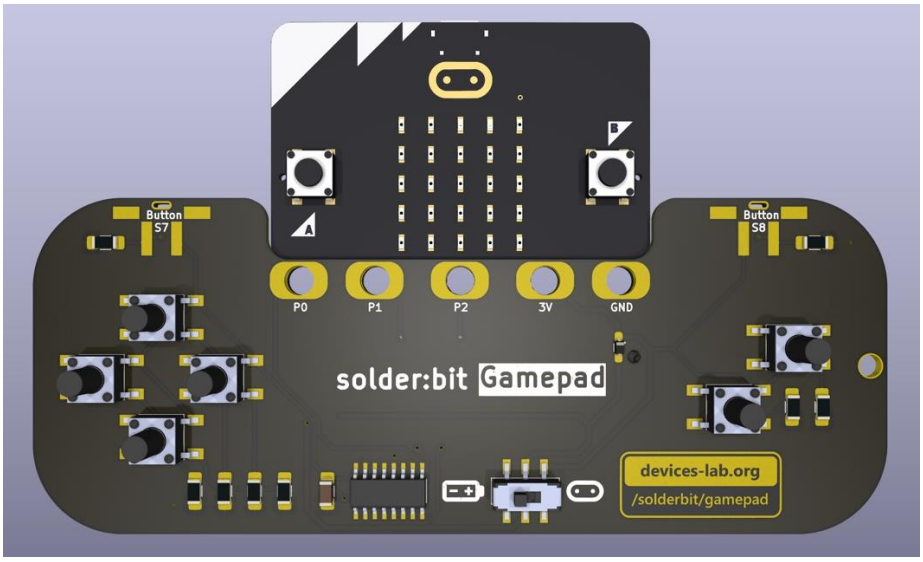
PCB Design Walkthrough with KiCAD

John Vidler and Aron Eggens
Computing and Communications
Lancaster University

https://github.com/devices-lab/pro2-kicad-workshop

You are given the task of extending the Solder:bit boards with more NeoPixel LEDs. A design review has highlighted that we need more blinking lights than the 5x5 LED matrix on the Micro:bit can supply!



Tasks:

1. Download the existing design files by cloning the Git repository:

   https://github.com/devices-lab/pro2-kicad-workshop

2. Add 5 Neopixels and their support components to the design in the Schematic Editor
3. Select which footprints you need for the new parts with the Part Selection tool
4. Move to the PCB Layout Editor and import the changes
5. Lay out your new components (extra points for being artistic with your placement! 😊)
6. Build a Bill Of Materials (BOM) and Gerber file set for the next workshop

Don't worry if you don't manage to get all the steps here completed, as we also supply a set of completed production files (BOM and Gerber files) in the same repository which can be used for the next session.

Handy Web Links:

- https://docs.kicad.org/8.0/en/ - The Version 8 KiCAD documentation
- https://education.github.com/git-cheat-sheet-education.pdf - A Git cheat sheet