International Undergraduate Research Program

Preliminary design

Due date: 21/12/31 (can be extended)

KAIST EE

SEED Lab

Advisor: Wanyeong Jung

TA: Michal Gorywoda

Your task will be designing a printed circuit board with STM32L082CZ microcontroller that is comparable to Nucleo or Arduino boards. This activity will prepare you for designing a more complex device later during this project. To design the PCB, you will use Altium Designer with academic license which you can obtain by registering with your university e-mail address.

Below is the list of requirements for the board:

- STM32L082CZ microcontroller
- Arduino-compatible female pin connectors
- External programming/debugging connector for loading the program
- USB Virtual COM Port to allow communication between the microcontroller and PC – this will require an USB connector and protection diodes
- USB interface will also be the power supply. However, since STM32 cannot operate with 5V provided by USB, you will need insert an LDO linear regulator to create 3.3V supply
- Sufficient amount of power supply noise decoupling capacitors
- A simple sensor with SPI – for example TMP125 or BME280
- 4 push buttons and 4 LEDs for basic user interaction
- Use 0603 sized package for passive components and SMD packages for integrated circuits
- 2-layer PCB with good power and ground connections (polygon planes)
- No warnings on DRC (design rule check) after board is finished

During first week you will focus on obtaining the tools and getting familiar with design process, the component datasheets and reference designs.

Next, you will need to make a schematic. It will be checked for correctness, as it will provide a reference for layout checking. You do not need to make your own components – use Manufacturer Part Search in Altium to obtain schematic symbols and footprints.

When the schematic is complete, you will define the PCB shape and place the components according to good practice rules. After that is done, you can proceed to connecting the components. Finally, we will discuss the work and point out things that can be improved in layout.

List of references:

- Altium Designer website
- Altium Designer tutorial
- STM32L082CZ information - look for datasheet and application note
- STM32 software tutorial 1
- STM32 software tutorial 2
- STM32 software tutorial 3
- Tutorial on PCBs with STM32
- STM32 Nucleo64 reference design