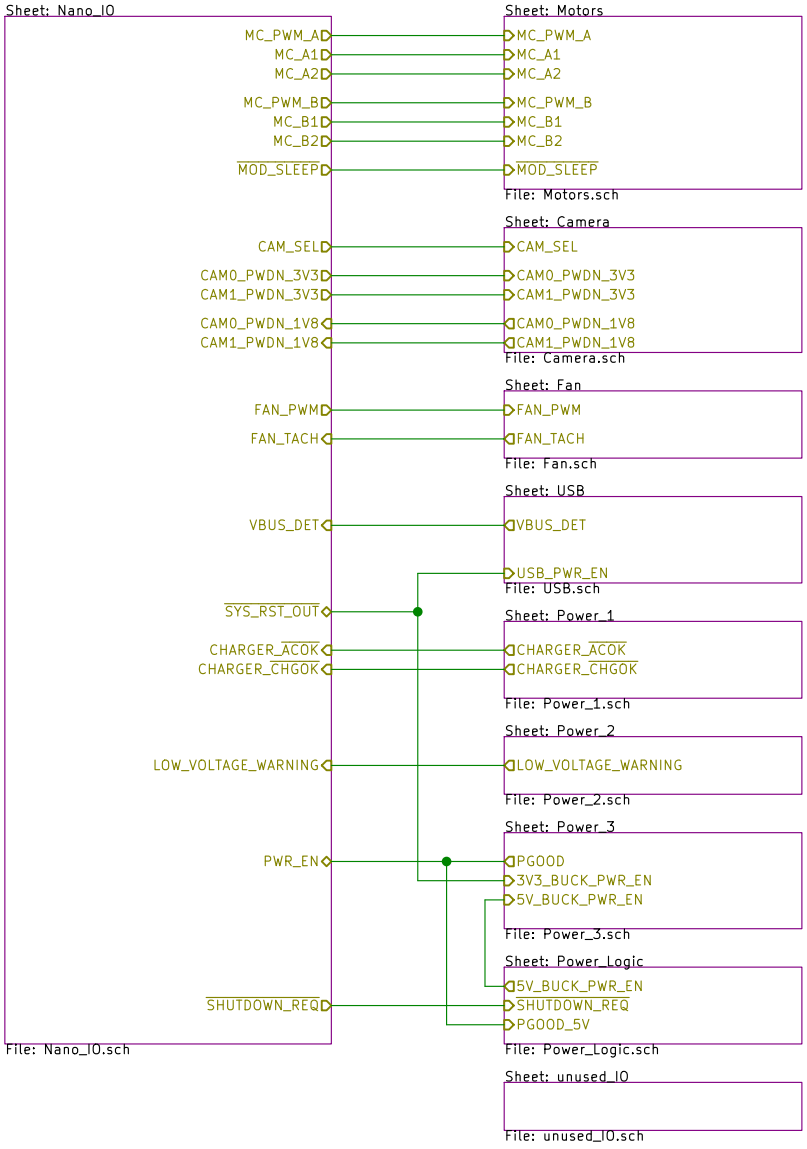
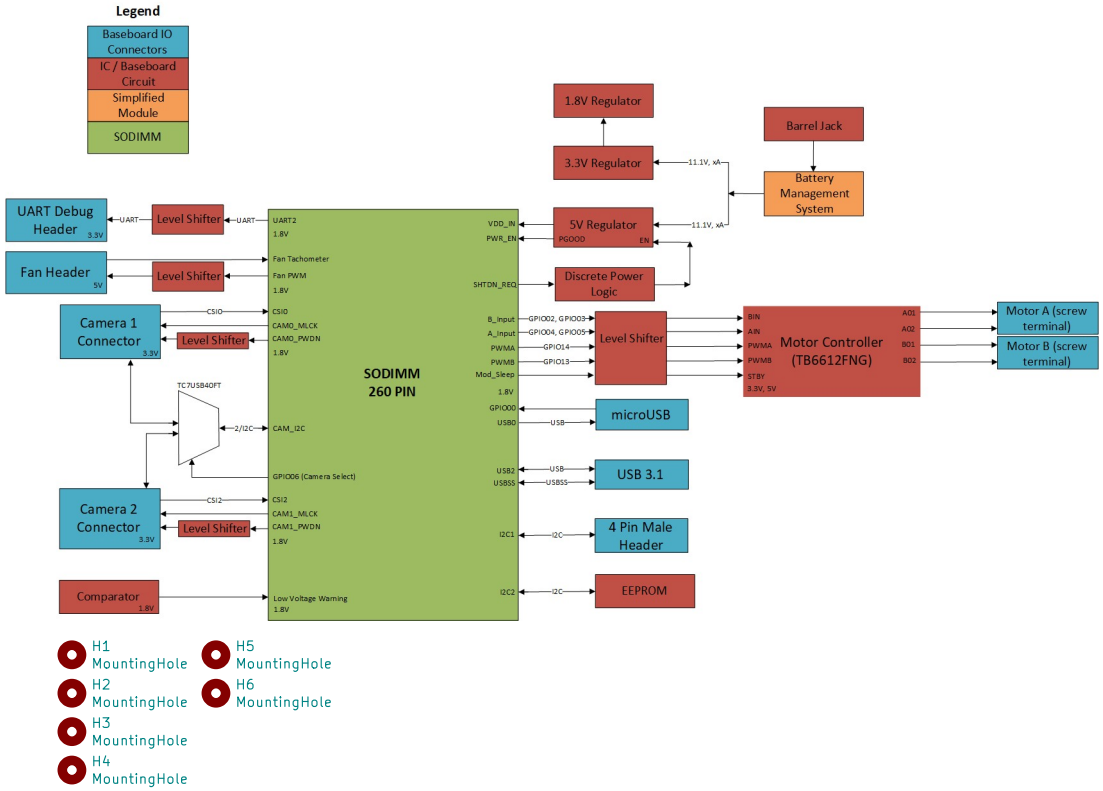


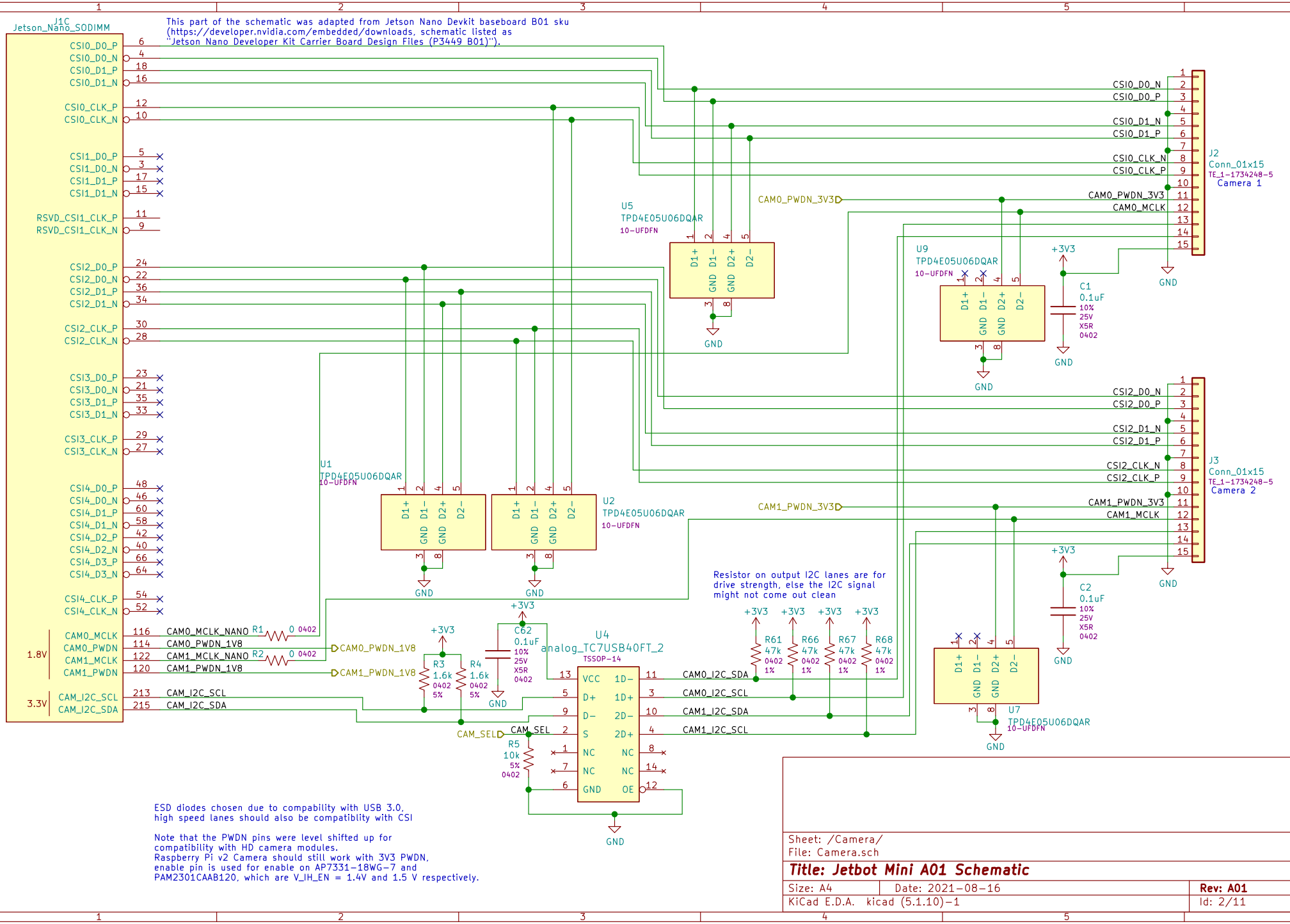


Please note that Power Input and Output nets to ICs and connectors are not explicitly labeled, as the power labels (5V, VDC, etc) have higher priority over the regular labels and thus show up on the layout over the regular labels.
PWR_FLAGS are used to denote power input or outputs to KiCAD.

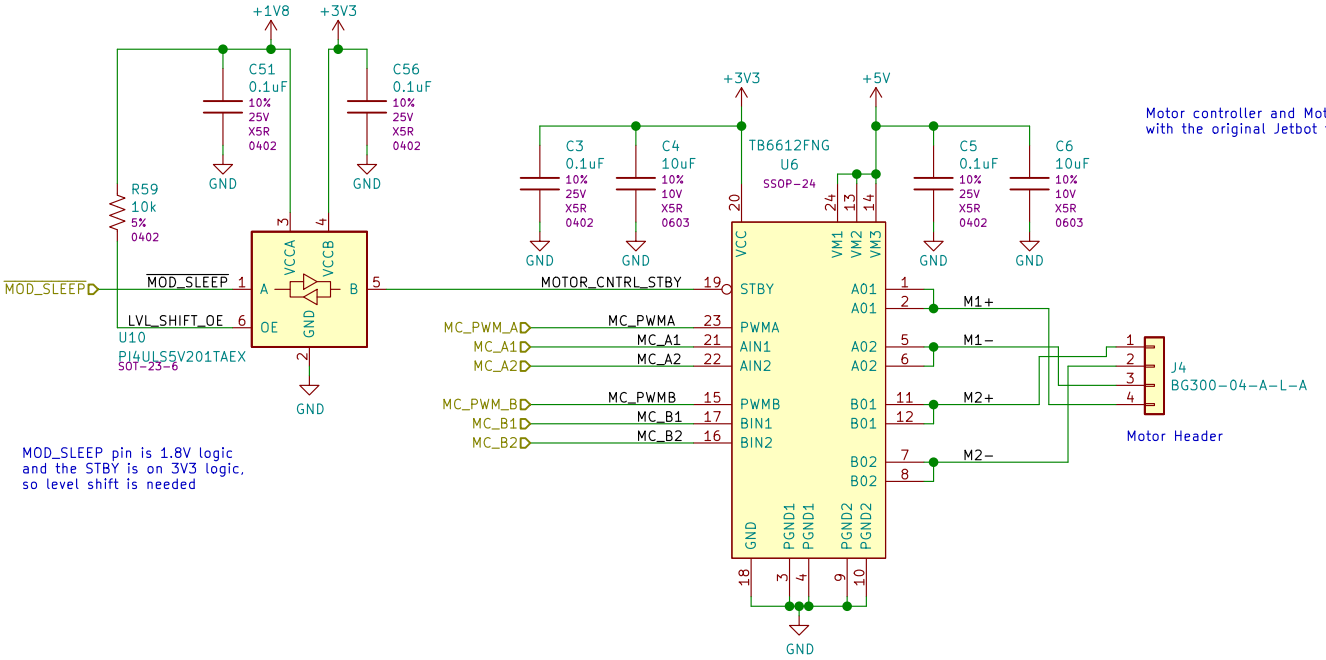
Power Sequence: Barrel jack or Battery --> Turns on 5V_AO --> 5V_AO powers Power Logic, which feeds high to 5V Buck Enable --> 5V PGOOD turns on Jetson Nano module --> SYS_RST turns on 3V3 buck --> 3V3 buck turns on 1V8 LDO

Circles on the pins denotes active low

Sheet: /		
File: Jetbot_Mini.sch		
Title: Jetbot Mini A01 Schematic		
Size: A4	Date: 2021-08-16	**Rev: A01**
KiCad E.D.A. kicad (5.1.10)-1		Id: 1/11


This part of the schematic was adapted from Jetson Nano Devkit baseboard B01 sku
(https://developer.nvidia.com/embedded/downloads, schematic listed as
"Jetson Nano Developer Kit Carrier Board Design Files (P3449 B01)").
J1C
Jetson_Nano_SODIMM
CSI0_D0_P
CSI0_D0_N
CSI0_D1_P
CSI0_D1_N
CSI0_CLK_P
CSI0_CLK_N
CSI1_D0_P
CSI1_D0_N
CSI1_D1_P
CSI1_D1_N
RSVD_CSI1_CLK_P
RSVD_CSI1_CLK_N
CSI2_D0_P
CSI2_D0_N
CSI2_D1_P
CSI2_D1_N
CSI2_CLK_P
CSI2_CLK_N
CSI3_D0_P
CSI3_D0_N
CSI3_D1_P
CSI3_D1_N
CSI3_CLK_P
CSI3_CLK_N
CSI4_D0_P
CSI4_D0_N
CSI4_D1_P
CSI4_D1_N
CSI4_D2_P
CSI4_D2_N
CSI4_D3_P
CSI4_D3_N
CSI4_CLK_P
CSI4_CLK_N
1.8V
CAM0_MCLK
CAM0_PWDN
CAM1_MCLK
CAM1_PWDN
3.3V
CAM_I2C_SCL
CAM_I2C_SDA
CAM0_MCLK_NANO
CAM0_PWDN_1V8
CAM1_MCLK_NANO
CAM1_PWDN_1V8
R1
R2
0 0402
U5
TPD4E05U06DQAR
10-UFDFN
U1
U2
U9
U7
D1+
D1-
D2+
D2-
GND
J2
Conn_01x15
TE_1-1734248-5
Camera 1
J3
Camera 2
C1
C2
0.1uF
10%
25V
X5R
0402
+3V3
CAM0_PWDN_3V3
CAM1_PWDN_3V3
CAM0_MCLK
CAM1_MCLK
U4
analog_TC7USB40FT_2
TSSOP-14
VCC
D+
D-
S
NC
OE
1D-
1D+
2D-
2D+
CAM0_I2C_SDA
CAM0_I2C_SCL
CAM1_I2C_SDA
CAM1_I2C_SCL
CAM_SEL
C62
R3
R4
1.6k
0402
5%
R5
10k
R61
R66
R67
R68
47k
1%
Resistor on output I2C lanes are for
drive strength, else the I2C signal
might not come out clean
ESD diodes chosen due to compability with USB 3.0,
high speed lanes should also be compatiblity with CSI
Note that the PWDN pins were level shifted up for
compatibility with HD camera modules.
Raspberry Pi v2 Camera should still work with 3V3 PWDN,
enable pin is used for enable on AP7331-18WG-7 and
PAM2301CAAB120, which are V_IH_EN = 1.4V and 1.5 V respectively.
Sheet: /Camera/
File: Camera.sch
Title: Jetbot Mini A01 Schematic
Size: A4
Date: 2021-08-16
Rev: A01
KiCad E.D.A. kicad (5.1.10)-1
Id: 2/11

This portion of the schematic was adapted from
the TB6612FNG datasheet



MOD_SLEEP pin is 1.8V logic
and the STBY is on 3V3 logic,
so level shift is needed

Motor controller and Motors chosen were kept consistent
with the original Jetbot to ensure software compatibility

Sheet: /Motors/
File: Motors.sch

Title: Jetbot Mini A01 Schematic

Size: A4	Date: 2021-08-16	**Rev: A01**
KiCad E.D.A. kicad (5.1.10)-1		Id: 3/11

This part of the schematic was adapted from Jetson Nano Devkit baseboard B01 sku (https://developer.nvidia.com/embedded/downloads, schematic listed as "Jetson Nano Developer Kit Carrier Board Design Files (P3449 B01)").

J1D
Jetson_Nano_SODIMM

GBE_LED_LINK 188
GBE_LED_ACT 194
GBE_MDI0_P 186
GBE_MDI0_N 184
GBE_MDI1_P 192
GBE_MDI1_N 190
GBE_MDI2_P 198
GBE_MDI2_N 196
GBE_MDI3_P 204
GBE_MDI3_N 202
SDMMC_DAT0 219
SDMMC_DAT1 221
SDMMC_DAT2 223
SDMMC_DAT3 225
SDMMC_CMD 227
SDMMC_CLK 229
USB0_D_P 111 — USB0+
USB0_D_N 109 — USB0−
USB1_D_P 117 — USB1+
USB1_D_N 115 — USB1−
USB2_D_P 123
USB2_D_N 121
USBSS_TX_P 168 — Nano_SS_TX_C+
USBSS_TX_N 166 — Nano_SS_TX_C−
USBSS_RX_P 163 — SS_RX−
USBSS_RX_N 161 — SS_RX+
RSVD_CAN_TX 145
RSVD_CAN_RX 143

VBUS_DET — VBUS_DET

MicroUSB

Q1
DMN26D0UT-7
SOT-523
Vgs = ±10V
Vds = 20V
Id = 0.23A
Vgs (th, max) = 1V

C10 0.1uF 10% 25V X5R 0402
R7 1M 1% 0402
D1 TPD1E10B06DPYR 0402 (1006 Metric)
uVBUS 1
J7 10103594-0001LF Micro-B
VBUS, D+, D−, ID, GND, Shield

U28
TPD4E05U06DQAR
10-UFDFN
D1+ D1− D2+ D2− GND GND

USB Type A

C7 0.1uF 10% 25V X5R 0402
C8 0.1uF 10% 25V X5R 0402

J8
692121130100
USB Type A
VBUS, D−, D+, SSRX−, SSRX+, SSTX−, SSTX+, GND, DRAIN, SHIELD

U20
TPD4E05U06DQAR
10-UFDFN

U25
TPD4E05U06DQAR
10-UFDFN

TPD4E05U06DQAR ESD diode pins can be the − or + data lines, datasheet says each pin is a "ESD protected channel".

R60 0 ohm resistor for pads just in case ferrite beads need to be stuffed during validation

+5V
U8
RT9742CGJ5
TSOT-23-5
VIN VOUT FLG EN GND
C9 10uF 10% 10V X5R 0603
R6 10k 5% 0402
USB_SWITCH_FLG
VBUS_EN
USB_PWR_EN
USB_SWITCH_OUTPUT
C12 0.1uF 10% 25V X5R 0402
R60 0 0402 1%
PWR_FLAG
VBUS
C11 120uF 20% 6.3 POSCAP 1411

Electrolytic recommended by datasheet to reduce ringing

Capacitor values recommended by Datasheet

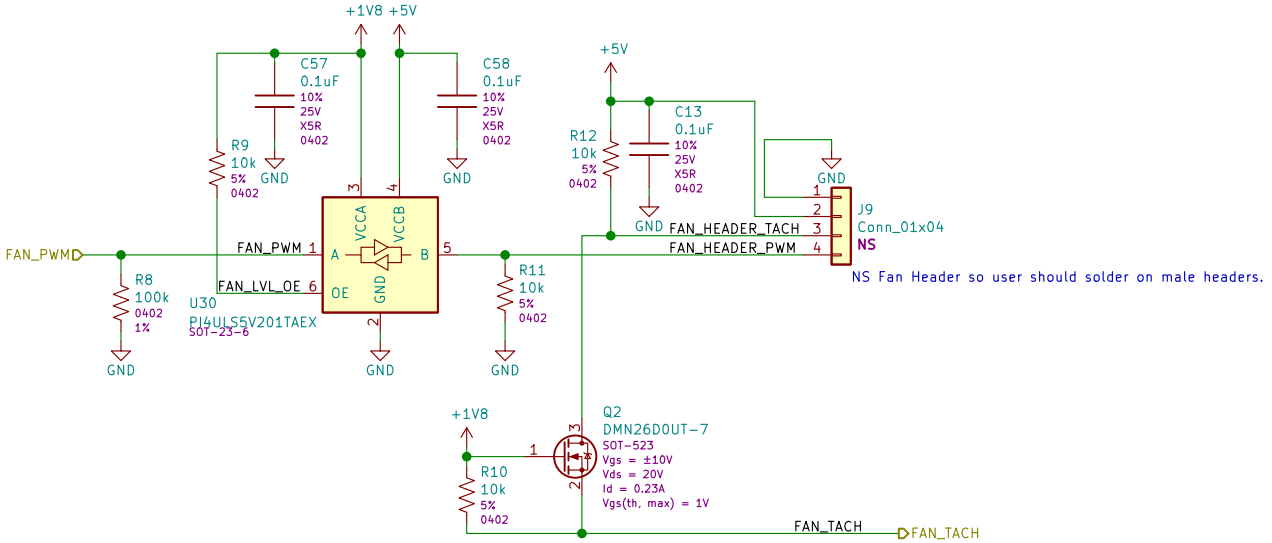
USB switch needed to protect the host, as the USB might be drawing too much current, and you can power cycle the USB.

Sheet: /USB/
File: USB.sch

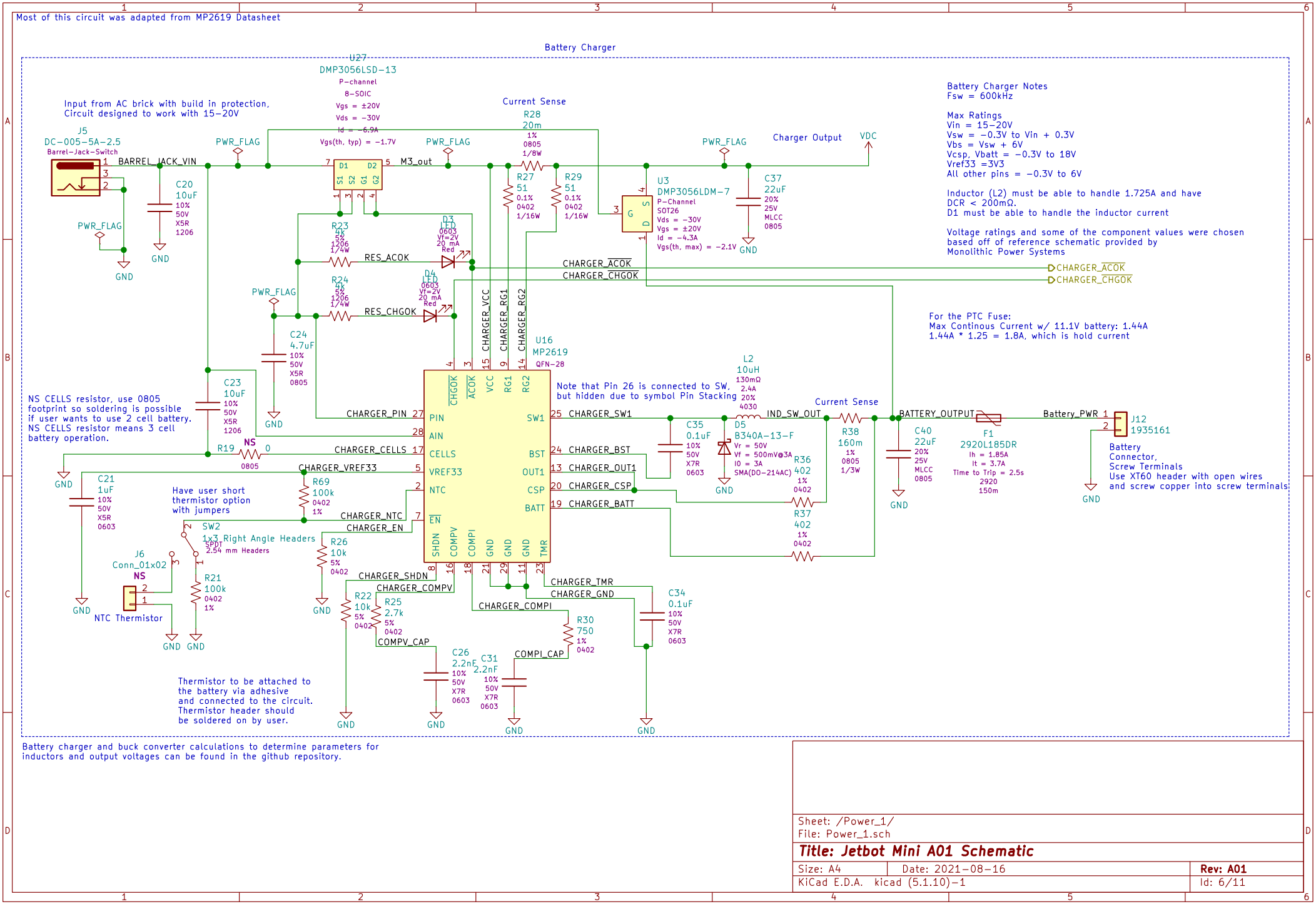
Title: Jetbot Mini A01 Schematic

Size: A4 | Date: 2021-08-16 | **Rev: A01**
KiCad E.D.A. kicad (5.1.10)-1 | Id: 4/11

This part of the schematic was adapted from Jetson Nano Devkit baseboard B01 sku
(https://developer.nvidia.com/embedded/downloads, schematic listed as
"Jetson Nano Developer Kit Carrier Board Design Files (P3449 B01)").



Sheet: /Fan/		
File: Fan.sch		
Title: Jetbot Mini A01 Schematic		
Size: A4	Date: 2021-08-16	**Rev: A01**
KiCad E.D.A. kicad (5.1.10)-1		Id: 5/11


Most of this circuit was adapted from MP2619 Datasheet
Battery Charger
U27
DMP3056LSD-13
P-channel
8-SOIC
Vgs = ±20V
Vds = -30V
Id = -6.9A
Vgs(th, typ) = -1.7V
Input from AC brick with build in protection,
Circuit designed to work with 15-20V
J5
DC-005-5A-2.5
Barrel-Jack-Switch
BARREL_JACK_VIN
PWR_FLAG
GND
C20
10uF
10%
50V
X5R
1206
Current Sense
R28
20m
1%
0805
1/8W
M3_out
R27
51
0.1%
0402
1/16W
R29
51
0.1%
0402
1/16W
U3
DMP3056LDM-7
P-Channel
SOT26
Vds = -30V
Vgs = ±20V
Id = -4.3A
Vgs(th, max) = -2.1V
Charger Output
VDC
C37
22uF
20%
25V
MLCC
0805
Battery Charger Notes
Fsw = 600kHz
Max Ratings
Vin = 15-20V
Vsw = -0.3V to Vin + 0.3V
Vbs = Vsw + 6V
Vcsp, Vbatt = -0.3V to 18V
Vref33 =3V3
All other pins = -0.3V to 6V
Inductor (L2) must be able to handle 1.725A and have
DCR < 200mΩ.
D1 must be able to handle the inductor current
Voltage ratings and some of the component values were chosen
based off of reference schematic provided by
Monolithic Power Systems
R23
4k
5%
1206
1/4W
RES_ACOK
D3
LED
0603
Vf=2V
20 mA
Red
CHARGER_ACOK
CHARGER_CHGOK
R24
4k
5%
1206
1/4W
RES_CHGOK
D4
LED
0603
Vf=2V
20 mA
Red
C24
4.7uF
10%
50V
X5R
0805
U16
MP2619
QFN-28
CHGOK
ACOK
VCC
RG1
RG2
PIN
AIN
CELLS
VREF33
NTC
EN
SHDN
COMPV
COMPI
GND
GND
GND
TMR
SW1
BST
OUT1
CSP
BATT
CHARGER_VCC
CHARGER_RG1
CHARGER_RG2
CHARGER_PIN
CHARGER_CELLS
CHARGER_VREF33
CHARGER_NTC
CHARGER_EN
CHARGER_SHDN
CHARGER_COMPV
CHARGER_COMPI
CHARGER_TMR
CHARGER_GND
CHARGER_SW1
CHARGER_BST
CHARGER_OUT1
CHARGER_CSP
CHARGER_BATT
Note that Pin 26 is connected to SW,
but hidden due to symbol Pin Stacking
L2
10uH
130mΩ
2.4A
20%
4030
IND_SW_OUT
C35
0.1uF
10%
50V
X7R
0603
D5
B340A-13-F
Vr = 50V
Vf = 500mV@3A
Io = 3A
SMA(DO-214AC)
R36
402
1%
0402
R37
402
1%
0402
Current Sense
R38
160m
1%
0805
1/3W
BATTERY_OUTPUT
C40
22uF
20%
25V
MLCC
0805
F1
2920L185DR
Ih = 1.85A
It = 3.7A
Time to Trip = 2.5s
2920
150m
Battery_PWR
J12
1935161
Battery
Connector,
Screw Terminals
Use XT60 header with open wires
and screw copper into screw terminals
For the PTC Fuse:
Max Continous Current w/ 11.1V battery: 1.44A
1.44A * 1.25 = 1.8A, which is hold current
NS CELLS resistor, use 0805
footprint so soldering is possible
if user wants to use 2 cell battery.
NS CELLS resistor means 3 cell
battery operation.
R19
NS
0
0805
C23
10uF
10%
50V
X5R
1206
C21
1uF
10%
50V
X5R
0603
Have user short
thermistor option
with jumpers
R69
100k
0402
1%
SW2
1x3 Right Angle Headers
SPDT
2.54 mm Headers
J6
Conn_01x02
NS
NTC Thermistor
R21
100k
0402
1%
R26
10k
5%
0402
R22
10k
5%
0402
R25
2.7k
5%
0402
COMPV_CAP
C26
2.2nF
10%
50V
X7R
0603
C31
2.2nF
10%
50V
X7R
0603
COMPI_CAP
R30
750
1%
0402
C34
0.1uF
10%
50V
X7R
0603
Thermistor to be attached to
the battery via adhesive
and connected to the circuit.
Thermistor header should
be soldered on by user.
Battery charger and buck converter calculations to determine parameters for
inductors and output voltages can be found in the github repository.
Sheet: /Power_1/
File: Power_1.sch
Title: Jetbot Mini A01 Schematic
Size: A4
Date: 2021-08-16
Rev: A01
KiCad E.D.A. kicad (5.1.10)-1
Id: 6/11

This part of the schematic was adapted from Jetson Nano Devkit baseboard B01 sku (https://developer.nvidia.com/embedded/downloads, schematic listed as "Jetson Nano Developer Kit Carrier Board Design Files (P3449 B01)").

PWR_FLAG tells KiCAD that the net is a power input. Used for DRC purposes.

Designed for use with MakerFocus I2C OLED Display, which has an onboard LDO that accepts Vin = 1 – 6 V and Vout = 3.3 V. Use 5V as 3V3 would yield a small voltage drop in the output.

Use I2C1_SCL and I2C1_SDA for compatibility with Jetbot SD Card Image

Use two 0 ohm resistors, one pulled up to 5V and one pulled up to 3V3, NS one of them, to be able to choose what voltage rail to connect the OLED display to.

Note that J13 is NS as user should solder on male headers.

MOD_SLEEP pin controls standby on motor controller, with VCC = 3.3V, V_IH (min) for standby pin is 2.31V, therefore needs level shifter.

SYS_RST controls 3V3 buck enable and USB power switch enable, which have V_IH (typ) = 2.5V and V_IH (min) = 1.5V respectively. AP6503 has V_EN ratings from −0.3 to 6V, and RT9742 has V_EN ratings from −0.3V to 7V

Cannot use regular level shifter as 3V3 rail will not be on without SYS_RST enable signal

Recommended operating conditions for 74LVC4245A state VCCA > VCCB, hence the flipped orientation.

MOSFET with large Vgs tolerance needed as input voltage from barrel jack is being fed into these pins.

OLED

EEPROM

2.2 kΩ pull-up to 1.8V on module for I2C_SCL and I2C_SDA lanes

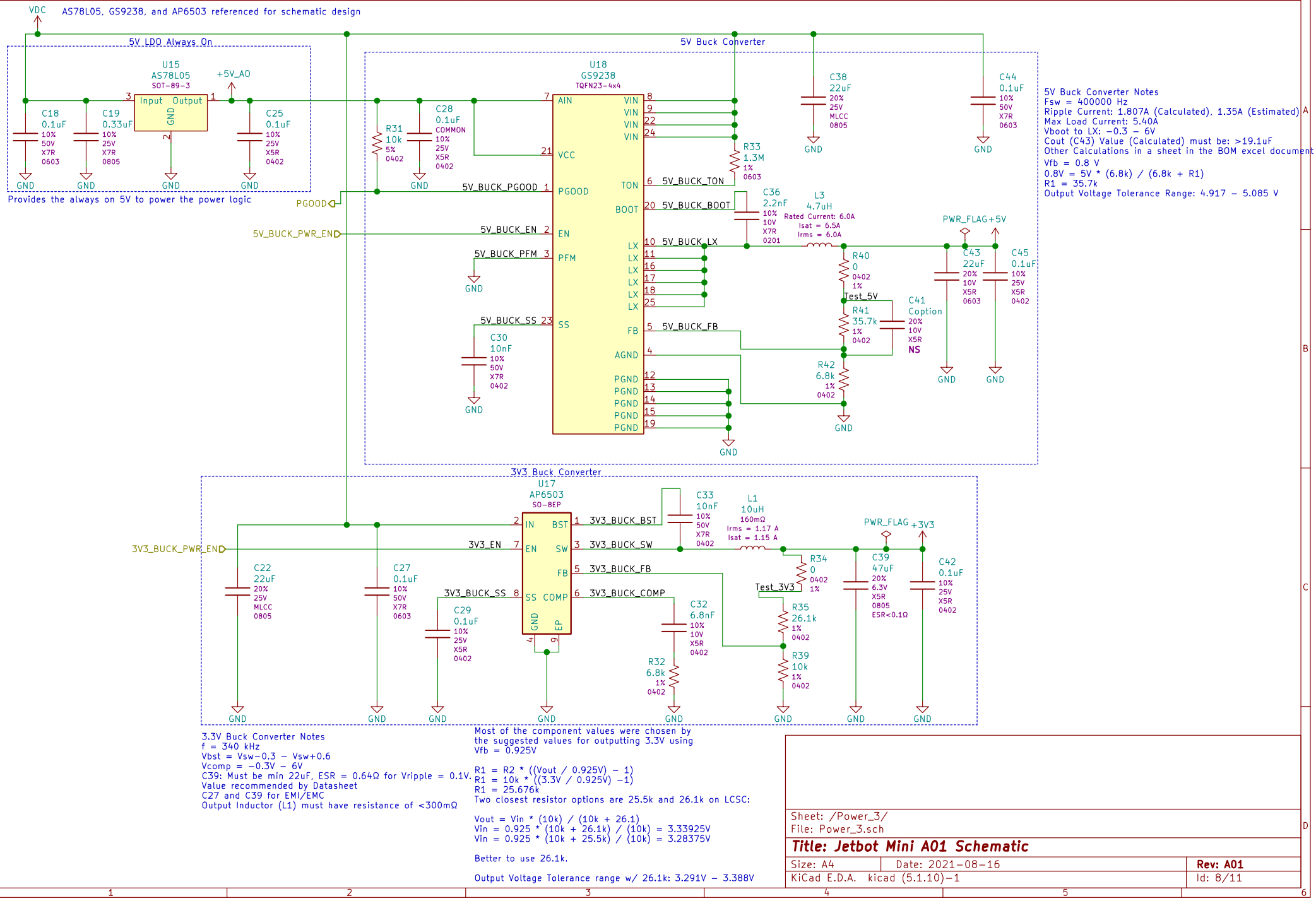
EEPROM is to store information about the baseboard, for example if remoting into the board you can check the EEPROM and determine information about the board. For example, A00 vs A01 iteration might have differences that software needs to know about, like GPIO defn, etc.

UART Debug

1x6 0.1" Male Pins
Debug Header
Designed to work with FTDI Friend
Connector NS so user can solder on male headers

R50 to hold Enable low until VCCA and VCCB are ready

Sheet: /Nano_IO/	
File: Nano_IO.sch	
Title: Jetbot Mini A01 Schematic	
Size: A4	Date: 2021-08-16
KiCad E.D.A. kicad (5.1.10)-1	Rev: A01
	Id: 7/11


AS78L05, GS9238, and AP6503 referenced for schematic design
VDC
5V LDO Always On
U15
AS78L05
SOT-89-3
Input
Output
GND
+5V_AO
C18
0.1uF
10%
50V
X7R
0603
C19
0.33uF
10%
25V
X7R
0805
C25
0.1uF
10%
25V
X5R
0402
Provides the always on 5V to power the power logic
5V Buck Converter
U18
GS9238
TQFN23-4x4
AIN
VCC
PGOOD
EN
PFM
SS
FB
AGND
PGND
VIN
TON
BOOT
LX
R31
10k
5%
0402
C28
0.1uF
COMMON
10%
25V
X5R
0402
R33
1.3M
1%
0603
C38
22uF
20%
25V
MLCC
0805
C44
0.1uF
10%
50V
X7R
0603
5V_BUCK_PGOOD
PGOOD
5V_BUCK_EN
5V_BUCK_PWR_EN
5V_BUCK_PFM
5V_BUCK_SS
5V_BUCK_TON
5V_BUCK_BOOT
5V_BUCK_LX
5V_BUCK_FB
C30
10nF
10%
50V
X7R
0402
C36
2.2nF
10%
10V
X7R
0201
L3
4.7uH
Rated Current: 6.0A
Isat = 6.5A
Irms = 6.0A
R40
0
0402
1%
Test_5V
R41
35.7k
1%
0402
C41
Coption
20%
10V
X5R
NS
R42
6.8k
1%
0402
PWR_FLAG+5V
C43
22uF
20%
10V
X5R
0603
C45
0.1uF
10%
25V
X5R
0402
5V Buck Converter Notes
Fsw = 400000 Hz
Ripple Current: 1.807A (Calculated), 1.35A (Estimated)
Max Load Current: 5.40A
Vboot to LX: −0.3 – 6V
Cout (C43) Value (Calculated) must be: >19.1uF
Other Calculations in a sheet in the BOM excel document
Vfb = 0.8 V
0.8V = 5V * (6.8k) / (6.8k + R1)
R1 = 35.7k
Output Voltage Tolerance Range: 4.917 – 5.085 V
3V3 Buck Converter
U17
AP6503
SO-8EP
IN
BST
EN
SW
FB
SS
COMP
GND
EP
3V3_BUCK_PWR_EN
3V3_EN
3V3_BUCK_BST
3V3_BUCK_SW
3V3_BUCK_FB
3V3_BUCK_SS
3V3_BUCK_COMP
C22
22uF
20%
25V
MLCC
0805
C27
0.1uF
10%
50V
X7R
0603
C29
0.1uF
10%
25V
X5R
0402
C33
10nF
10%
50V
X7R
0402
L1
10uH
160mΩ
Irms = 1.17 A
Isat = 1.15 A
R34
0
0402
1%
Test_3V3
R35
26.1k
1%
0402
R39
10k
1%
0402
R32
6.8k
1%
0402
C32
6.8nF
10%
10V
X5R
0402
C39
47uF
20%
6.3V
X5R
0805
ESR<0.1Ω
C42
0.1uF
10%
25V
X5R
0402
PWR_FLAG
+3V3
3.3V Buck Converter Notes
f = 340 kHz
Vbst = Vsw−0.3 – Vsw+0.6
Vcomp = −0.3V – 6V
C39: Must be min 22uF, ESR = 0.64Ω for Vripple = 0.1V.
Value recommended by Datasheet
C27 and C39 for EMI/EMC
Output Inductor (L1) must have resistance of <300mΩ
Most of the component values were chosen by
the suggested values for outputting 3.3V using
Vfb = 0.925V
R1 = R2 * ((Vout / 0.925V) – 1)
R1 = 10k * ((3.3V / 0.925V) −1)
R1 = 25.676k
Two closest resistor options are 25.5k and 26.1k on LCSC:
Vout = Vin * (10k) / (10k + 26.1)
Vin = 0.925 * (10k + 26.1k) / (10k) = 3.33925V
Vin = 0.925 * (10k + 25.5k) / (10k) = 3.28375V
Better to use 26.1k.
Output Voltage Tolerance range w/ 26.1k: 3.291V – 3.388V
Sheet: /Power_3/
File: Power_3.sch
Title: Jetbot Mini A01 Schematic
Size: A4
Date: 2021-08-16
Rev: A01
KiCad E.D.A. kicad (5.1.10)-1
Id: 8/11

Capacitors on the Nano module's 5V rail were adapted from Jetson Nano Devkit baseboard B01 sku (https://developer.nvidia.com/embedded/downloads, schematic listed as "Jetson Nano Developer Kit Carrier Board Design Files (P3449 B01)").

J1G Jetson_Nano_SODIMM

GND: 1, 2, 7, 8, 13, 14, 19, 20, 25, 26, 31, 32, 37, 38, 43, 44, 49, 50, 55, 56, 61, 62, 67, 68, 73, 74, 79, 80, 85, 86, 102, 107, 113, 119, 125, 129, 132, 135, 138, 141, 144, 146, 147, 152, 153, 158, 159, 164, 165, 170, 171, 176, 177, 200, 201, 217, 231

+5V

C46 330uF 6.3V POSCAP (Tant Poly) 25mΩ 2917 (7343 Metric) 6TPE330MAP

C47 100uF 20% 6.3V X5R 1206

C48 100uF 20% 6.3V X5R 1206

J1H Jetson_Nano_SODIMM

VDD: 251, 252, 253, 254, 255, 256, 257, 258, 259, 260

GND: 241, 242, 243, 244, 245, 246, 247, 248, 249, 250

POSCAPs used to reduce footprint size and save space

5V − 2V = 3V
R = 3V / 20mA = 150Ω

POWER_LED

+5V

R47 500 0402 1%

LED_CAT_PWR

D6 LED 0603 Vf=2V 20 mA Green

FET_SRC_PWR

+1V8

Q8 DMG1012T SOT523 Vgs = ±6V Vds = 20V Id = 0.63 A Vgs(th, max) = 1V

GND

LOW_VOLTAGE_WARNING LED

1.8 LDO

+3V3 +3V3

R51 10k 5% 0402

U24 TLV70218DBVR_SOT23-5 SOT-23-5

1V8_REG_EN

IN OUT EN GND

+1V8

C49 0.1uF 10% 25V X5R 0402

C50 1uF 10% 6.3V X5R 0603

C52 0.1uF 10% 25V X5R 0402

GND

VDC

+5V_AO

R46 100k 0402 1%

R63 4.12M 1% 0402

R44 47k 0402 1%

R48 40.2k 0402 1%

R45 6.8k 1% 0402

+5V

COMPARATOR_V+

COMPARATOR_V−

U19 GS331 SOT23-5 Open-Collector

R43 4.7k 5% 0402

COMPARATOR_OUT

D7 LED 0603 Vf=2V 20 mA Red

LED_CATHODE_LV

R58 500 0402 5%

FET_SOURCE_LV

Q5 DMG1012T SOT523 Vgs = ±6V Vds = 20V Id = 0.63 A Vgs(th, max) = 1V

C60 0.1uF 10% 25V X5R 0402

C61 0.1uF 10% 25V X5R 0402

Level shifter required to translate 5V to 1V8 to be compatible with tegra GPIO. Note that OE is enabled to 1V8, as level shifter should only be on when 5V is on.

U13 PI4ULS5V201TAEX SOT-23-6

VCCB VCCA GND A B OE

R72 10k 5% 0402

LOW_VOLTAGE_WARNING

LOWV_LVL_OE

Designed so "low battery" means approximately 20% of charge left. Choice of 20% charge due to comparison with phone case, where around 20% battery is when the phone prompts user to charge. Used the following website: https://blog.ampow.com/lipo-voltage-chart/ as a reference for voltage vs SOC curve for LiPo batteries. Note that 11.18V corresponds with 20% charge, but due to potentially different LiPo batteries having different curves, a more conservative estimate of 11.24V or 25% charge was used for the lower voltage for the hysteresis. The voltage high threshold was chosen to be around 50% charge, or 11.51V, to bring the value of the hysteresis resistor (in this case R63) to a "reasonable" value.

Thus the operation of the hysteresis should proceed as follows: R44 and R45 resistor divider brings the voltage down so that it can be compared on a 5V scale. Initially "COMPARATOR_V−" net should be greater than "COMPARATOR_V+" net, so output is pulled low and "COMPARATOR_V+" net should be at the lower threshold voltage, in this case about 1.42V (which corresponds to 11.24V via the R44 and R45 divider). When "COMPARATOR_V−" net dips below this lower threshold voltage, output of the comparator is pulled high via R43 turning on the LED and "COMPARATOR_V+" is set to the higher threshold voltage, in this case about 1.45V (which corresponds to 11.51V via R44 and R45 divider). Thus the battery voltage on the "COMPARATOR_V−" must rise above 11.51V for the low power LED to turn off.

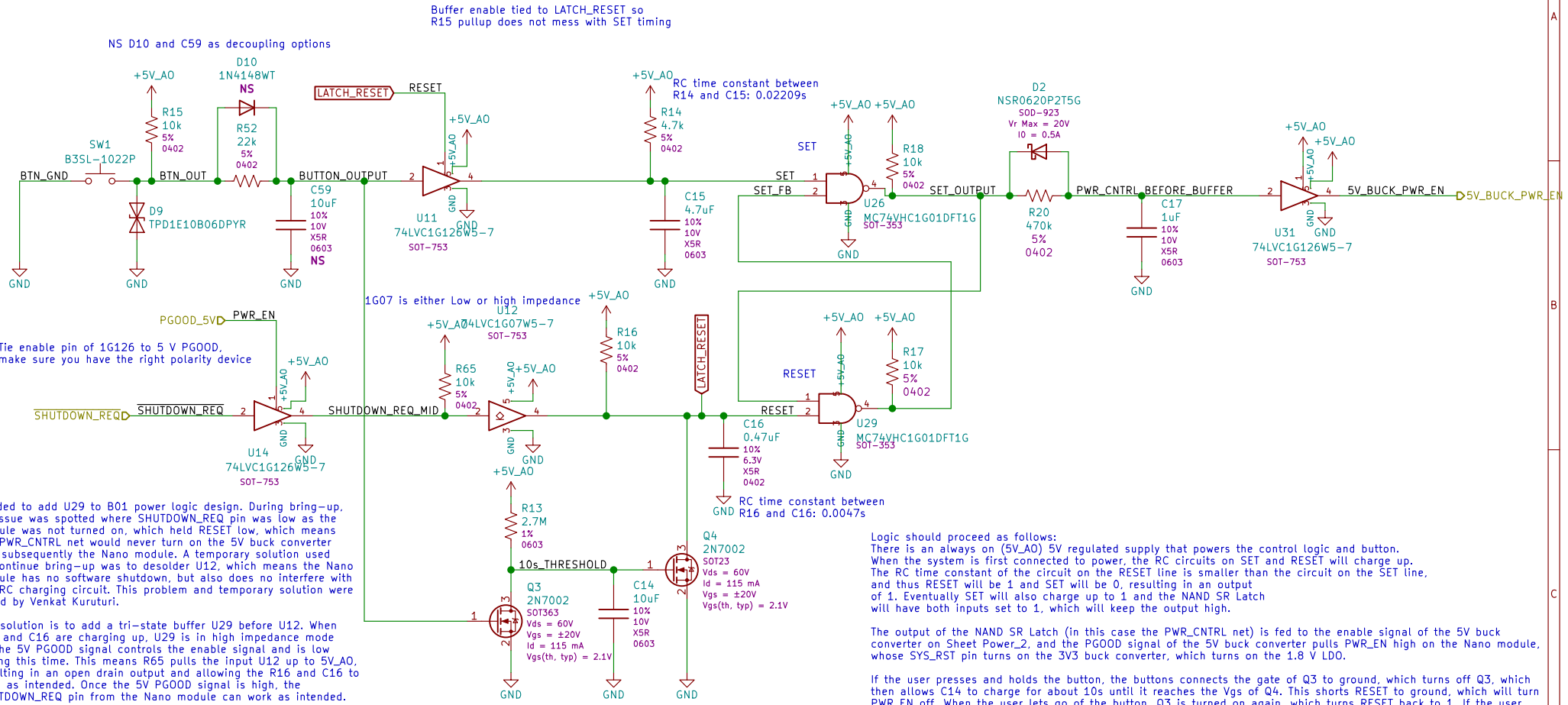
PLEASE NOTE that this hysteresis circuit is WIP, as R63 is a huge resistor value. Also note that Rh, in this case R63, should be at least 100 * R_pullup, in this case R43, to avoid R43 messing with the hysteresis threshold voltages. Source: https://www.ti.com/lit/ug/tidu020a/tidu020a.pdf?ts=1623787384403&ref_url=https%253A%252F%252Fwww.google.com%252F

Sheet: /Power_2/
File: Power_2.sch

Title: Jetbot Mini A01 Schematic

Size: A4	Date: 2021-08-16	Rev: A01
KiCad E.D.A. kicad (5.1.10)-1		Id: 9/11

This part of the schematic was adapted from Jetson Nano Devkit baseboard B01 sku
(https://developer.nvidia.com/embedded/downloads, schematic listed as
"Jetson Nano Developer Kit Carrier Board Design Files (P3449 B01)").



Needed to add U29 to B01 power logic design. During bring-up, an issue was spotted where SHUTDOWN_REQ pin was low as the module was not turned on, which held RESET low, which means the PWR_CNTRL net would never turn on the 5V buck converter and subsequently the Nano module. A temporary solution used to continue bring-up was to desolder U12, which means the Nano module has no software shutdown, but also does no interfere with the RC charging circuit. This problem and temporary solution were found by Venkat Kuruturi.

The solution is to add a tri-state buffer U29 before U12. When R16 and C16 are charging up, U29 is in high impedance mode as the 5V PGOOD signal controls the enable signal and is low during this time. This means R65 pulls the input U12 up to 5V_AO, resulting in an open drain output and allowing the R16 and C16 to work as intended. Once the 5V PGOOD signal is high, the SHUTDOWN_REQ pin from the Nano module can work as intended.

NAND-Gate SR Latch

Set	Reset	Output
1	1	No Change
0	1	Q=1
1	0	Q=0
0	0	Invalid State

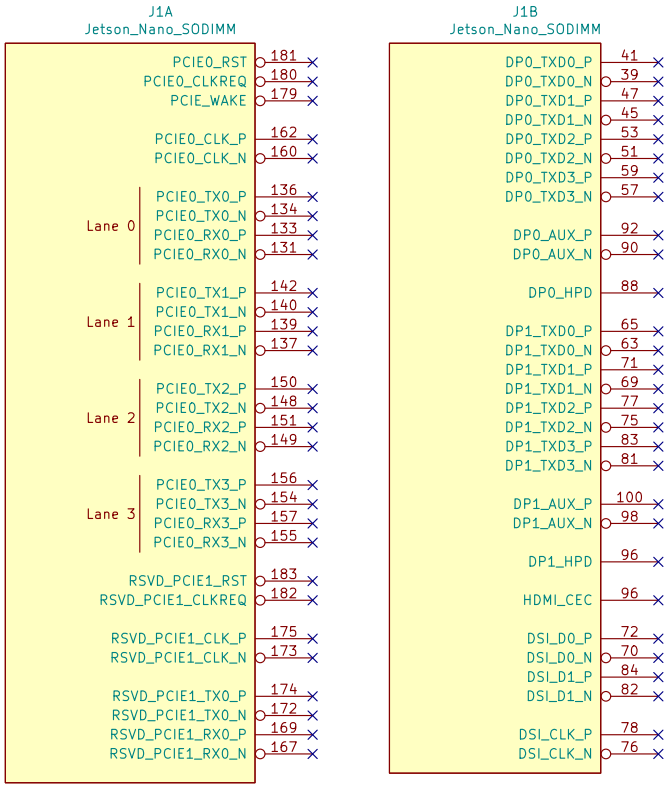
Logic should proceed as follows:
There is an always on (5V_AO) 5V regulated supply that powers the control logic and button.
When the system is first connected to power, the RC circuits on SET and RESET will charge up.
The RC time constant of the circuit on the RESET line is smaller than the circuit on the SET line,
and thus RESET will be 1 and SET will be 0, resulting in an output
of 1. Eventually SET will also charge up to 1 and the NAND SR Latch
will have both inputs set to 1, which will keep the output high.

The output of the NAND SR Latch (in this case the PWR_CNTRL net) is fed to the enable signal of the 5V buck converter on Sheet Power_2, and the PGOOD signal of the 5V buck converter pulls PWR_EN high on the Nano module, whose SYS_RST pin turns on the 3V3 buck converter, which turns on the 1.8 V LDO.

If the user presses and holds the button, the buttons connects the gate of Q3 to ground, which turns off Q3, which then allows C14 to charge for about 10s until it reaches the Vgs of Q4. This shorts RESET to ground, which will turn PWR_EN off. When the user lets go of the button, Q3 is turned on again, which turns RESET back to 1. If the user wants to turn the system back on, the user must press the button once, which will short SET to 0, which turns the system back on.

Note that the circuit should auto power on the Jetson Nano module and that SET and RESET are both active low.

Sheet: /Power_Logic/		
File: Power_Logic.sch		
Title: Jetbot Mini A01 Schematic		
Size: A4	Date: 2021-08-16	**Rev: A01**
KiCad E.D.A. kicad (5.1.10)-1		Id: 10/11



Sheet: /unused_IO/
File: unused_IO.sch

Title:

Size: A4	Date: 2021-08-16	**Rev:**
KiCad E.D.A. kicad (5.1.10)-1		Id: 11/11