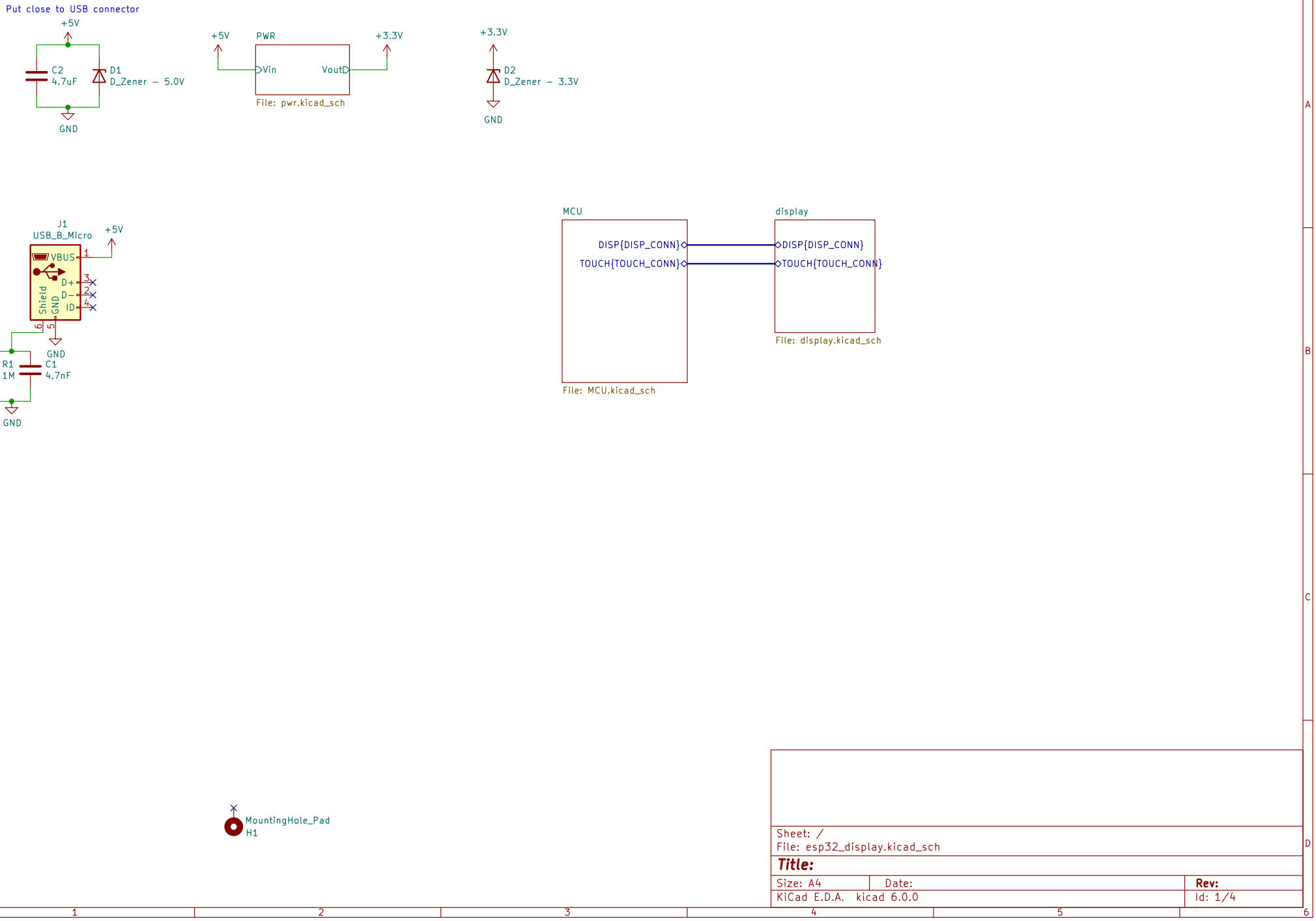
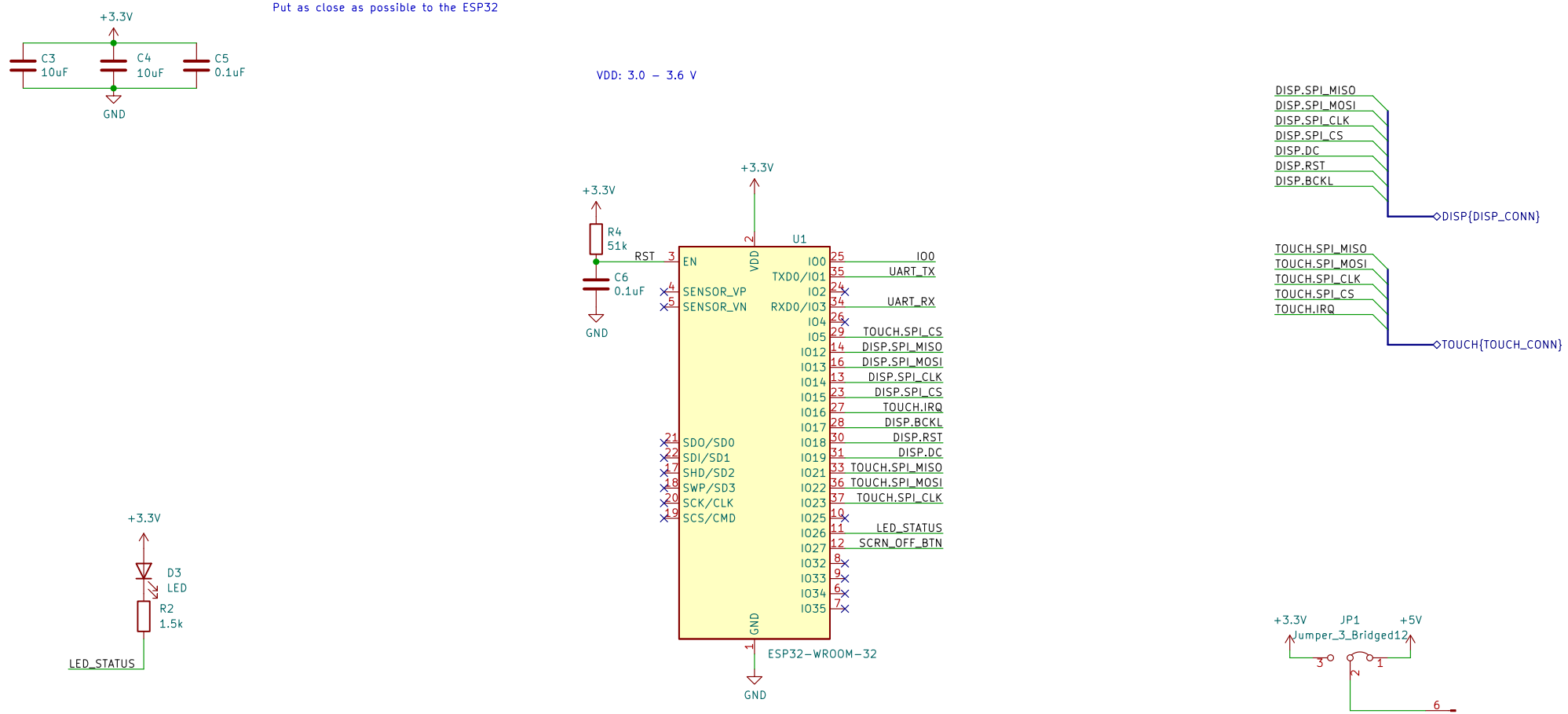
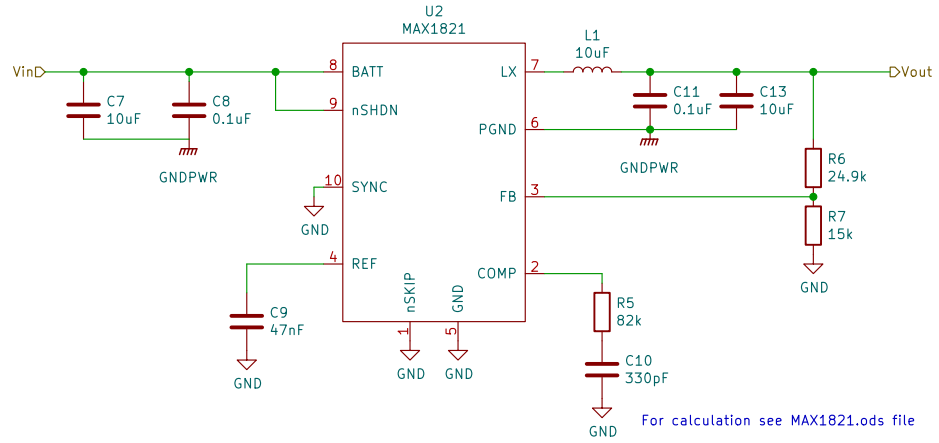

Put close to USB connector
+5V
C2
4.7uF
D1
D_Zener – 5.0V
GND
+5V
PWR
+3.3V
Vin
Vout
File: pwr.kicad_sch
+3.3V
D2
D_Zener – 3.3V
GND
J1
USB_B_Micro
+5V
VBUS
D+
D–
ID
Shield
GND
GND
R1
1M
C1
4.7nF
GND
MCU
DISP{DISP_CONN}
TOUCH{TOUCH_CONN}
File: MCU.kicad_sch
display
DISP{DISP_CONN}
TOUCH{TOUCH_CONN}
File: display.kicad_sch
MountingHole_Pad
H1
Sheet: /
File: esp32_display.kicad_sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 6.0.0
Id: 1/4


Put as close as possible to the ESP32
+3.3V
C3
10uF
C4
10uF
C5
0.1uF
GND
VDD: 3.0 – 3.6 V
+3.3V
R4
51k
RST
C6
0.1uF
GND
+3.3V
U1
EN
VDD
SENSOR_VP
SENSOR_VN
SDO/SD0
SDI/SD1
SHD/SD2
SWP/SD3
SCK/CLK
SCS/CMD
GND
IO0
TXD0/IO1
IO2
RXD0/IO3
IO4
IO5
IO12
IO13
IO14
IO15
IO16
IO17
IO18
IO19
IO21
IO22
IO23
IO25
IO26
IO27
IO32
IO33
IO34
IO35
UART_TX
UART_RX
TOUCH.SPI_CS
DISP.SPI_MISO
DISP.SPI_MOSI
DISP.SPI_CLK
DISP.SPI_CS
TOUCH.IRQ
DISP.BCKL
DISP.RST
DISP.DC
TOUCH.SPI_MISO
TOUCH.SPI_MOSI
TOUCH.SPI_CLK
LED_STATUS
SCRN_OFF_BTN
ESP32-WROOM-32
GND
DISP.SPI_MISO
DISP.SPI_MOSI
DISP.SPI_CLK
DISP.SPI_CS
DISP.DC
DISP.RST
DISP.BCKL
DISP{DISP_CONN}
TOUCH.SPI_MISO
TOUCH.SPI_MOSI
TOUCH.SPI_CLK
TOUCH.SPI_CS
TOUCH.IRQ
TOUCH{TOUCH_CONN}
+3.3V
D3
LED
R2
1.5k
LED_STATUS
+3.3V
JP1
+5V
Jumper_3_Bridged12
GND
UART_RX
UART_TX
RST
IO0
J2
Conn_01x06_Male
RST
C16
4.7nF
RST_BTN
SW1
GND
+3.3V
R3
51k
SCRN_OFF_BTN
C17
4.7nF
SCREEN_OFF_BTN
SW2
GND
Sheet: /MCU/
File: MCU.kicad_sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 6.0.0
Id: 2/4

Vin: 2.6 – 5.5 V
Supply Voltage Input. Connect BATT to a 2.6V to 5.5V.
Bypass BATT to PGND with a low–ESR 10µF capacitor.
The output capacitor must have low impedance at the switching frequency (1 MHz).



Vfb = 1.25 V
R2: 5k – 30k –> R2 = 15k –> R1 = 24.9k
(for calculation details chech MAX1821.ods file)
R1 and R2 must have at least 1% tolerance!

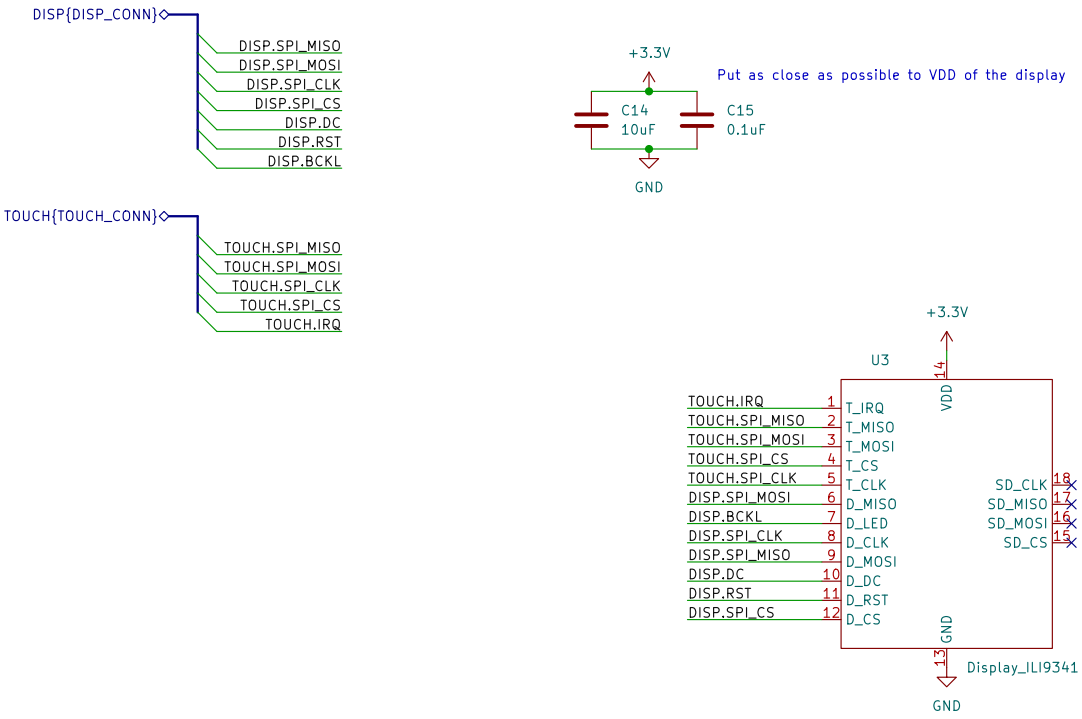
For calculation see MAX1821.ods file

Connect the inductor, input filter capacitor, and output filter capacitor as close together as possible, and keep their traces short, direct, and wide. Connect their ground pins at a single common node in a star–ground configuration.
The external voltage–feedback network should be very close to the FB pin, within 0.2in (5mm). Keep noisy traces (the LX pin, for example) away from the voltage–feedback network; also, keep them separate, using grounded copper.
Connect GND and PGND at a single point, as close as possible to the MAX1820/MAX1821.

Sheet: /PWR/
File: pwr.kicad_sch

Title:

Size: A4 | Date: | **Rev:**
KiCad E.D.A. kicad 6.0.0 | Id: 3/4



Sheet: /display/		
File: display.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.0		Id: 4/4