10_PCB

Intro

The goal of the PCB is to create a cleaner, smaller version of the robot, to have a professional look. The entire design is created with the free version of Eagle (but could be ported on other CAD software such as KiCAD if improvement not included in the free version of Eagle are needed), with the free version of Eagle, the size of the board is limited (100mm by 80mm) as well as the number of layers (2 layers max).

Mini intro to Eagle

The following mini intro can only help you if you are new to Eagle PCB CAD, to get a more in-depth intro, please watch the following serie of video

Eagle is a CAD software used to create PCB out of schematics.

In the software, you can draw the schematic (high abstraction upon the final PCB design) and then "pick and place" the different components on the board design. Eagle is flexible enough to allow you to add components in the design even if you already placed all the other components. you can go back and forth between the schematic and the board design.

Once all your components are place, you can see so-called "airwires" which are basically "connections yet to be made". To make the connections, you can either connect them by hand with wires or yo can use the tool "autorouter". The easiest solution is t use the autorouter but you will have to check that everything is OK and maybe make a few changes.

One thing that is fundamental when working on the board design, is knowing on which physical layer of the PCB you are working on (for instance: top layer, bottom layer, vias, dimensions, etc).

Eagle comes with built-in scripts you can use to simplify you design, for example, adding a picture of a lobster in the PCB uses "import-bmp"

Current PCB Design

The current schematic of the board is essentially the current sensors, the rest of the board is only wiring.

Improvement on the current PCB design

On the current version of the PCB, a few things could be improved: * All the squared angles in signals wires should be changed to softer angles * The width of the "high" current wires could be enlarged. You can use this to calculate the width of the wires * A ground plane could be added at the back of the PCB