



Important notes for the experimental design

- The PCB has been designed in KiCAD
- KiCAD is open-source software (kicad.org)
- The PCB design is optional in HOMS project
- The KiCAD design-files are for experimental purposes
- The initial design is for 2-digit seven-segment display
- Extra wiring is needed for placing the 4-digit seven-segment display
- There is no theoretical schematic in current project
- The circuit is developed directly in the PCB editor
- The GND connections are valid only through Arduino and display module

Advices for more advanced designers

- Draw the theoretical schematic of the electrical circuit, before the PCB design
- Use GND components and labels properly in the schematic for implementing a ground plane in the PCB editor