

Construction of electronic systems

Exercise 2: PCB design

You will design a *printed circuit board* (PCB) based on the schematic that you prepared during the first lab exercise.

Exercise tasks:

1. Add a PCB design file (.PcbDoc) to the existing "LED driver" project and transfer the schematics to the PCB design.
2. Design the PCB according to the **requirements** below:

2.1. **PCB dimensions:** 22 mm × 14 mm (see Figure 1 below)

2.1.1. place the *power supply part* of the circuit on the left side of the board with dimensions 16 mm × 14 mm

2.1.2. place the *LED diode* in the center of the remaining part of the board with dimensions 6 mm × 14 mm

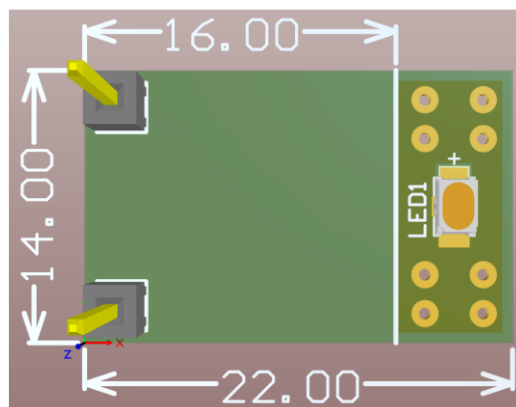


Figure 1 - geometrical requirements for the PCB (dimensions in millimeters)

2.2. **LED cooling:** ensure a good *heat transfer* away from the LED:

2.2.1. create a *copper pour* on the top surface surrounding the LED (Figure 1, the yellow-green surface around the LED)

2.2.2. connect the *thermal pad* of the LED to the surrounding copper pour on the top side

2.2.3. create a copper pour on the entire bottom surface of the board

2.2.4. connect both copper pours using vias to ensure good *thermal conductivity* from the smaller top copper pour to the bigger bottom copper pour

2.3. **board edge clearance:** all components and copper connections must be at least 0.5 mm away from the PCB edge (tip: you can use guide lines (slo. pomožne črte) or use a "board outline clearance" design rule).

2.4. **component and track placement:** all components and connections must be placed on the top side of the PCB; the bottom side of the PCB can be used as a ground (GND) copper pour that provides a wide return path for all currents

2.5. **connector location:** the two single-pole pin header connectors must be placed along the shorter PCB edge (see Figure 1)

Explanation of the exercise

During this lab exercise you will learn how to use the electrical circuit schematic to design a printed circuit board (PCB) that is ready to be manufactured. You will see how a circuit schematic can quickly *help you make a good starting point for designing the PCB* by using the component libraries to prepare all the necessary component footprints and, also, by showing you how these PCB components are interconnected. We will look at the most basic guidelines regarding the PCB *component placement* and *track routing*. Also, we will introduce you to the *basics* of PCB design in light of *thermal management* and in light of *minimizing the electromagnetic interferences*.

Preparation for the lab. exercise

In order to prepare for this exercise you should see the intro part of the Altium Designer video tutorials, covering the basics of the PCB design:

- [Altium intro #20: Preparing PCB design](#)
- [Altium intro #21: Importing schematics into PCB](#)
- [Altium intro #22: Component placing](#)
- [Altium intro #23: Component routing](#)
- [Altium intro #24: Design rules](#)
- [Altium intro #25: Final editing of PCB](#)
- [Altium intro #26: Copper plane](#)
- [Altium intro #27: PCB Design rules check](#)