

Exercise 3: PCB Design

In this exercise, you will learn how to use the EDA (Electronic Design Automation) tool KiCAD to create PCBs (Printed Circuit Boards). The first task will familiarise you with the basics of the tool, while the second one will also consider requirements by a potential manufacturer as well as the use of libraries for custom components.

Exercise Description:

1. Task 1 (5 points) - Astable Multivibrator

- Schematic
 - Replicate the schematic provided for the Astable Multivibrator
 - Utilize components available in the default libraries of KiCad
 - Place components and connect their pins using traces
 - Configure components with the specified values
 - Assign appropriate footprints for the components
 - Annotate the schematic and ensure all components are correctly labeled
- Printed Circuit Board
 - Create a PCB from the schematic
 - Place the components considering their intended usage and physical dimensions
 - Route the traces manually (F.Cu and B.Cu) and use vias only if necessary
 - Draw the PCB's contour (Edge.Cuts)

2. Task 2 (10 points) - Microcontroller-based Running Light

- Create an advanced microcontroller-based circuit that would be capable of generating an 8-channel running light
- Follow the manufacturer's design rules, like those from EuroCircuits, (available at <http://www.eurocircuits.com/blog/kicad-design-rules/>)
- Use the external library for the TinyPico Nano microcontroller that can be found on <https://www.tinypico.com/tinypico-nano> and <https://github.com/tinypico/tinypico-hardware>
- Place the components thoughtfully to ensure clear routing paths and label them with appropriate values
- Arrange 8 LEDs in a row and configure them
- Add two buttons: one for start/stop functionality and the other to switch between different modes
- Label the traces (e.g., VCC, GND, SIG_LED1, SIG_BUT1, ...)
- Adjust the trace widths considering current flow and limit heat rise to $\leq 10^{\circ}\text{C}$ (using KiCAD's internal tool)
- Route the traces to ensure signal integrity, avoid crossovers, and minimize the number of vias
- Miniaturize the PCB (e.g., optimized trace routing, SMT, double-sided design, ...)
- Perform the DRC to identify and resolve design violations

Deliverables:

Deadline: Tuesday, 14 January 2025, 11:59 PM

Individual submission of a *.zip archive named "[family name]_exercise3.zip" to TUWEL, containing the following files:

- Two folders for "Task1" and "Task2"
- Both folders must contain:
 - The KiCAD project, schematic, and board files (*.kicad_sch, *.kicad_pcb and *.kicad_pro)
 - The exported/printed schematic and board as *.pdf
 - Two screenshots of the PCB's 3D view, one of each side
- A short reflection (max 1 page):
 - Technical aspects of implementation (design rules, component placement, and trace routing)
 - Reflect specifically on the miniaturization (task 2) strategy implemented or intended