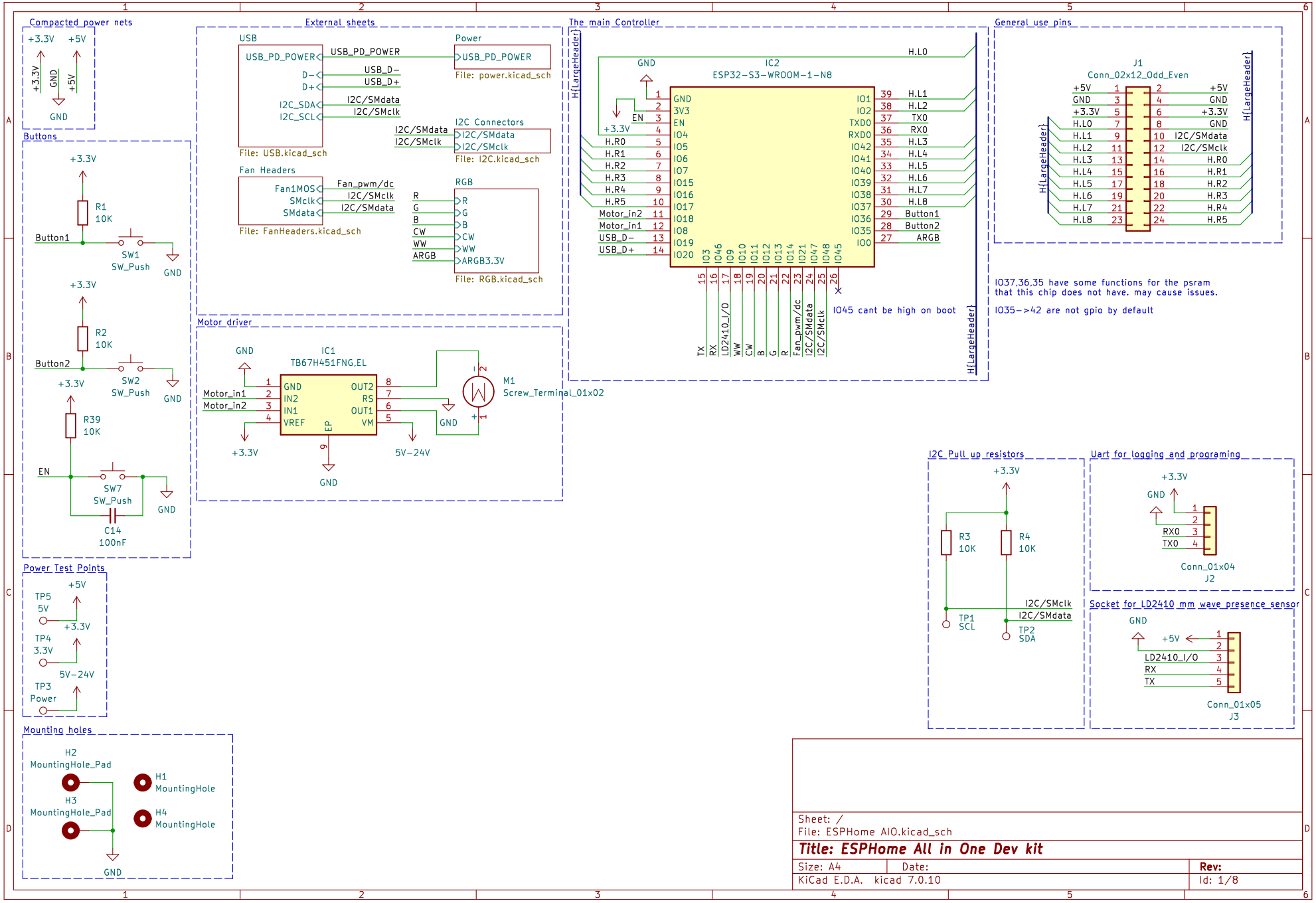
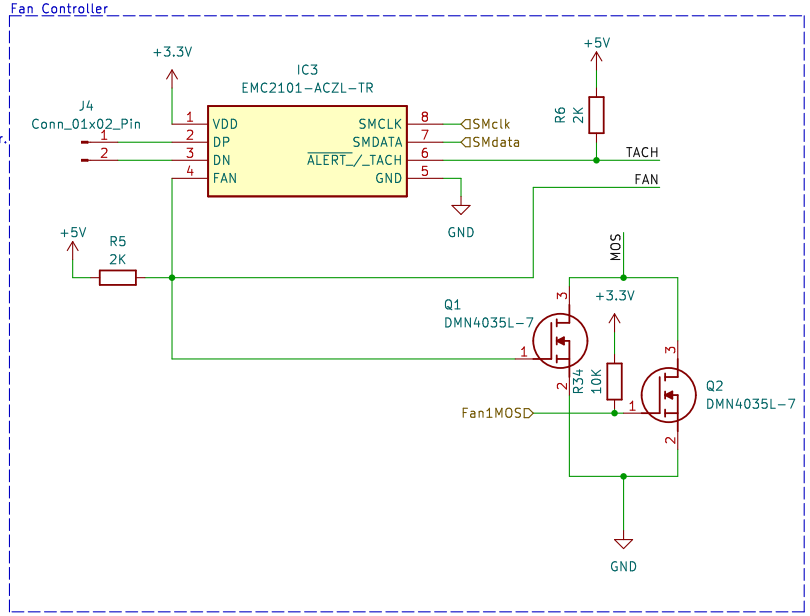

Compacted power nets
+3.3V
+5V
GND
Buttons
R1
10K
Button1
SW1
SW_Push
R2
10K
Button2
SW2
SW_Push
R39
10K
EN
SW7
SW_Push
C14
100nF
Power Test Points
TP5
5V
TP4
3.3V
TP3
Power
5V–24V
Mounting holes
H2
MountingHole_Pad
H3
MountingHole_Pad
H1
MountingHole
H4
MountingHole
External sheets
USB
USB_PD_POWER
D–
D+
I2C_SDA
I2C_SCL
File: USB.kicad_sch
Power
File: power.kicad_sch
I2C Connectors
I2C/SMdata
I2C/SMclk
File: I2C.kicad_sch
Fan Headers
Fan1MOS
SMclk
SMdata
Fan_pwm/dc
File: FanHeaders.kicad_sch
RGB
R
G
B
CW
WW
ARGB
ARGB3.3V
File: RGB.kicad_sch
Motor driver
IC1
TB67H451FNG,EL
Motor_in1
Motor_in2
M1
Screw_Terminal_01x02
5V–24V
The main Controller
IC2
ESP32-S3-WROOM-1-N8
H[LargeHeader]
Motor_in2
Motor_in1
USB_D–
USB_D+
IO45 cant be high on boot
TX
RX
LD2410_I/O
WW
CW
B
G
R
Fan_pwm/dc
I2C/SMdata
I2C/SMclk
General use pins
J1
Conn_02x12_Odd_Even
IO37,36,35 have some functions for the psram
that this chip does not have. may cause issues.
IO35–>42 are not gpio by default
I2C Pull up resistors
R3
10K
R4
10K
TP1
SCL
TP2
SDA
Uart for logging and programing
RX0
TX0
Conn_01x04
J2
Socket for LD2410 mm wave presence sensor
LD2410_I/O
RX
TX
Conn_01x05
J3
Sheet: /
File: ESPHome AIO.kicad_sch
Title: ESPHome All in One Dev kit
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 7.0.10
Id: 1/8

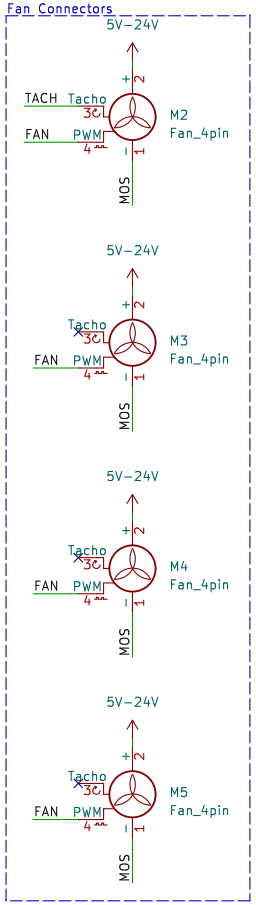
Connector for external temp sensor transistor.
refrence design uses a MMBT3904.



Only fan header M2 has the Tach connected.
The fan controller will adjust pwm only depending on the speed of M2.
So make sure to use M2 first.
Also i suggest not mixing fans
if you are using the power pin to adjust speed.

Turn off fan1MOS for 3Pin/DC Mode

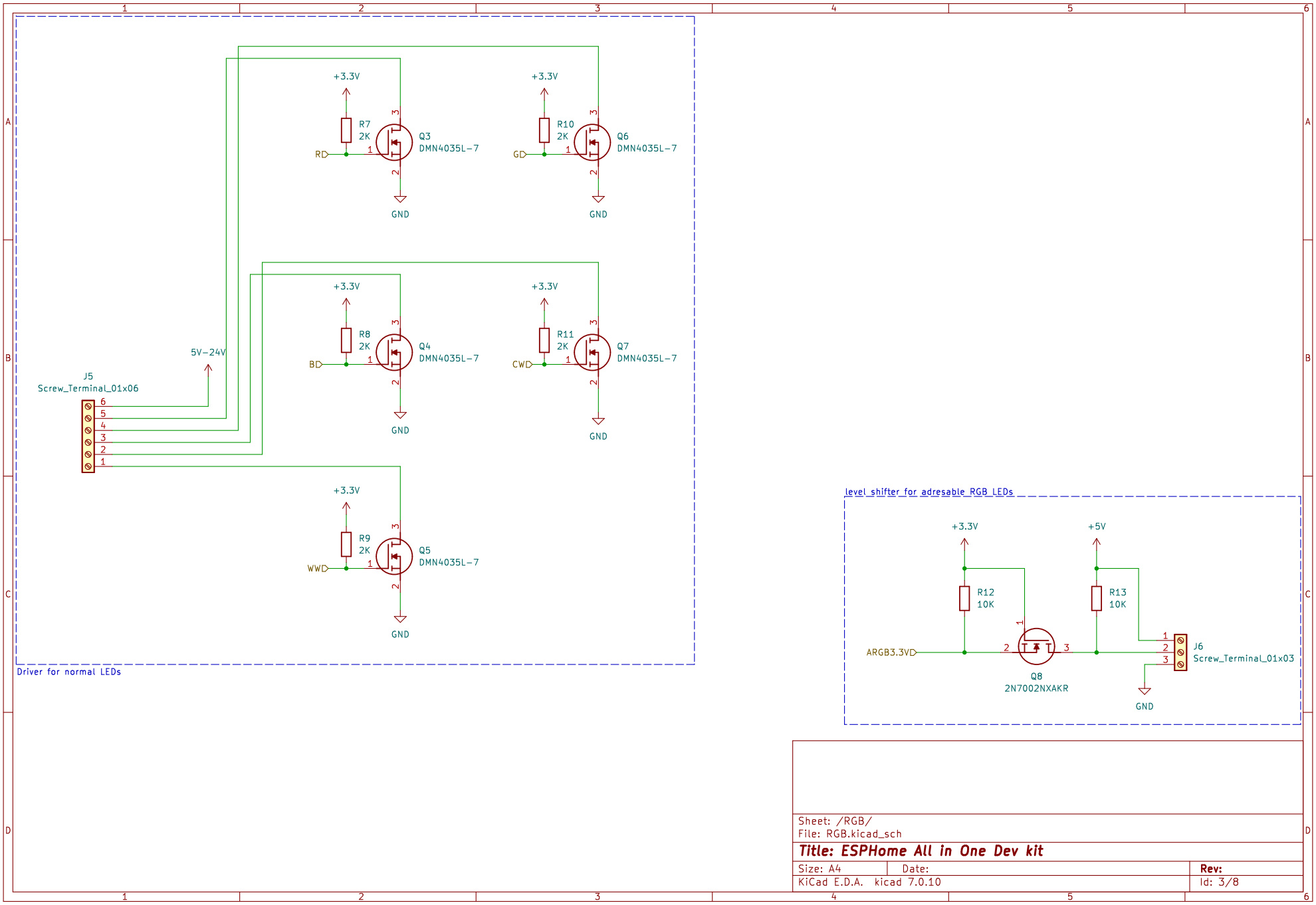
At 5V using 4 fans might be too much current for the mosfet.
At 12v you could connect way more fans.
Keep an eye on the temprature of the mosfets.



Sheet: /Fan Headers/
File: FanHeaders.kicad_sch

Title: ESPHome All in One Dev kit

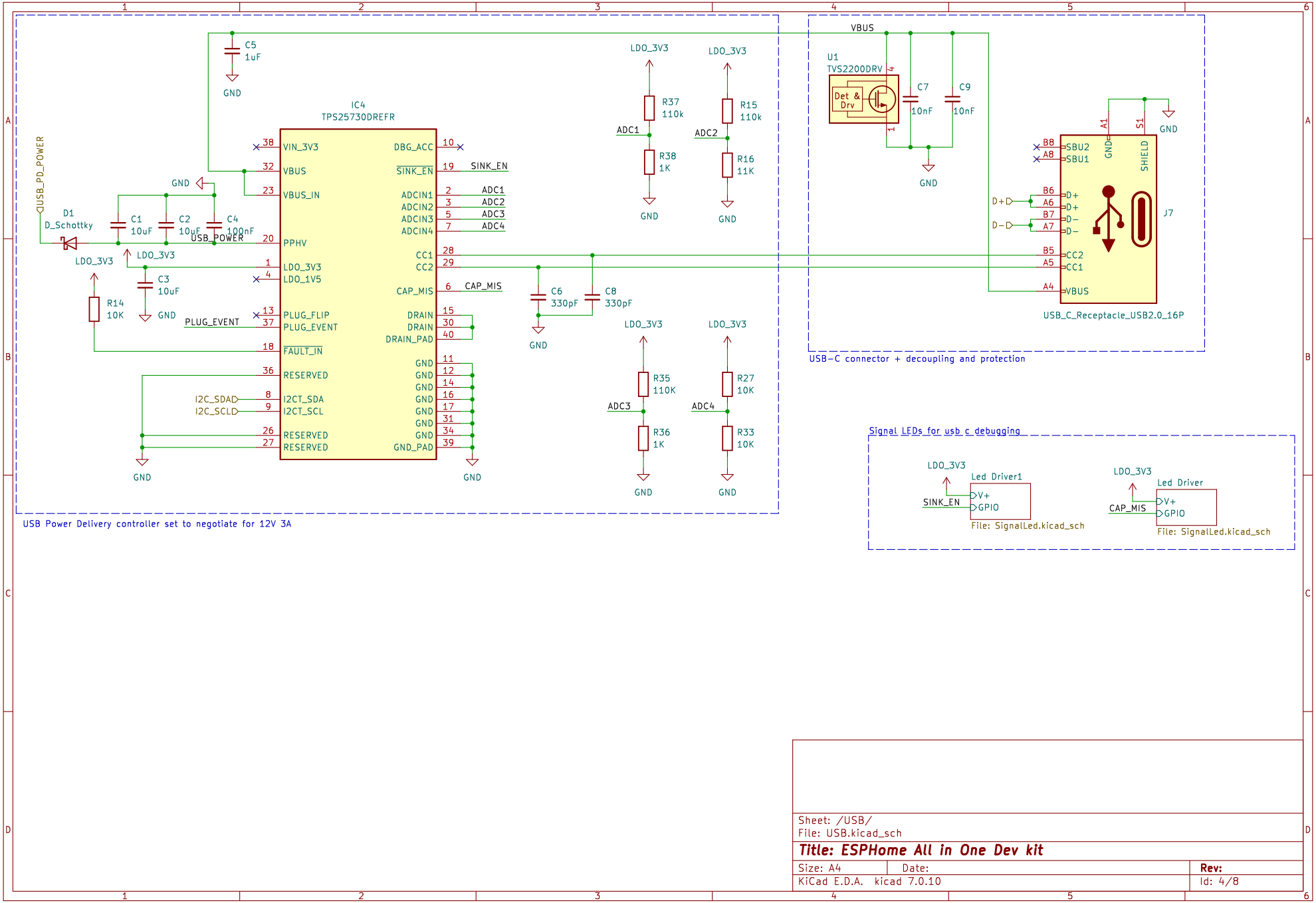
Size: A4	Date:	Rev:
KiCad E.D.A. kicad 7.0.10		Id: 2/8



Sheet: /RGB/
File: RGB.kicad_sch

Title: ESPHome All in One Dev kit

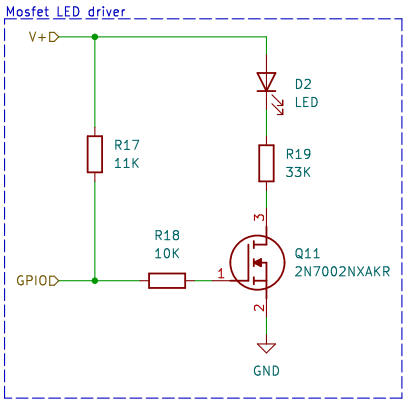
Size: A4	Date:	Rev:
KiCad E.D.A. kicad 7.0.10		Id: 3/8



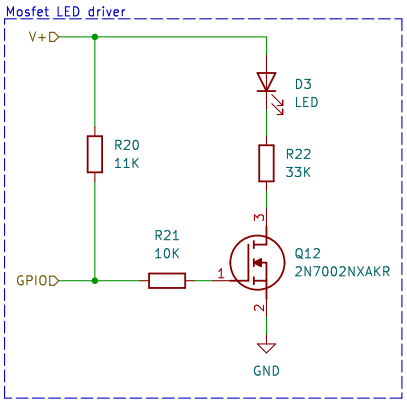
Sheet: /USB/
File: USB.kicad_sch

Title: ESPHome All in One Dev kit

Size: A4	Date:	Rev:
KiCad E.D.A. kicad 7.0.10		Id: 4/8



Sheet: /USB/Led Driver/		
File: SignalLed.kicad_sch		
Title: ESPHome All in One Dev kit		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 7.0.10		Id: 5/8

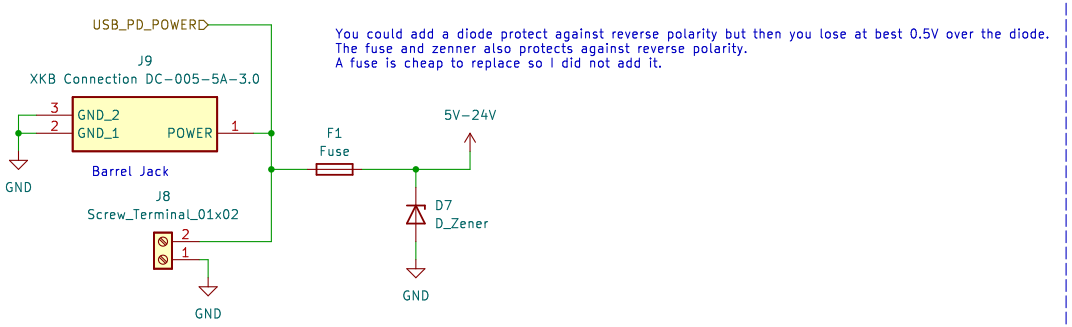


Sheet: /USB/Led Driver1/
File: SignalLed.kicad_sch

Title: ESPHome All in One Dev kit

Size: A4	Date:	Rev:
KiCad E.D.A. kicad 7.0.10		Id: 6/8

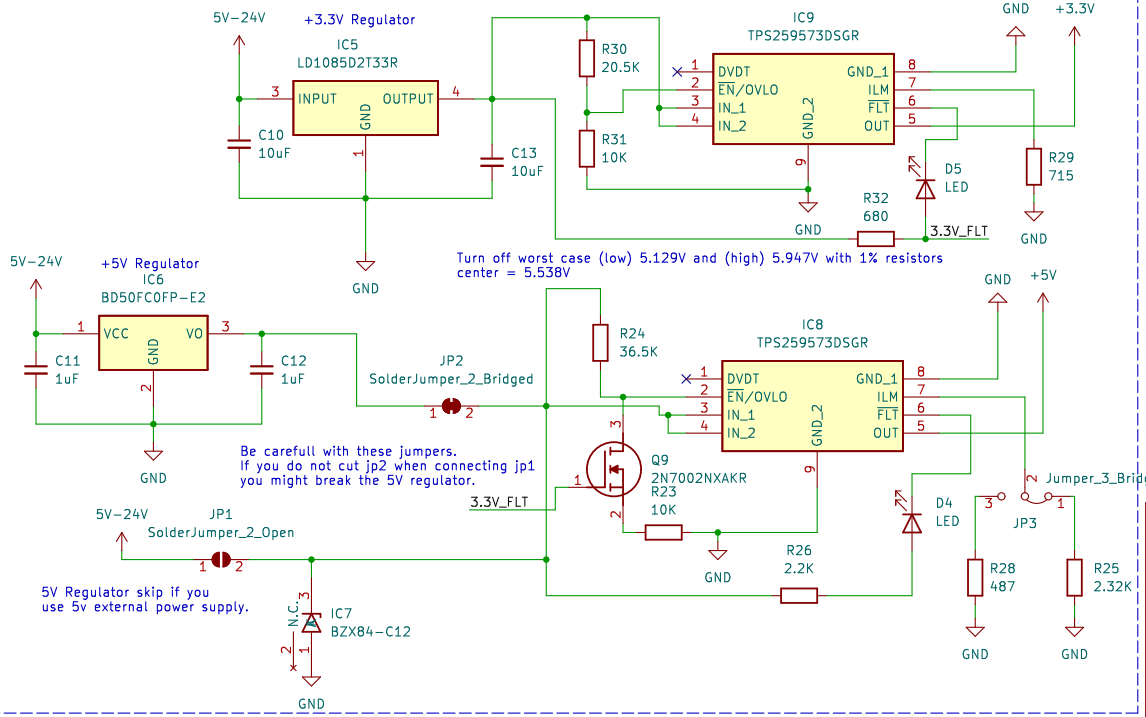
Alternative power connectors and basic circuit protection



You could add a diode protect against reverse polarity but then you lose at best 0.5V over the diode.
The fuse and zenner also protects against reverse polarity.
A fuse is cheap to replace so I did not add it.

Don't use a fuse above 5A.
Use a fuse apropriate for your power supply and current consumption.
Example: If you use a 5V 3A power supply, acidetaly reverse polarity and use a fuse larger than 3A
the zener diode might burn before the fuse breaks.
This is because some power supplys will be able to clamp the current to its max rating
while dropping the voltage to in this case the forward voltage of the zener.
Lets say a forward voltage of 1V current of 3A so 3W of power.
The total power dissipation(Tamb ≤ 25 °C) of the zener with the sandard footprint is 0.682W.
The zener will let out the magic smoke and then all the other components that can not withstand reverse polarity.

Power regulators



Turn off worst case (low) 5.129V and (high) 5.947V with 1% resistors
center = 5.538V

Be carefull with these jumpers.
If you do not cut jp2 when connecting jp1
you might break the 5V regulator.

5V Regulator skip if you
use 5v external power supply.

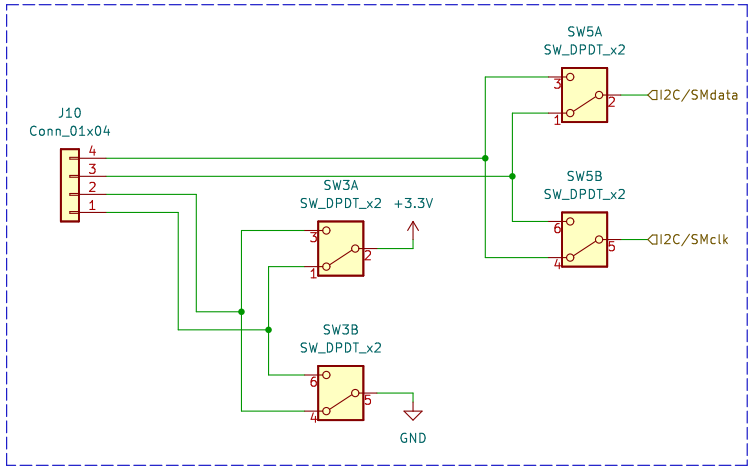
All resistor values are calculated to garantee function with a 1% tollerance

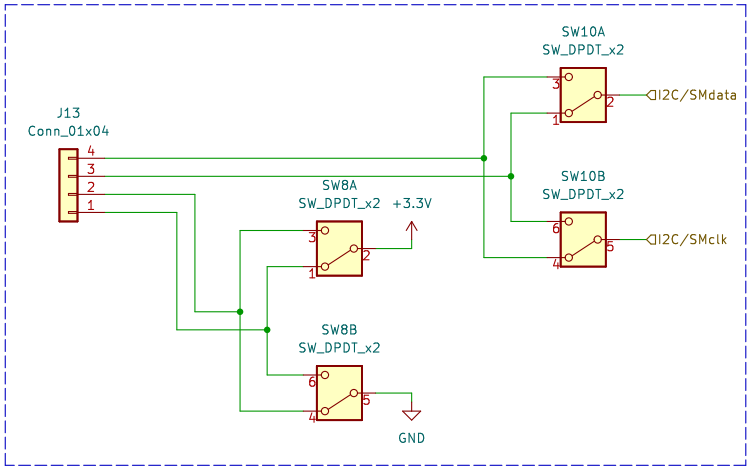
with a margin of 0.07A and a 1% resistor R25 the max current will be 910.78mA
R28 is set to 4A max for the electronic fuse

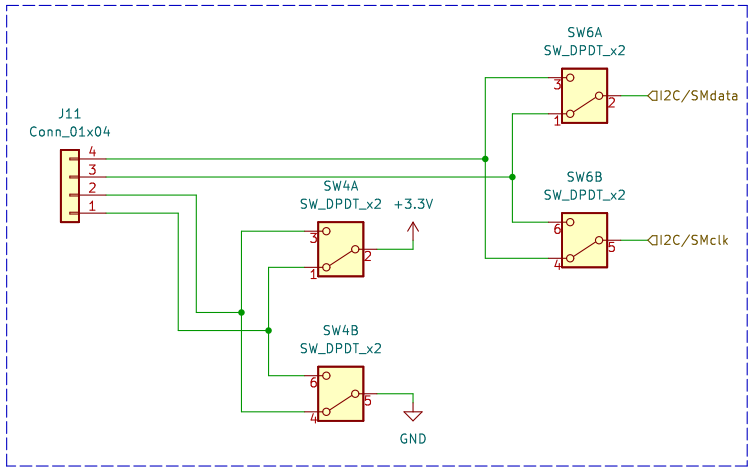
Sheet: /Power/
File: power.kicad_sch

Title: ESPHome All in One Dev kit

Size: A4	Date:	Rev:
KiCad E.D.A. kicad 7.0.10		Id: 10/8







You can use the switches to flip the clock and data pins or the 3.3V and gnd pins.
This ensures compatablity with almost all I2C based modules.

Sheet: /I2C Connectors/ File: I2C.kicad_sch		
Title: ESPHome All in One Dev kit		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 7.0.10		Id: 11/8