

Lab 03: PCB Layout

You will be creating a PCB from the schematic that you captured last week.

- Assign footprints to all parts in your schematic.
 - You can choose 0805 SMD packages for your resistors and capacitors and other passive components, except for C1 and C2, which are decoupling capacitors that should be *polarized* THT.
 - Your voltage regulator (U1 : L7805) should use a THT TO220 package.
 - Your IC DIPs should be SMD.¹
 - Your SPST SW and BATT should use a Pin Socket : 1x02, P2.54mm, Vertical to allow external wires to be attached to your PCB.
- Your board can be two-sided.
- In addition to your Default Net Class, create a Power Net Class that only differs from the Default specifications by having twice the trace width and clearance.²
- Optimize the size of your board to be as small as possible in your layout (as outlined with Edge.Cuts).
- Place your mounting holes on each corner of the board, no closer than 2 mm to any edge.
- Keep decoupling capacitors as close to their supported voltage pins as possible.
- Remember that THT components should be on the opposite side of the board from their traces (in contrast to SMD components).
- Use filled ground pours where possible.
- Make sure that your schematic passes the Design Rules Check (DRC).
- Submit a combined PDF of the following:
 - Front layer of board.
 - Back layer of board.
 - Screenshot of passed DRC.

¹Hint: the 556 IC is a 14-pin DIP.

²It is up to you to choose which nets should be assigned to which net classes.