ASSIGNMENT 01 – EAGLE

Objective

In this assignment, you are required to design a schematic, a PCB, and custom devices using Autodesk EAGLE software.

Context

Your company, the LTEBS srl, would like to use the PSoC 5LP micro-controller in a project requested by a customer. In order to evaluate the performance of the micro-controller and to determine if such uC is suitable for the project, the R&D department would like to design a PCB embedding the PSoC 5LP. This way, testing and assessment of the performance would be a lot easier than using a simple breadboard. As Cypress offers a development kit for the PSoC 5LP, namely the CY8CKIT-059, you are requested to design a PCB embedding suck kit, which will be soldered on male jumpers, and additional components required to conduct the proper testing by the R&D department. As some of the requested components are not present in EAGLE installation libraries, you are required to design a library containing such components. The PCB will be manufactured by MDSrl.

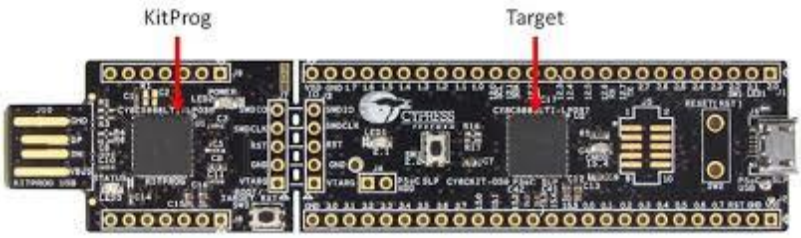

Figure 1: The CY8CKIT-059 development kit offered by Cypress.

Assignment Description

This assignment requires you to use GitHub. The suggested workflow for GitHub setup is the following one:

1. One team member creates a **private** GitHub repository following this naming convention: "LTEBS_2021_II_Group_XX_EAGLE", where XX must be substituted with your group number
2. Following the repository creation, the same team member fills in the following Microsoft Form to provide use the link to the GitHub repository
3. The other team member(s) are added to the repository as collaborators via GitHub (see Figure 2)
4. Each team member clones the created repository locally
5. After the deadline for assignment delivery (March 26, 6 PM), the repository must be made **public**

The repository must abide the following folder structure:

- **Library**
  - "Group_XX_Library.lbr"
- **Project**
  - "Group_XX.brd"

- “Group_XX.sch”

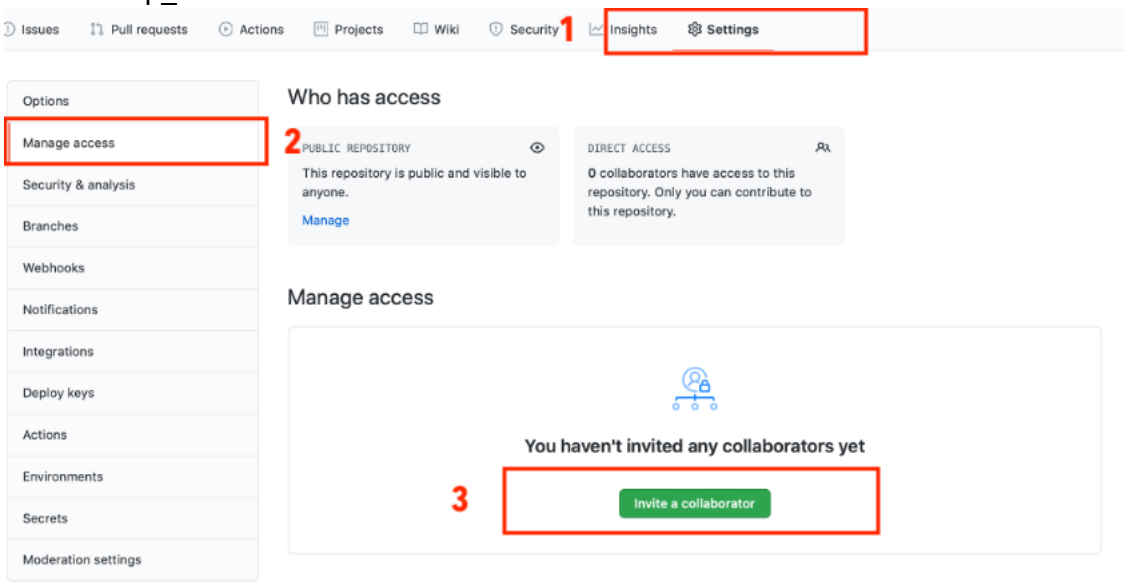

Figure 2: GitHub repository access management.

- **Design Rules**
  - “Group_XX.dru”
- README.md
- .gitignore

Starting from the schematic reported in the attached PDF file, and the list of components in the table below, design a EAGLE schematic and save it in your repository.
Then, create the associated PCB. Route all the components, and check that the ratsnest command does not provide any error message.
Create a design rule files according to the technical specifications reported in the manufacturer website. Perform a DRC (Design Rule Check) using the created DRC file and check that no error or warning is generated. No errors/warnings can be approved, they must be all solved.

Schematic Name	EAGLE Device	EAGLE Footprint	EAGLE Library
CY8CKIT059-TARGET	CY8CKIT-059_TARGET		CY8KIT-059
J1	POWER_JACKPTH_LOCK	POWER_JACK_PTH_LOCK	SparkFun-Connectors
C2	0.33UF/330NF	0805	SparkFun-Capacitors
C1, C3	0.1UF-KIT-EZ-50V-20%	CAP-PTH-SMALL-KIT	SparkFun-Capacitors
U1	V_REG_78XX7812	TO-252	SparkFun-IC-Power
J2…5, J7	CONN_02	1X02	SparkFun-Connectors
R2…9	4.7KOHM-0603	0603	SparkFun-Resistors
R10	PHOTOCELLPTH	PHOTOCELL	SparkFun-Sensors
R11	10KOHM	0603	SparkFun-Resistors
U2	TMP36GT9	TO-92	SparkFun-Sensors
J6	CONN_06	1X06_LOCK	SparkFun-Connectors
D1	LED-BLUE1206	LED-1206	SparkFun-LED
R1	RESISTORAXIAL-0.3	AXIAL-0.3	SparkFun-Resistors
U4	TMUX1208	QFN16	**
U3	TCA9546A	TSSOP16	**

** These components must be designed by you.

Assignment Delivery

The project version that will be used for evaluation purposes must be committed with commit message “Final version”. Only one commit must have this message. The deadline is March 26, at 6 PM.

Assignment Evaluation

The following criteria will be used for evaluation:

- EAGLE Project and Library
    - Organization of the schematic: proper use of net labels, component placement, and comments
    - Correct connections on the schematic
    - Organization of the library: documentation and proper file names for symbols, footprints, and devices
    - Correctness of the library devices
    - Dimensions of the board: lower dimensions are preferred, as PCB are priced by size
    - Correct component placement on the PCB (*always consider that components are three-dimensional, not two-dimensional as you see in EAGL*E)
    - SMD component placement on TOP layer
    - GND and 5V placed on TOP and BOTTOM layer with correct values of isolate and spacing
    - Minimum trace widths of 10 mils
    - Correct routing of the PCB
    - Proper and useful silkscreen on PCB
    - Proper placement of M3 holes at PCB corners
- Git Collaboration
    - Correct .gitignore for EAGLE projects
    - Collaboration among team members
    - Proper use of branches
    - Successful delivery of the project (final version)

Additional Resources

The following documents/tutorial can be useful for this assignment:

- SparkFun EAGLE Tutorial
- EAGLE Layers
- Getting Started with EAGLE