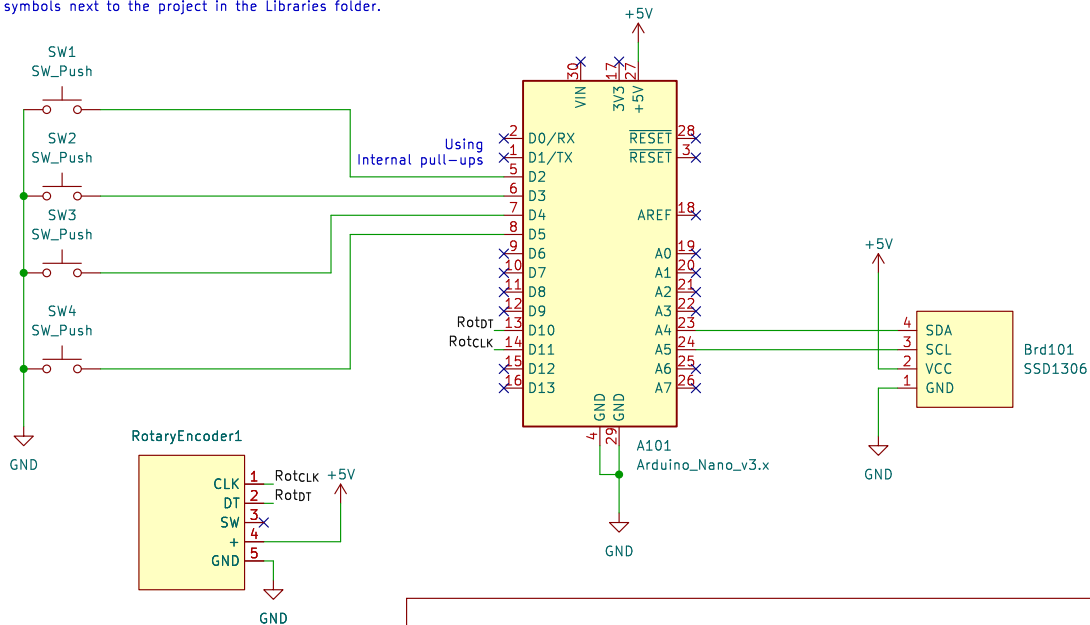


You can add Power symbols with the "P" shortcut
 You had the power rail (+5V) connected to GND, that's a short circuit
 You're not using the rotary encoder switch?
 Use no-connection flags (the small x's) on every un-used pin.
 What function do these buttons have, maybe name those functions in the schematic?
 I like to keep all symbols next to the project in the Libraries folder.



Sheet: /

File: MusicControllerPCB.kicad_sch

Title: Add some sort of a title?

Size: A5 Date: 2023-12-01

KiCad E.D.A. eeschema 7.0.9-ubuntu22.04.1

Rev:

Id: 1/1