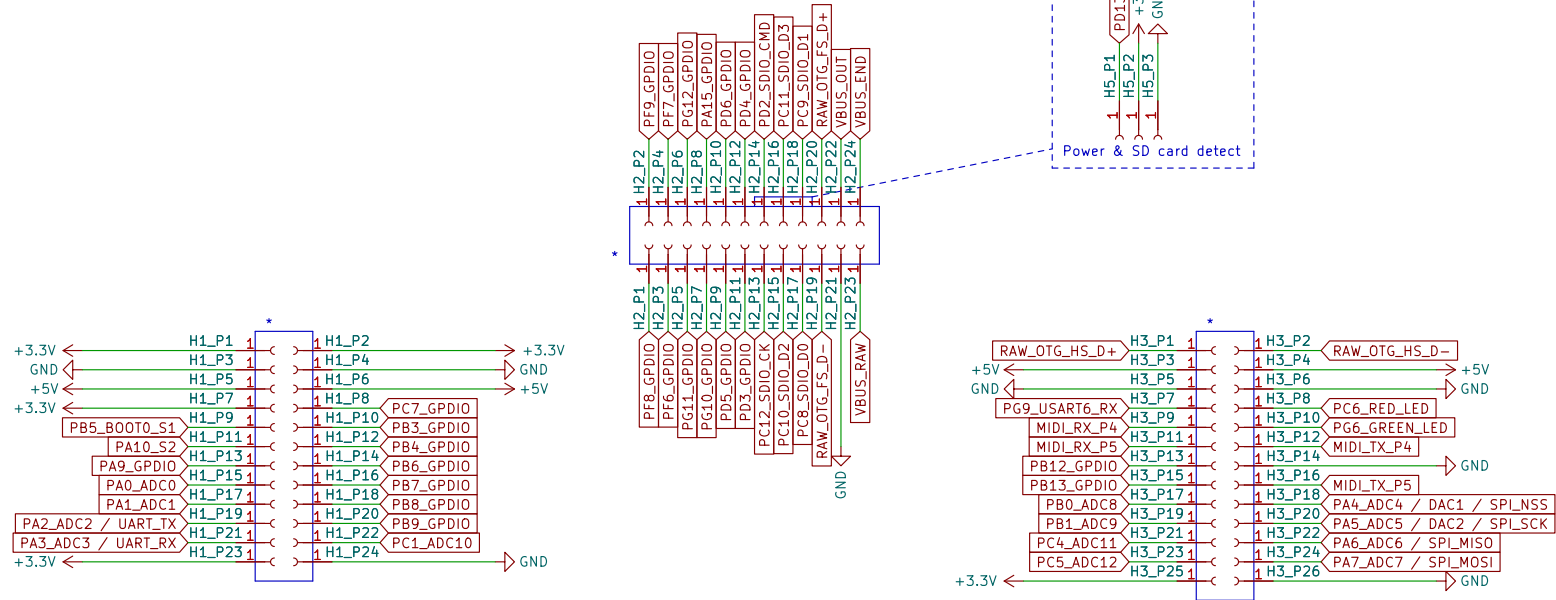
Size: A4 Date: 2023-08-21

KiCad E.D.A. kicad (6.0.11)

Rev: v0.6

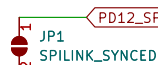
Id: 2/8

"GPIO" headers (PCB component side view)



Power & SD card detect

RESERVED for future SPILINK implementation
Leave as is for regular use.



Digital microphone header



Sheet: /headers/
File: headers.kicad_sch

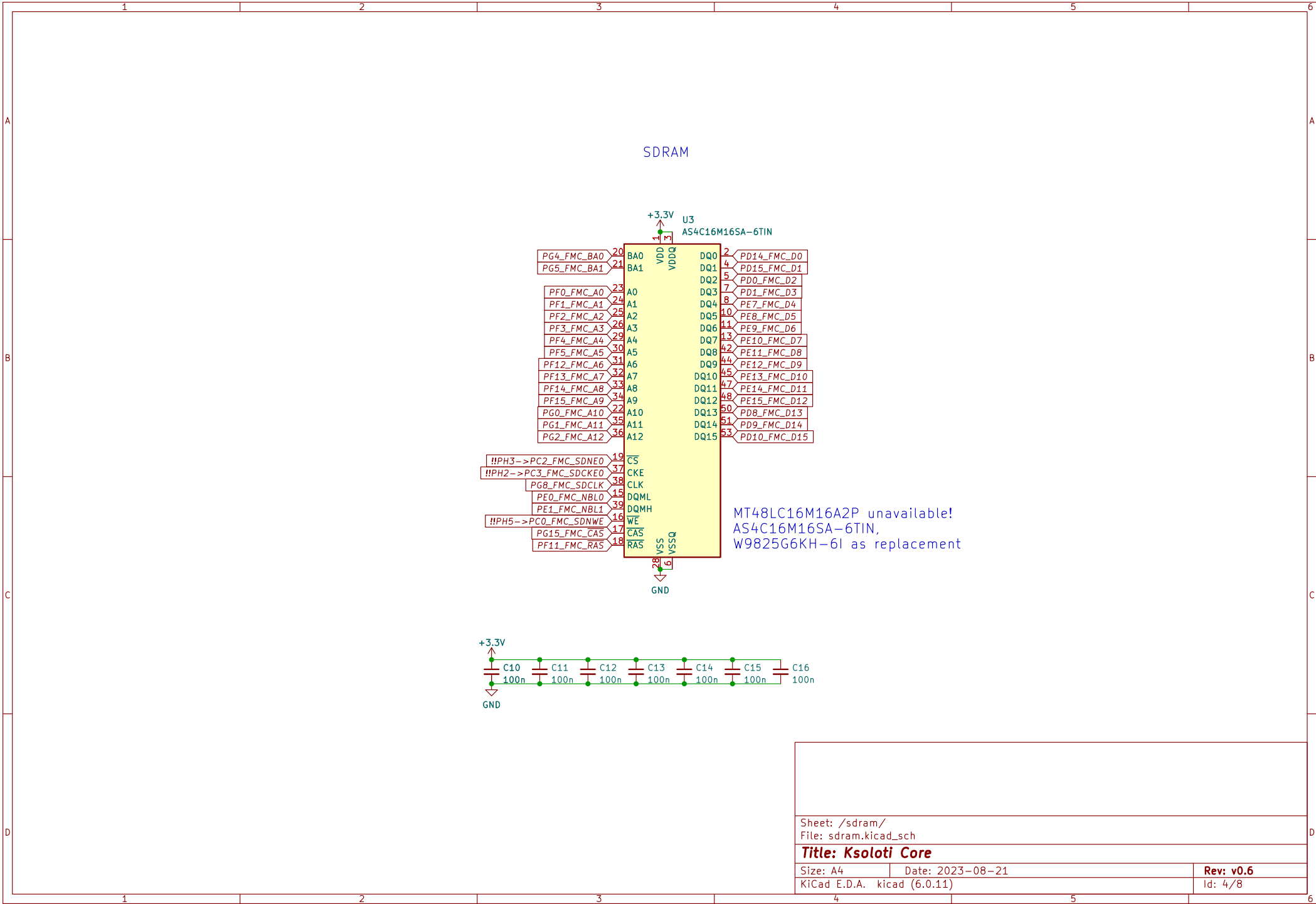
Title: Ksoloti Core

Size: A4 Date: 2023-08-21

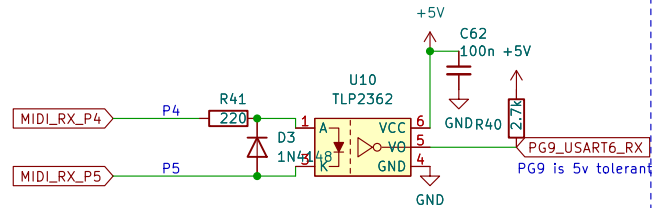
KiCad E.D.A. kicad (6.0.11)

Rev: v0.6

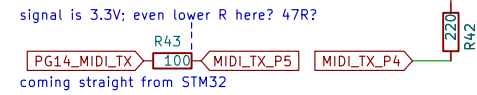
Id: 3/8



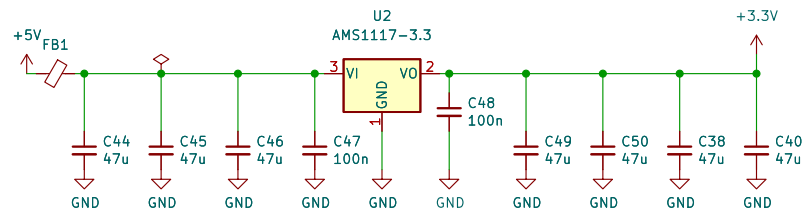
MIDI in. (OPTIONAL footprints on the bottom side of the PCB – not placed by default)



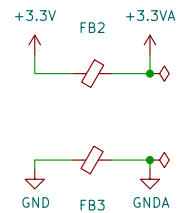
MIDI out. You can connect a MIDI jack directly to M04, GND, and M05.



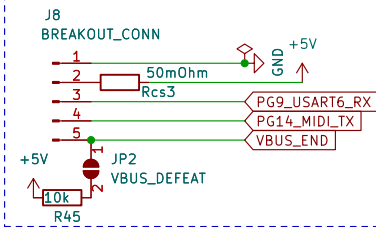
3.3V digital power supply



Analog power
(audio path only)



To power/midi board



TODO: pinout inconsistent with Axoloti Core

Sheet: /power_midi/
File: power.kicad_sch

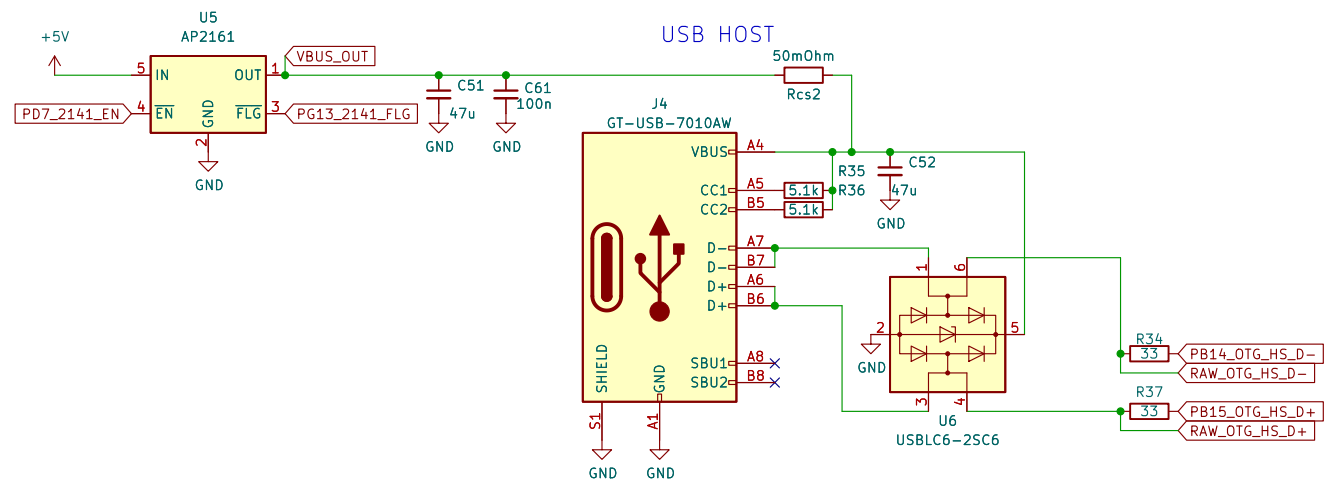
Title: Ksoloti Core

Size: A4 Date: 2023-08-21

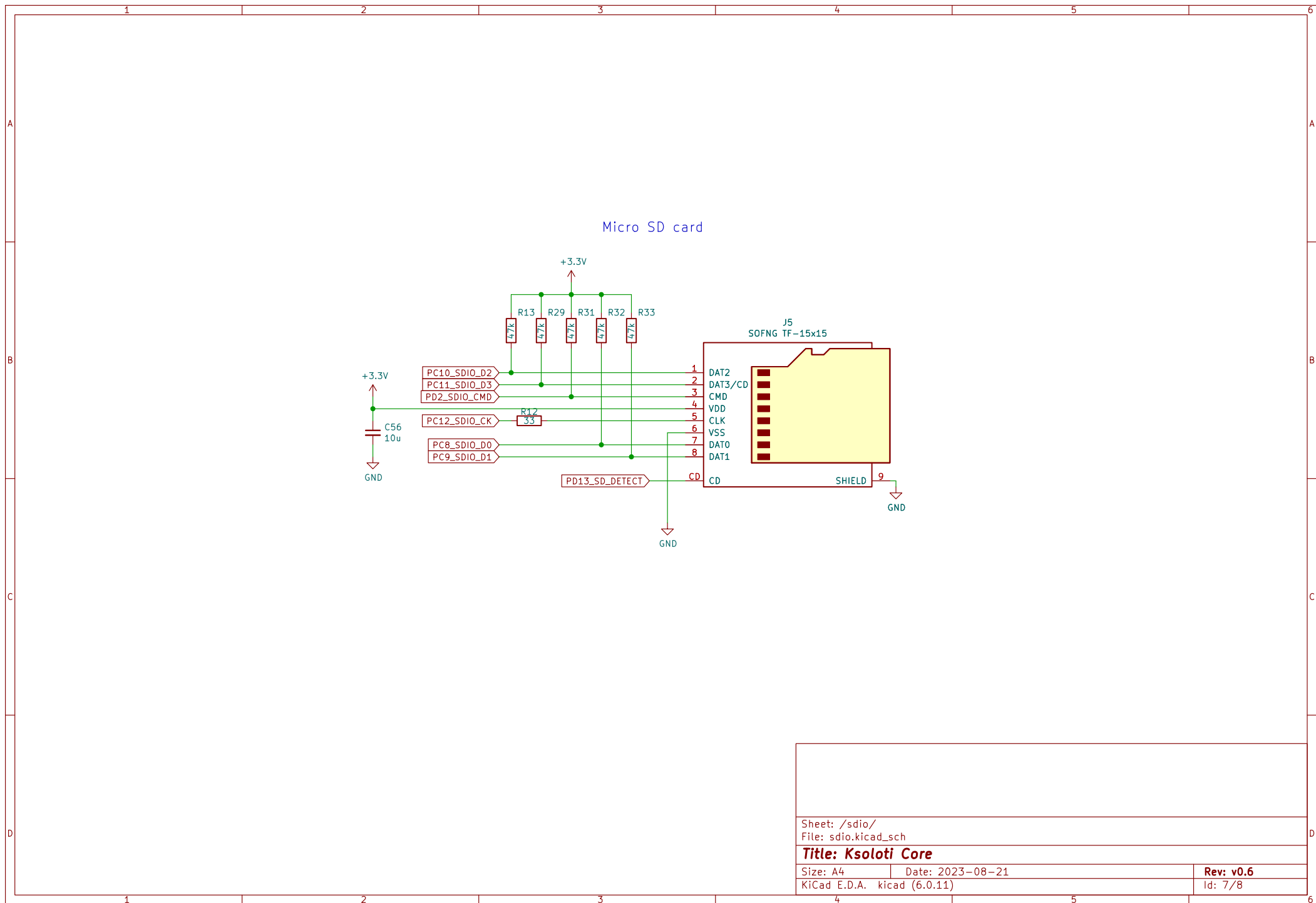
KiCad E.D.A. kicad (6.0.11)

Rev: v0.6

Id: 5/8



Rev: v0.6
Id: 6/8



	1	2	3	4	5	6
A	<p>v0.1 done -- prototype ordered 2023/04/13</p> <p>v0.2 done -- prototype ordered 2023/04/26</p> <p>Changes in v0.2:</p> <ul style="list-style-type: none">– (even) higher series resistors for LEDs– break out PA15 as GPDIO17, PD3 as GPDIO18– combine S1/BOOT0 pin and GPDIO2 (PB5)– untent some vias to make them usable as test points– move SD card slot to make space for mounting hole– add fiducials and M2.5 mounting holes– add footprint for SMD 3.5mm headphone jack (PJ–320D) on bottom– add digital PDM microphone connected to SPI3 (I2S3)– pull PG3 up to VDD to have a way in firmware to check board version– change HP output pinout so that it matches line out. One thing less that could go wrong!					A
B	<p>v0.3 done -- prototype ordered 2023/06/21</p> <p>Changes in v0.3:</p> <ul style="list-style-type: none">– Break out PA9 as GPDIO19, PC7 as GPDIO20, PB12 as GPDIO21, PB13 as GPDIO22 (was running out of PWMable and general digital pins)– Increase capacitors on 5V rail (3.3V regulator input) and 3.3V output to avoid MCU reset due to power drop when connecting external 5V while core is powered via USB.– Connect both physical legs of each SMD switch to their respective nets (not just to one physical leg of each side)– Increase mounting holes to M3, squeeze in two more mounting holes – total 4 now– Modify footprints of electrolytic caps so that they can alternatively accomodate 1206 ceramic caps– Remove via “untent” test points – they were creating risks when hand soldering ICs– Reroute PDM mic to interface directly with the codec. Now the codec can be setup in your patch to take either the mono mic or the stereo line in as input.– Add expansion header for optional second digital microphone– Move power–midi board connector J11 north by 2.54mm (to align with new power–midi board)– Revise ferrite beads (some unnecessary)– Revise 3.3V versus 3.3VA (3.3VA and GNDA now reserved for audio and codec only, 3.3V and GND for digital and now also for STM32 ADC/DAC).– add dedicated pins for 3.3VA and GNDA in the codec section, should they be required.– Add H3_P25 and H3_P26 so that 3.3V is available on the right header.– Make 5V available on the left header.– Add current sensing resistors to 5V in, USB port in, and USB host out, to be able to measure current requirements.– Add MIDI out circuit. MIDI output can now be used out of the box by connecting a connector to the header.– Add optional MIDI in circuit on the back side of the PCB. Users can solder the respective parts and get MIDI input by soldering a DIN or TRS connector to the header.					B
	<p>v0.4 done -- prototype ordered 2023/08/11</p> <p>Changes in v0.4:</p> <ul style="list-style-type: none">– Change R43 (MIDI out) from 220R to 100R.– Adjust 3.3V zone around U2 (3.3V regulator).– Adjust keepout zone around Y1 (8Mhz crystal).– Move USB sockets north by 0.5mm to let them poke through panels. Note that this will increase overall Y–dimension of the board to ca. 90mm.– Add SPILINK master/sync jumper (JP1) and “Multiprocessor Sync” header (X4). RESERVED for future SPILINK implementation. Leave as is for regular use.– Change header pinout so that related pins are next to each other:<ul style="list-style-type: none">* H3: Move SD card pins together. Move PF6..9 pins so that they are together with the other GPDIO pins.* Move MIDI–related pins together.* Move SW1, SW2 position and header pins together.* Align pins in order where possible (PA0..3, PA4..7 etc.)– Add 10k pull–down resistor to Switch 2.					
C	<p>v0.5 done -- ordered 2023/09/22</p> <p>Changes in v0.5:</p> <ul style="list-style-type: none">– Remove PDM mic, add optional gyroscope/acceleroscope via I2C2.– Rename MIDI pins to Mlx, MOx.– Fix swapped labeling of LED1 and LED2 to conform with Axo tradition. Rename LED pins to L1G and L2R to indicate color.– Add 47uF caps C38, C40 to 3.3V rail for more stability when USB or external power is (accidentally) unplugged					C
D	<p>v0.6 WIP – second prototype ordered 2024/03/06</p> <p>Changes in v0.6:</p> <ul style="list-style-type: none">– In USB host port, use AP2161 instead of AP2141 to allow for up to 1A of current.– MT48LC16M16A2P (SDRAM) part shortage. Use AS4C16M16SA–6TIN (W9825G6KH–6I; IS42S16160J–6TLI?)– SRV05–4 price fluctuation. Use USBLC6–2SC6.– Add jumper to disable VBUS power (useful if synced via spilink)– Remove gyroscope (feature creep, firmware bloat)– Change layer sequence – SDRAM lines run on inner layer.– Realign R42, R43 (MIDI out circuit)– Improve MI4, MI5, optional MIDI in circuit labeling– Adjust USB Type–C connector footprint– Use TLP2362 as optional optocoupler (smaller footprint)					D
	1	2	3	4	5	6

Sheet: /changelog/ File: changelog.kicad_sch	
Title: Ksoloti Core	
Size: A4	Date: 2023–08–21
KiCad E.D.A. kicad (6.0.11)	
Rev: v0.6	
Id: 8/8	