

Lab 8: Printed Circuit Board (PCB) Design Using Eagle

I. Objectives

1. Using Eagle to design a schematic of a given circuit
2. Using Eagle to design Printed Circuit Board (PCB) from a schematic
3. Designing PCB for a simple STM32F103 development board

II. Introduction

A printed circuit board (PCB) is a board which is **made of several conducting and non-conducting layers**. A PCB is **used to mechanically support and electrically connect electronic components** or electrical components **using conductive tracks, pads and other features** etched from one or more **sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate**. Components are generally soldered onto the PCB to both electrically connect and mechanically fasten them to it. An example of a PCB is shown on Figure 1 for an STM32F103C8T6 development Board.

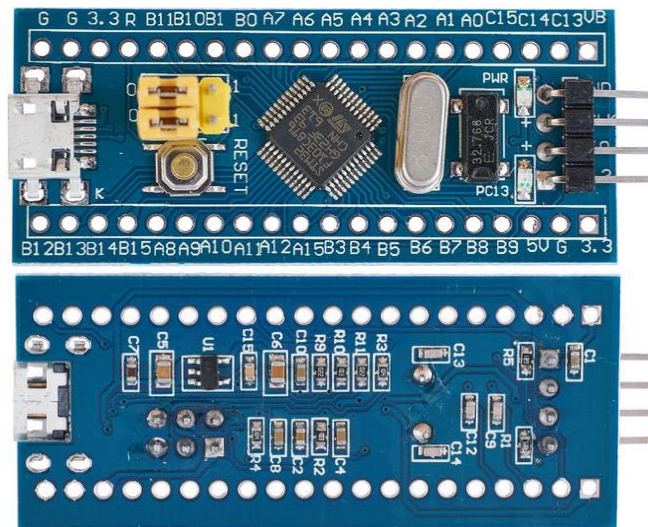


Figure 1. STM32F103C8T6 Board (top and bottom sides)

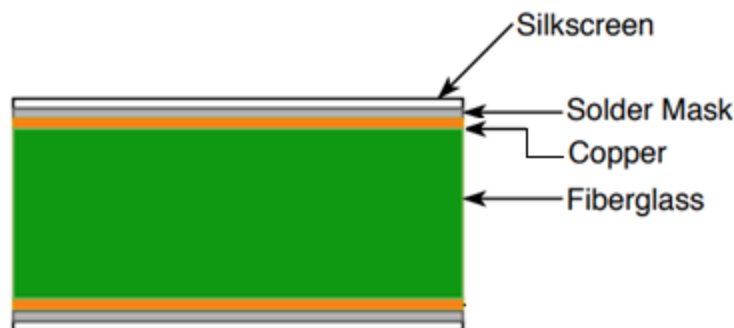


Figure 2. Double-Sided PCB Layers

A PCB is sort of like a sandwich of layers. There are alternating layers of different materials which are laminated together with heat and adhesive such that the result is a single object. The layers are illustrated by Figure 2. These layers are:

- **The base material**, or substrate (middle layer), is usually fiberglass. Historically, the most common designator for this fiberglass is "FR4". This solid core **gives the PCB its rigidity and thickness**.
- **Copper**: The next layer is a thin copper foil, which is laminated to the board with heat and adhesive. On common double sided PCBs, copper is applied to both sides of the substrate. In lower cost electronic applications, the PCB may have copper on only one side. In sophisticated applications (e.g., laptop motherboard), **the numbers of copper layers could be 16 or more**. The copper thickness can vary and is specified by weight, in ounces (1 ounce is approximately 28.35 grams) per square foot. The vast majority of PCBs have 1 ounce of copper per square foot but some PCBs that handle very high power may use 2- or 3-ounce copper. Each ounce per square translates to about 35 micrometers.
- **Solder Mask**: is the layer on top of the copper foil. This layer gives the PCB its green (also, blue, red, black, yellow, ...) color. It is overlaid onto the copper layer **to insulate the copper traces from accidental contact** with other metal, or solder, or conductive bits. This layer helps the user to solder to the correct places and prevent solder jumpers (check Figure 3).
- **Silkscreen**: is an **ink layer** (usually white ink but any color can be used) applied on top of the solder mask layer. The silkscreen **adds letters, numbers, and symbols to the PCB** that allow for easier assembly and indicators for humans to better understand the board. We often use silkscreen labels to identify different circuit components (check Figure 3).

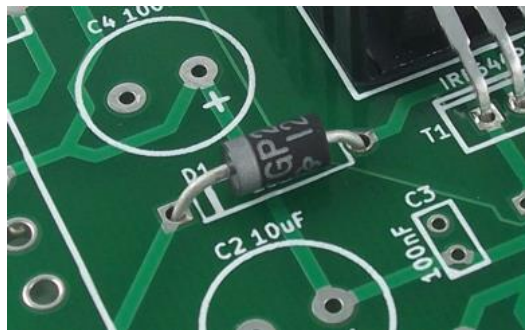


Figure 3. A part of a PCB showing the Solder mask and Silkscreen

The fabrication starts with Copper Clad Laminate (CCL) like the one shown on Figure 4. There are several ways for fabricating PCBs; however, the most commonly used fabrication process involves photo lithography using masks for every PCB layer and acid etching.



Figure 4. Copper Clad

The masks used in fabrication must be designed beforehand based on the circuit we want to implement. **The design process starts with drawing the schematic of the circuit to be implemented using a schematic capturing tool such as Eagle schematic editor (Figure 5).** The schematic is **built out of components from component libraries that come with the schematic capturing tool or created by the designer.** The components included in the library are **real components that can be purchased from component distributors** (such as mouser.com, digikey.com, ...). Every library component has few views. For designing a PCB, we will use the symbol view in the schematic capturing step.

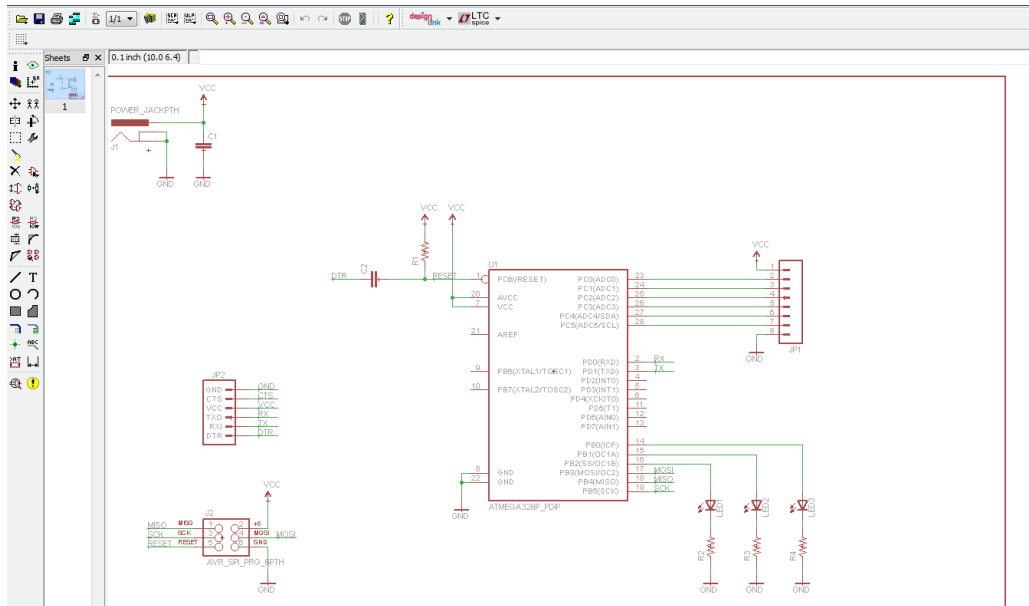


Figure 5. Eagle Schematic Capturing tool

Once the schematic is done, we can export it to **Eagle PCB design tool** as shown on Figure 6. There, you **define the board dimensions, place the components** (the footprint view is used) on the board **and route the connections**. Once done (check Figure 7), you may export the files needed for manufacturing (Gerber files).

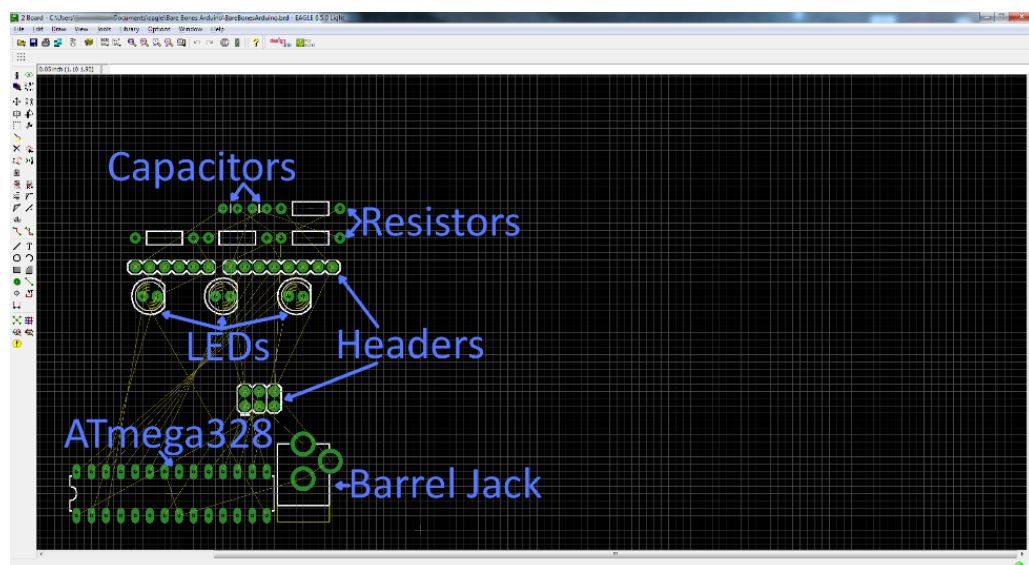


Figure 6. Eagle PCB design tool

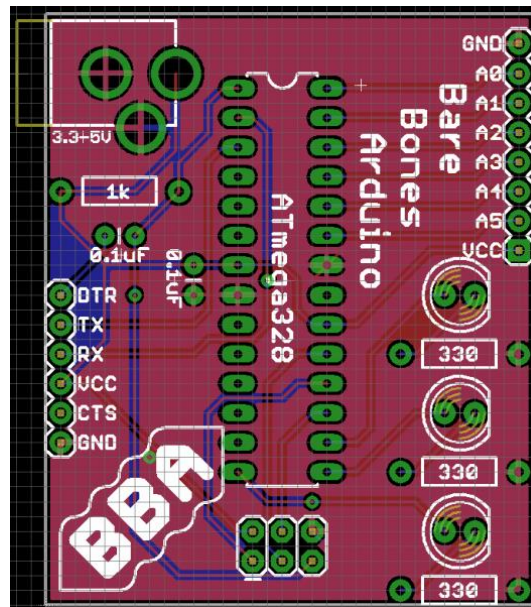


Figure 7. Finished PCB design

III. PCB Terminologies

Pad: is a **portion of exposed metal on the surface of a board to which a component is soldered** (check Figure 8). There are 2 types of pads; thru-hole and smd (surface mount) (check Figure 9).

- **Thru-hole pads** are **intended for introducing the pins of the components**, so they can be soldered from the opposite side from which the component was inserted.
- The **smd pads** are **intended for surface mount devices**, or in other words, for soldering the component on the same surface where it was placed.

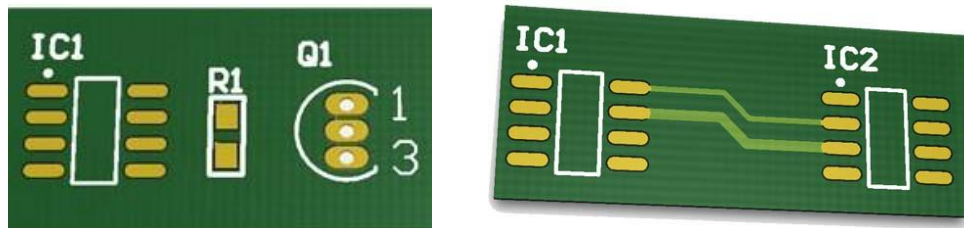


Figure 8. Pads and Tracks

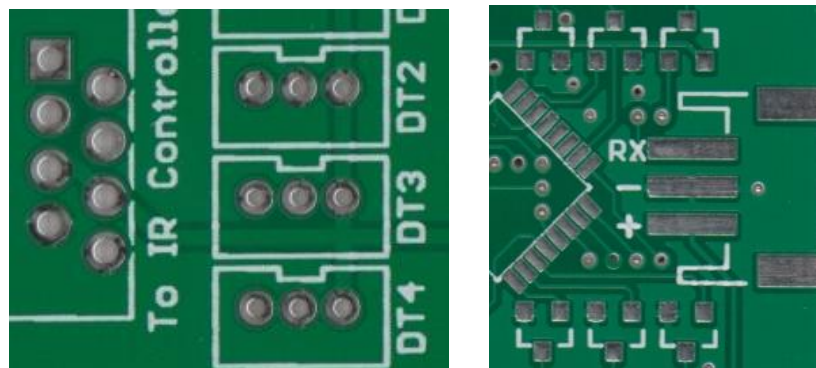


Figure 9. PTH (plated through-hole) pads on the left, SMD (surface mount device) pads on the right

Copper Track: A track is a **conductive path** that is **used to connect 2 points in the PCB**. For example, for connecting 2 pads or for connecting a pad and a via, or between vias. The tracks can have different widths depending on the currents that flow through them. Power tracks are wider and signal tracks are narrower. Usually we use the following table to decide on the track width given the maximum current it may carry.

Table 1. Recommended Track widths

Current/A	Track Width(mil)	Track Width(mm)
1	10	0.25
2	30	0.76
3	50	1.27
4	80	2.03
5	110	2.79
6	150	3.81
7	180	4.57
8	220	5.59
9	260	6.60
10	300	7.62

1 inch = 1000 mil

VIA: is a Pad with a plated hole **connecting copper tracks from one layer of board to other layers** (check Figure 10).

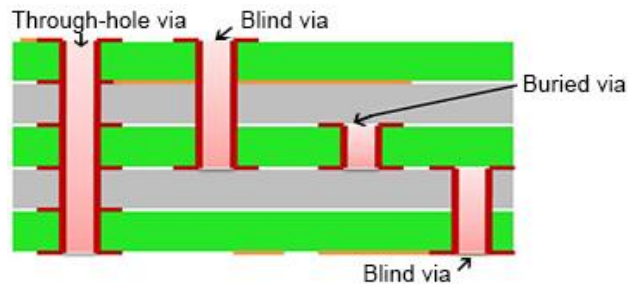
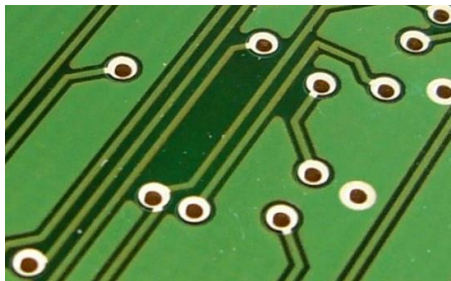


Figure 10. VIAs

Design rules: The PCB design has to abide to design rules specified by the PCB manufacturer (to address the fabrication process limitation). PCB layout tools usually have built-in **Design Rules Checker (DRC)**. DRC is software check of your design **to make sure the design does not contain errors** such as **traces that incorrectly touch, traces too skinny, or drill holes that are too small**. Example of a simplified set of design rules is given by Figure 11.

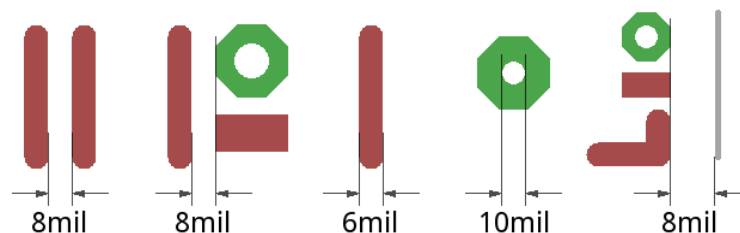


Figure 11. Design Rules Example

IV. Experiments

In experiment 1 and 2 we will design a PCB for a circuit that implements the block diagram of Figure 12, for the STM32F103C8 LQFP48 microcontroller along with its power input, st-link debug interface, clock oscillator, USB, and GPIO connections.

Note that:

- The down arrow in the figure is the ground signal.
- A **header** is just **an electrical connector comprised of multiple pins to solder to the PCB using one connection**.

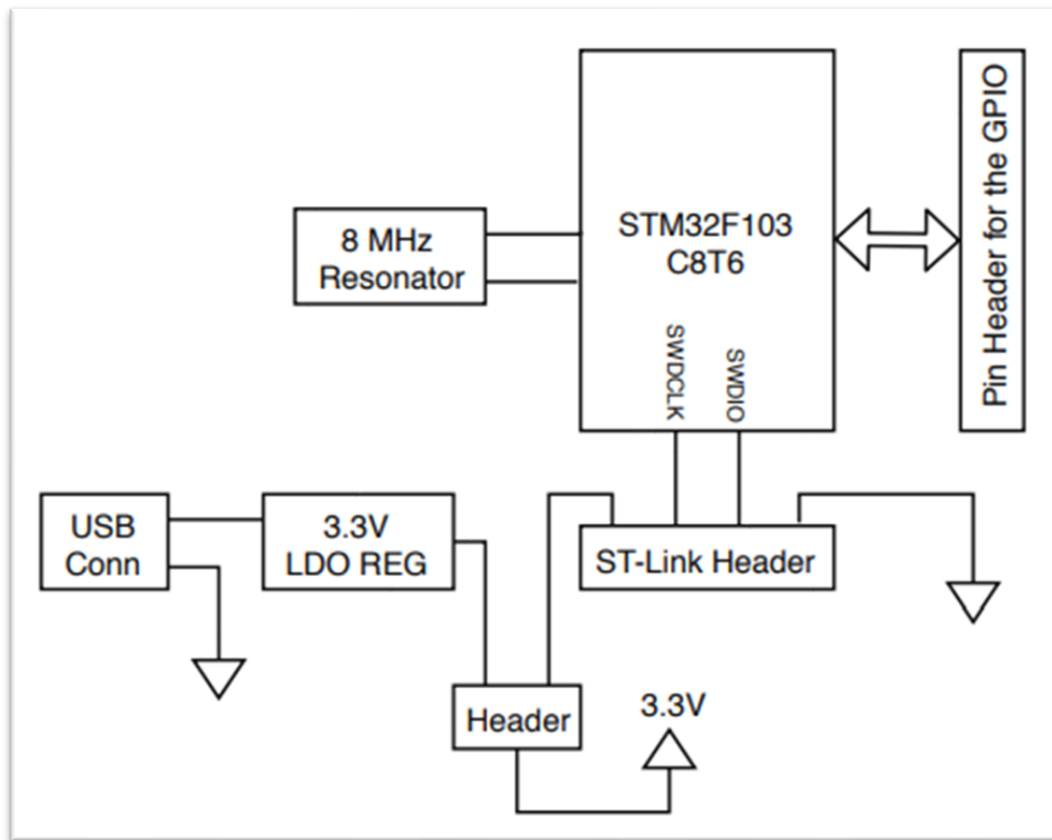


Figure 12. STM32F103C8 LQFP48 microcontroller board

Experiment 1:

Create a schematic for the STM32F103C8 LQFP48 board shown in Figure 12. You need to consult the microcontroller datasheet and application note [AN2586](#) provided on BB to create the schematic as the block diagram of Figure 12 does not show **all connections**. Few component libraries can be downloaded from BB to help with the PCB design.

Steps:

1. Download EAGLE from <https://www.autodesk.com/products/eagle/free-download>
2. Open EAGLE and add the libraries provided on BB:
Open **Options** -> **Directories** -> **Libraries**

Add a semicolon (;) following the default Libraries folder, then click **Browse** for the directory of your libraries to add them, then click **Ok** to close the window.

Right click on **Libraries** and click "Use all"

3. Check the Eagle schematic tutorial on <https://learn.sparkfun.com/tutorials/using-eagle-schematic> to understand how to create a schematic. Note that the tutorial does not implement our required design but will just be a guideline to follow for the schematic we need to create.
4. Now create a new project, then create a schematic in the opened project.
5. Add a frame to put your schematic in. Then add the power inputs V_{DD} and Gnd (or V_{SS}).
6. Add our STM32F103C8 microprocessor pinouts.
7. Check page 24 in *AN2586_getting started with stm32f10xxx hardware development.pdf* document (provided on BB) to understand the connections and supporting circuitry you need to add for your board. The reference design in that page is based on STM32F103ZET6 MCU but can be tailored to our board. Just change the pin numbers in diagram to match our 48-pin MCU, according to table 6 in page 25.
8. Wire up your schematic using the **NET tool** not the WIRE tool.
9. Name and label your net stubs.
Note: We can give a net a name rather than routing a wire all over the schematic. Nets with the same name are assumed to be connected even though there is no visible wire connecting them.
10. Add the necessary power supply connections as depicted in page 24 and in figure 2 page 8.
11. Add the reset circuit that connects to the NRST pin as depicted in page 24 and in figure 5 page 10.
Note: You can search for switches using SPST (Single Pole, Single Throw) and SPDT (Single Pole, Double Throw) keywords.
12. Add the 8MHz resonator circuit as depicted in page 24 and in figure 7 page 11.
13. Add the boot pins connections as depicted in page 24 and in figure 10 page 15.
14. Add a JTAG connector for the st-link header and connect it the board as depicted in page 24 and in table 2 page 18.
15. Add the USB connection and regulator. You may add a micro-b USB connection and connect its D+ and D- to PA12(USBDP) And PA11(USBDM) pins.
16. Continue adding any remaining supporting circuitry and headers. You may add one or more headers to have all GPIO pins soldered to the MCU in one or more connections.

Experiment 2:

Follow the following Eagle layout tutorial <https://learn.sparkfun.com/tutorials/using-eagle-board-layout> as a guideline to create your board layout from the schematic of experiment 1, place parts and route them, and to finally generate Gerber files that can be sent to a fab house to manufacture.

Below should be the pinout of the final design for the STM32F103C8 microcontroller board (Figure 13).

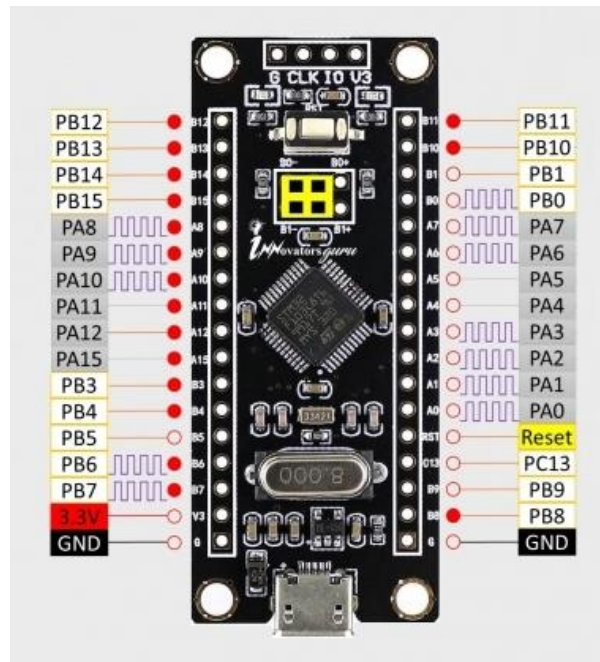


Figure 13. Pinout of STM32F103C8 board

V. References and Tutorials:

- STM32F103C8 Datasheet
- AN2586_getting started with stm32f10xxx hardware development.pdf
- PCB Basics: <https://learn.sparkfun.com/tutorials/pcb-basics/all>
- PCB Terminologies: <http://www.pcb.electrosoft-engineering.com/04-articles-custom-system-design-and-pcb/01-printed-circuit-board-concepts/printed-circuit-board-pcb-concepts.html>
- PCB Trace width calculator: <https://www.digikey.com/en/resources/conversion-calculators/conversion-calculator-pcb-trace-width>
- Eagle video tutorial: <https://www.youtube.com/watch?v=R4DYztYB6d4>
- Eagle schematic tutorial: <https://learn.sparkfun.com/tutorials/using-eagle-schematic>
- Eagle layout tutorial: <https://learn.sparkfun.com/tutorials/using-eagle-board-layout>

Lab Report [10 pts] (Individual submission)

1. [2 pts] Provide the project zipped folder of your experiment along with clear screenshots of your schematic and layout.
2. [8 pts] Create a schematic for our NUCLEO-L432KC board. You need to consult its user manual provided on BB in Lab 4 folder under the name "UM1956_STM32 Nucleo-32 boards user manual (dm00231744)". Create the schematic of the diagrams in page 32,33.

Provide the following:

- the project zipped folder of your experiment
- .lbr files of any external libraries used
- clear screenshots of the schematic
- 2-minute video explaining your design