Schematic Design

Resources

Lecture 3 Slides: **Link**
EAGLE CAD: **Link**
EAGLE Tutorial: **Link**
EAGLE Library: **Link**
EAGLE Commands: **Link**
Parts Library: **Link**
EAGLE Part Creation Tutorial: **Link**
Digikey: **Link**
Mouser: **Link**

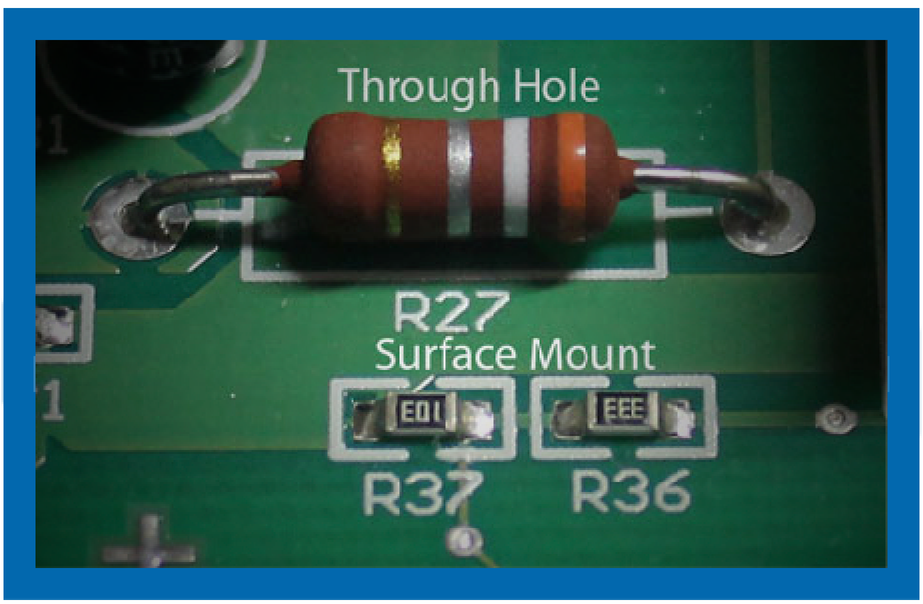
Overview

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. The first step is to create a schematic to lay out all the necessary parts and connections.

A. Part Selection

Last quarter, we covered the different parts needed for making your quadcopter (e.g. microcontroller, voltage regulator), but now you need to decide which specific ones you will be using. We will require all teams to use the same radio module, IMU, and motors as we did during the fall, and we highly recommend you use the Atmega328p MCU (the microcontroller found on the Arduino Pro Mini), but other than that you are free to choose any parts you desire (within reason). For your voltage regulator we recommend you use either a buck-boost converter or a charge-pump regulator: our battery voltage is 3.7V, but can drop below 3.3V when under heavy load (i.e. high motor current), so you will need a regulator that can both decrease and increase voltage. The two main websites for finding parts are [Digikey]() and [Mouser](): we suggest going there first for finding parts. You should record all parts you use in your bill of materials, in your team folder we shared with you.

When choosing parts, make sure to take into account all the parameters mentioned in [the previous lecture](): for example, you should check the on-resistance for MOSFETs. Aside from those parameters, you also need to take the package into account: you want parts that are small, but also relatively easy to solder. The two main categories for device packages are through hole and surface mount (SMD). So far all the components we have used have been through hole: they have pins that go through a hole in your PCB (or in your breadboard). Through hole components are good for prototyping, but they also take up a large amount of space on your PCB, so we will use SMD parts for our quadcopters. SMD parts are soldered on top of your board, without requiring any holes in your PCB. SMD parts are also usually much smaller, as seen in this image of through hole and surface mount resistors.



You should use SMD for all your components, except for connectors (e.g. JST for the motors), or components you cannot find in SMD packages.

Passive components (e.g. resistors) usually come on rectangle packages (like in the picture), and the package name tells you their dimensions. For example, an 0603 capacitor is 0.06in x 0.03in. In general it is best to use 0603 passives in your designs: smaller components are hard to solder by hand, while larger components waste space on your board. The only exception is when a part is not available in 0603: for example, very high capacitance capacitors are usually not available in 0603.

B. Schematic

After choosing your parts, you must create a schematic in EAGLE including all your components and the circuits connecting them. Most of the necessary circuits needed for your quadcopter have already been covered in the labs: for example we covered the motor driver circuit in lab 3. More complex components such as microcontrollers and voltage regulators usually require external components such as decoupling capacitors; you can usually find this information in the “Application Information” section of the datasheet.

On top of the components mentioned in lecture, we suggest you add a few other circuits to your schematic. First, you should add a voltage divider that allows you to read your battery voltage through an analog input pin. Second, you should add a 100uF capacitor in parallel across your battery, to help provide current to the motors. This capacitor will most likely be larger than 0603, as 100uF is a very large capacitance. Third, as mentioned in the slides, your digital circuits will need decoupling capacitors. If you are using the Atmega328p, you will need three 100nF capacitors: one for AV_{cc}, for V_{cc}, and for AREF. If you are using another MCU, check the datasheet for recommended decoupling capacitors. The radio modules needs 10uF and 100nF decoupling capacitors.

In order to upload code, you will need to add female headers for the FTDI programmers. You also need a proper reset circuit. The reset pin is active low, so you want a button than connects reset to ground and a pull-up resistor that pulls reset high when the button is not pressed. The DTR pin on the FTDI programmer is also used for resetting the microcontroller, but the programmer will pull it low for a prolonged period of time while we only want a short pulse. To accomplish this, you put a 100nF capacitor between DTR and the reset pin: pulling DTR low will temporarily set reset low, but then the capacitor will charge up through the pull-up resistor, setting reset high again. Finally, you should add a few SMD LEDs as indicators.

In order to add a component to your schematic in EAGLE, you will need an EAGLE device which corresponds to it. Here is a library containing many useful EAGLE devices, which you can install by unzipping it and placing it in the lbr subdirectory of your EAGLE folder. If you are using a part not in this library, you will have to either find a library for it online or make the component in EAGLE yourself. When using parts you find online, make sure to double check the footprint against the recommended footprint in the datasheet: if the footprint is wrong you will not be able to assemble your quadcopter. If you choose to make your own component, follow the tutorial here.

C. Design Review

After finishing your bill of materials and schematic, your team must go through a design review. During your design review, you will explain your parts choices and schematic to us, and we will look for any flaws or improvements that could be made. Please prepare a presentation for your design review, walking through the different subcircuits in your schematic and explaining why you chose certain components. Your team must pass a design review before you can continue with building your quadcopter.