

Use CS11102.3 for buttons  
instead. Footprint is  
identical and I can't be  
bothered to change em all

Licensed under CERN-OHL-S v2

Sheet:

File: buton\_matrix.kicad.sch

Title: 71-83x Button Matrix

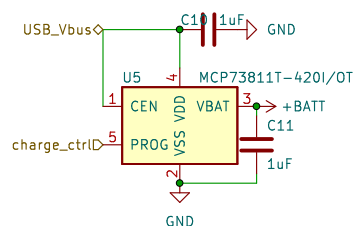
Size: USLetter Date: 2023-02-21

KiCad E.D.A. kicad 7.0.1

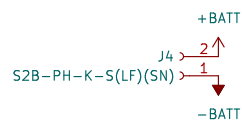
Rev: A

Id: 2/3

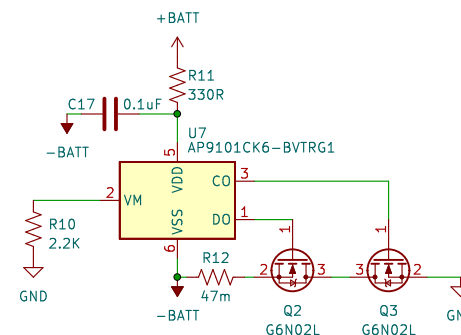
## LP904040 85mA/450mA Battery Charger



## LP904040 Battery

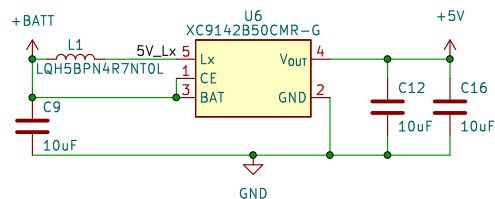


## Battery Management System

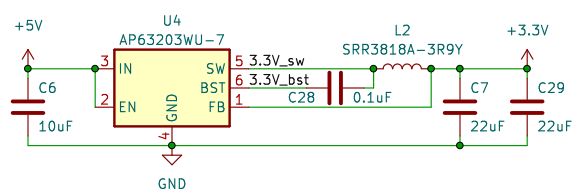


R12 acts like a 50 milliohm resistor;  
the traces add about 3 milliohms

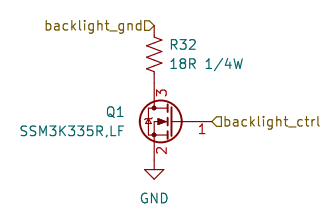
## Battery to 5V 700mA



## 5V to 3.3V 2A Supply



## LCD Backlight 100mA Driver



For R32, I actually put a 39 ohm and 27 ohm resistor  
in parallel to make a 15.95 ohm resistor; this gives  
us a max current of 119 mA which is still within spec  
for our display.

Q1 can be any old NMOS, so you can use a G6N02L to  
match Q2 and Q3 if you like -- looks like the pinout  
is the same.

Licensed under CERN-OHL-S v2

Sheet:

File: usb\_power\_supply.kicad\_sch

Title: TI-83x Power Supply

Size: USLetter Date: 2022-10-04

KiCad E.D.A. kicad 7.0.1

Rev: A

Id: 4/3