Construction of electronic systems

Exercise 1: electrical circuit schematic

You will prepare a schematic of the electrical circuit based on a data sheet. This schematic will later serve as the basis for the design of a printed circuit board (PCB). You will design the schematic using the electronic components that you will find or define in the component libraries.

Exercise tasks:

1. Browse the web and find a *datasheet* for the integrated circuit ZXSC300 manufactured by Diodes Inc. You will use this chip to design a *switched-mode power supply* for a LED diode (also called a "LED driver").

2. In this data sheet, find the schematic that is *optimized for the maximum brightness* of the LED.

3. Create a new Altium Designer PCB project called "LED driver" and add an empty schematic to the project. Also add the component *libraries to the project*. You will find them in the "attachment" folder in the materials on the eFE.

4. Design an Altium Designer schematic for a LED driver based on the datasheet schematic. The LED driver is to be powered by a single 1.5 V AA battery.

5. Replace the original circuit components from the datasheet with the components found in the *bill of materials* (BOM) below. **Note:** make sure that you use the correct *component footprints*.

battery connector	2 x single-pole pin headers
inductor	Würth elektronik inductor, SKU=74406042101
Schottky diode	SK14B
transistor	PBSS4120T
resistor	0,13 Ω (size 2012 metric, 0805 imperial)
capacitor	2,2 μF (size 2012 metric, 0805 imperial)
LED	Würth Elektronik LED, SKU=158302250

Table 1 - bill of materials

6. Following the rules of good practice, we will improve the circuit by adding a generic *decoupling capacitor* to the power supply of the integrated circuit (slo. blokirni kondenzator). Use a 100 nF 0805 ceramic SMD capacitor.
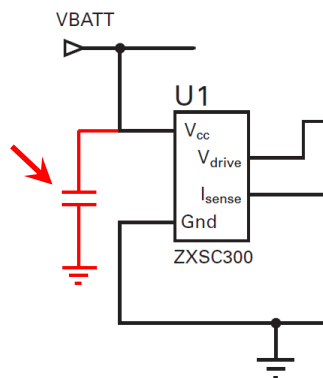

Figure 1 - adding a generic decoupling capacitor to stabilize the power supply voltage for the IC

Explanation of the exercise – electrical circuit schematic

This first exercise will serve as an introduction to Altium Designer, where you will learn the basics of making an electrical circuit schematic using component libraries. Creating a circuit schematic is an important step in the design because with a schematic *you describe your circuit on an abstract level*: you define the components of the circuit and the connections between these components. In this way you are actually using a schematic to *"capture" the key idea of how your electrical circuit works*. And this is the reason why the schematic is a very useful tool which helps you *develop*, *improve*, *organize* and *graphically represent* your idea about how to build a functional electrical circuit.

But all of the above can be simply achieved by drawing a schematic on a piece of paper. That is true, but the design software tools like Altium Designer offer much more than just this. For instance, such a schematic can contain the information about the actual component *footprints*, which makes it very easy to speed-up the process of printed circuit design – this information is simply transferred from the schematics to the PCB design tool. At the same time, such a schematic can also help us with PCB design by showing us which physical components on the PCB are actually connected. There are also many other advantages of such schematics as you will find out during this course.

You can imagine by now that it is reasonable to make an effort to *prepare good schematics and component libraries*. Therefore, we will discuss how to do this during this lab exercise.

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 electric circuit schematics design:

- **Altium intro #02: First project**
- **Altium intro #03: Libraries**
- **Altium intro #04: Component placement**
- **Altium intro #05: Component wiring**
- Altium intro #06: Net labels
- **Altium intro #07: Selecting and naming the components**
- Altium intro #08: Shortcuts
- **Altium intro #09: File based libraries**
- Altium intro #10: Component editing
- **Altium intro #11: New symbol creation**
- Altium intro #12: New footprint creation
- Altium intro #13: Manufacturer part search