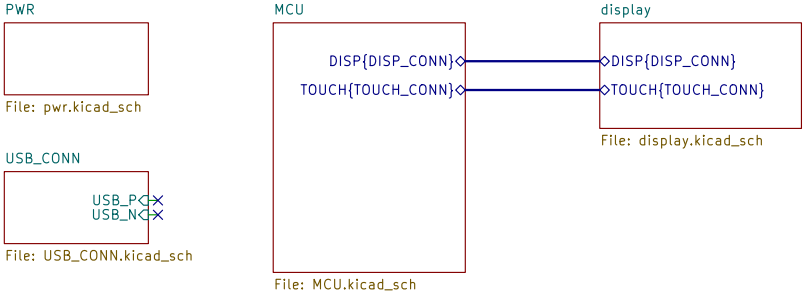
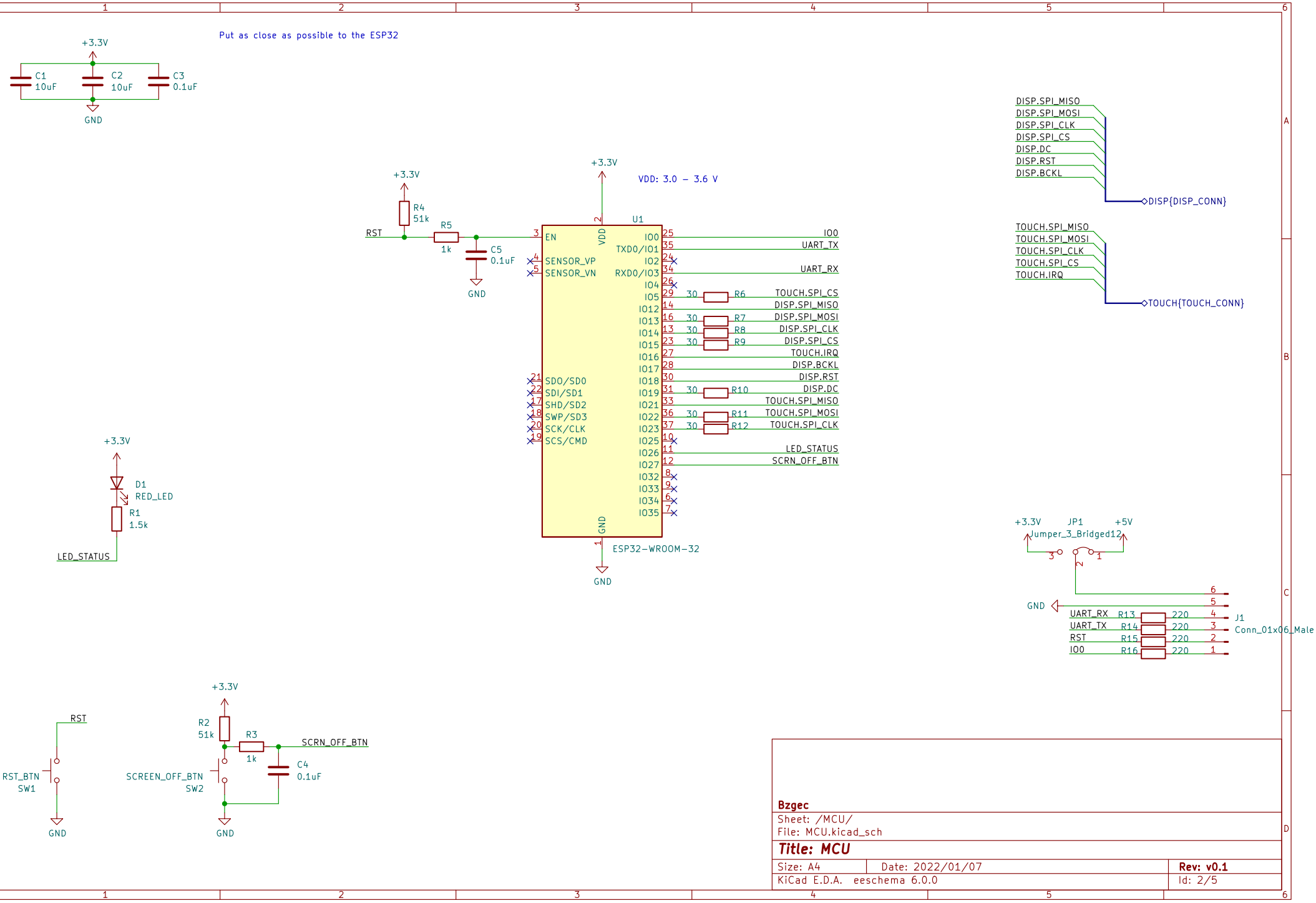


H1
MountingHole

Bzgec
Sheet: /
File: esp32_display.kicad_sch

Title: ESP32_display

Size: A5	Date: 2022/01/07	**Rev: v0.1**
KiCad E.D.A. eeschema 6.0.0		Id: 1/5



Bzgec

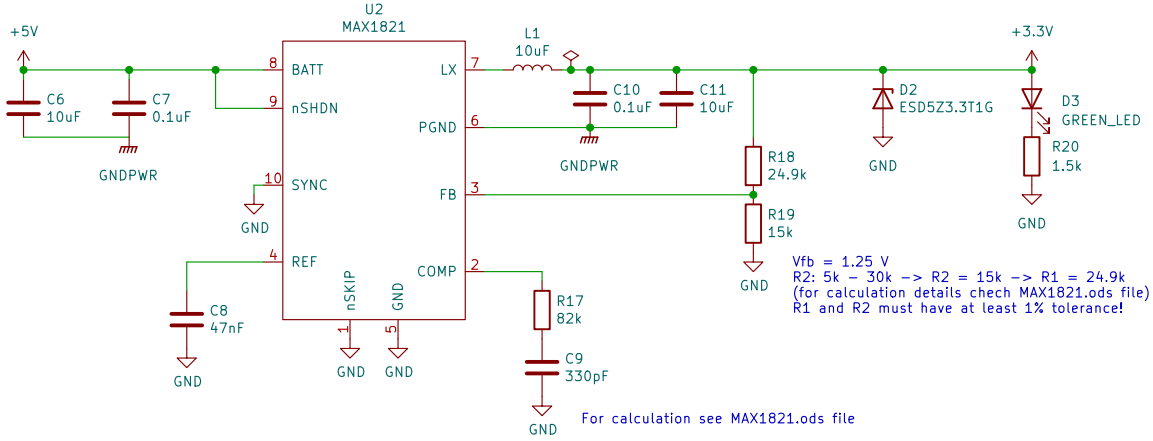
Sheet: /MCU/

File: MCU.kicad_sch

Title: MCU

Size: A4	Date: 2022/01/07	**Rev: v0.1**
KiCad E.D.A. eeschema 6.0.0		Id: 2/5

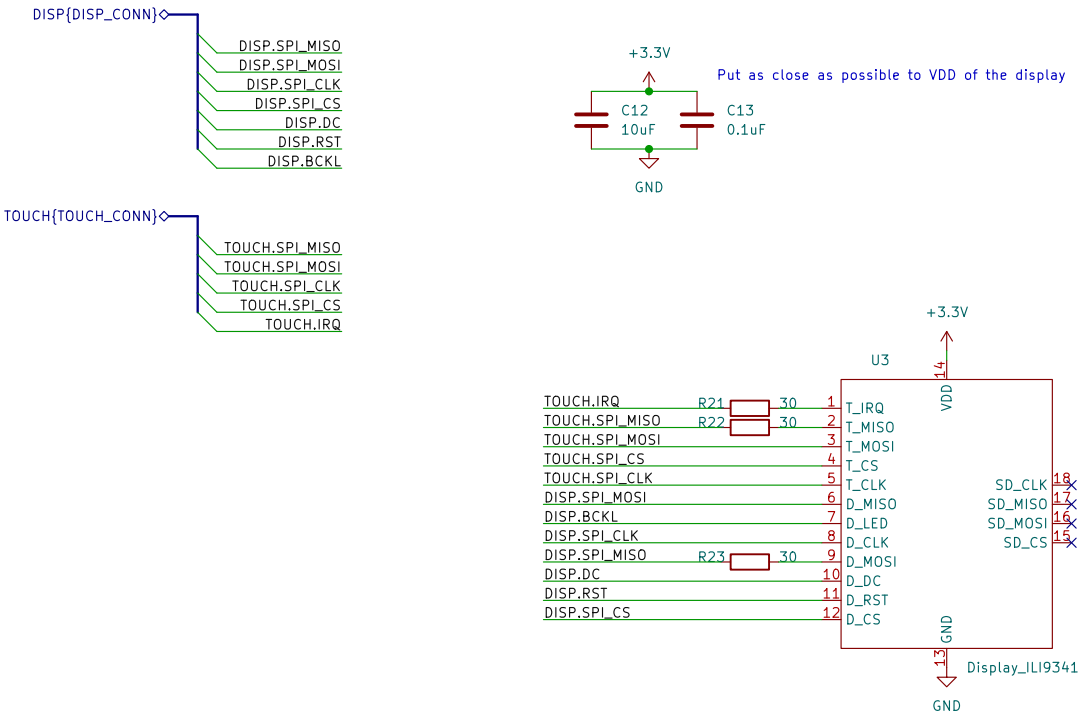
Vin: 2.6 – 5.5 V
Supply Voltage Input. Connect BATT to a 2.6V to 5.5V.
Bypass BATT to PGND with a low–ESR 10µF capacitor.
The output capacitor must have low impedance at the switching frequency (1 MHz).



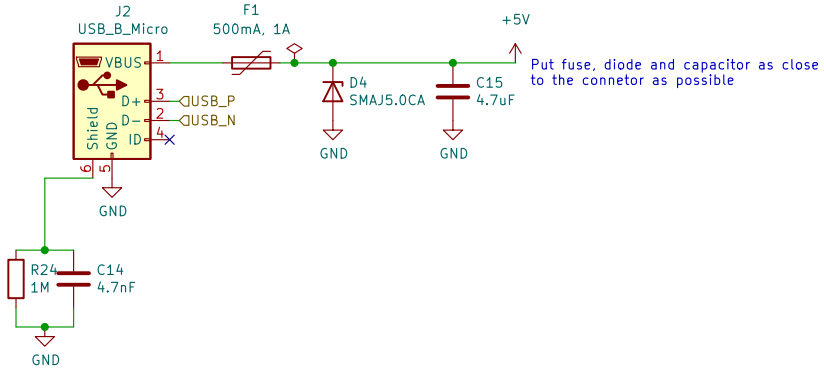
Vfb = 1.25 V
R2: 5k – 30k –> R2 = 15k –> R1 = 24.9k
(for calculation details chech MAX1821.ods file)
R1 and R2 must have at least 1% tolerance!

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.

Bzgec		
Sheet: /PWR/		
File: pwr.kicad_sch		
Title: Power supply		
Size: A4	Date: 2022/01/07	**Rev: v0.1**
KiCad E.D.A. eeschema 6.0.0		Id: 3/5



Bzgec		
Sheet: /display/		
File: display.kicad_sch		
Title: Display		
Size: A4	Date: 2022/01/07	**Rev: v0.1**
KiCad E.D.A. eeschema 6.0.0		Id: 4/5



Bzgec		
Sheet: /USB_CONN/		
File: USB_CONN.kicad_sch		
Title: USB connector		
Size: A4	Date: 2022/01/07	**Rev: v0.1**
KiCad E.D.A. eeschema 6.0.0		Id: 5/5