

PCB Design

Resources

Lecture 4 Slides: [Link](#)

EAGLE CAD: [Link](#)

EAGLE Tutorial: [Link](#)

EAGLE Library: [Link](#)

EAGLE Commands: [Link](#)

Parts Library: [Link](#)

EAGLE Part Creation Tutorial: [Link](#)

DRC Rules: [Link](#)

Overview

You will be designing a quadcopter control PCB including an MCU and motor drivers. By designing a custom PCB, you can ensure that you have all the necessary components in a compact layout, which is especially important for a space-sensitive application such as this one. Now that you have finished your parts selection and schematic design, the next step is to lay out the PCB itself.

A. Check Component Footprints

If you downloaded any Eagle libraries from online or created your own components, you should double check the physical footprint against the recommended land pattern in the datasheet. If the footprint is incorrect you will not be able to assemble your board, so you should verify before beginning your board layout. If you do not know how to view the part footprint, read the [part creation tutorial](#).

B. Board Dimensions

All our quadcopters will be using the same frames, so it is very important that your board has the correct physical dimensions. Specifically, there are 3 requirements:

1. 47mm x 28mm total board size
2. 4 rectangular cutouts, 3mm x 2mm. These cutouts should be centered at (3, 1), (44, 1), (3, 27), and (44, 27), with (0, 0) being the lower left corner of the board.
3. The board can only have two copper layers (top and bottom).

C. Board Layout

After creating the board dimensions, you must route all the traces for your board, as covered in [lecture 4](#). When designing your board, there are a few guidelines you should keep in mind:

1. Focus on part placement. If you can find the right way to arrange parts on your board, it will be very easy to route all your traces well. If your parts are not arranged well, it will be impossible to create good traces.
2. Always be thinking about parasitic components and how to reduce their effect (e.g. proper trace thickness). If you ignore parasitics, your board will not work well, even if all the components are connected properly.
3. Don't assume you'll get it right on your first try. You will probably need several revisions before getting a high-quality design, and your final layout will probably be very different than what you started with.

D. Board Submission

After you think you have a good design, upload your board and schematic files to your team's folder, and ask us to review it. Before asking for a board review, run a DRC check using Osh Park's [DRC rules](#): if your board has any errors or warnings, you must fix them, otherwise your board will not be manufactured correctly. DO NOT ask us to review a board that fails DRC, or which has incorrect dimensions (as detailed in the **Board Dimensions** section). If you ask us to review such a board, we will simply tell you to fix the DRC errors/warnings or the board dimensions, without looking at the rest of your design.