VICTORIA UNIVERSITY OF WELLINGTON
TE HERENGA WAKA
NEW ZEALAND

Demo 4 – PCB Design

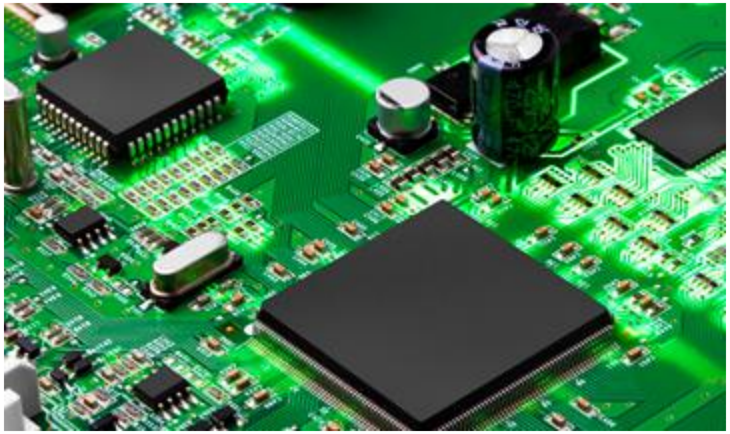
XMUT204 Electronic Design

Overview

1. Introduction to PCB.
2. Basic PCB design process.
3. PCB design best practices.
4. Designing schematic of the circuit.
5. Designing PCB layout.
6. Design project documentation.

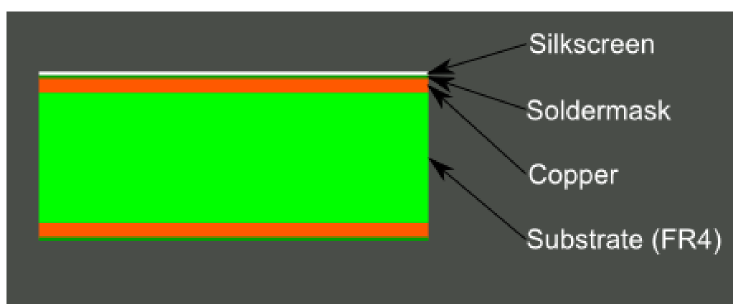
Introduction

- PCB is an acronym for printed circuit board. It is a board that has lines and pads that connect various connectors and components to each other.
- A PCB allows signals and power to be routed between physical devices.
- Solder is the metal that makes the electrical connections between the surface of the PCB and the electronic components.
- Being metal, solder also serves as a strong mechanical adhesive.

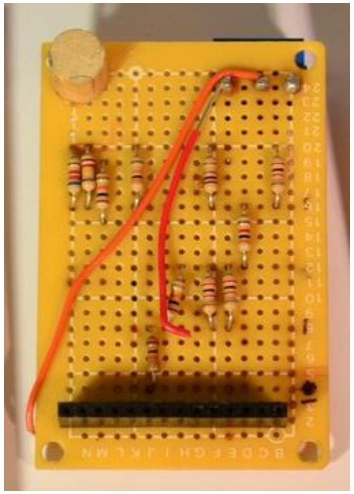


PCB Materials & Construction

- A PCB is sort of like a layer of sheets consisting of alternating layers of different materials laminated together with heat and adhesive such that the result is a single object.
- In terms of its materials, typically PCB is made of layers of:
    - substrate (FR4),
    - copper,
    - soldermask, and
    - silkscreen.



Substrate (FR4) Layer



- The base material, or substrate, is usually fiberglass. Historically, this fiberglass is "FR4".
- Provide the PCB its rigidity and thickness. There are also flexible PCBs built on flexible high-temperature plastic (Kapton or the equivalent).
- Many different thickness PCBs and the most common thickness for PCBs is 1.6 mm (0.063").
- Some of PCBs for digital circuits are typically available as a 0.8 mm thick board.

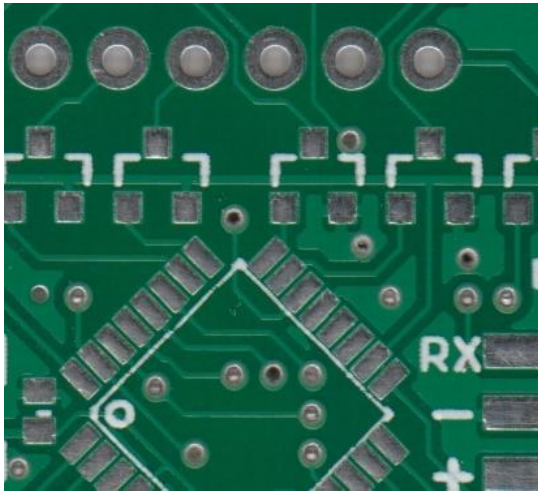
- Cheaper PCBs and perf boards i.e. made with other materials such as epoxies or phenolics which lack the durability of FR4 but less expensive.
- These types of substrates are also typically found in low-end consumer electronics.
- Phenolics have a low thermal decomposition temperature -> delaminate, smoke and char when the soldering iron is held too long on the board.

Copper Layer

- Thin copper foil laminated to the board with heat and adhesive.
- For typical double-sided PCBs, copper is applied to both sides of the substrate. In lower cost electronic gadget the PCB may have copper on only one side.
- When we refer to a double sided or 2-layer board for a typical PCB. Can be as few as 1 layer or as many as 16 layers or more.
- Figure below shows PCB with copper exposed, no solder mask or silkscreen.
- The copper thickness can vary and is specified by weight, in ounces per square foot.
- The vast majority of PCBs have 1 ounce of copper per square foot.
- High power PCB may use 2 or 3 ounce copper.
- Each ounce per square foot = 35 mm or 1.4 thousandths of an inch of thickness of copper.

Soldermask Layer

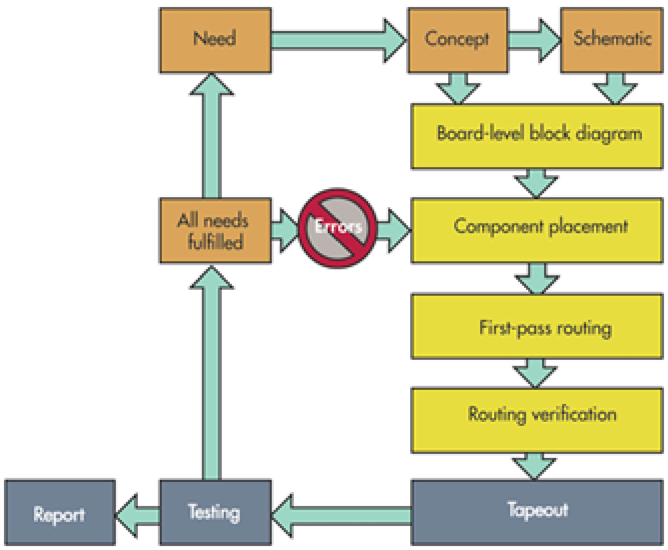
- The layer on top of the copper foil.
- Give the PCB its green or red colour. PCBs are most commonly green in colour but nearly any colour is possible.
- It is overlaid onto the copper layer to insulate the copper traces from accidental contact with other metal, solder, or conductive bits.
- This layer helps the user to solder to the correct places and prevent solder jumpers.
- In the example below, the green solder mask is applied to the majority of the PCB, covering up the small traces but leaving the silver rings and SMD pads exposed so they can be soldered to.



Silkscreen Layer

- It is typically applied on top of the solder mask layer.
- Adds letters, numbers, and symbols to the PCB that allow for easier assembly and indicators for humans to better understand the board.
- Often silkscreen labels are used to indicate what the function of each pin of IC or other components.
- Silkscreen is most commonly white, but any ink colour can be used i.e. black, grey, red, and even yellow.
- It is, however, uncommon to see more than one colour on a single board.

The Basic PCB Design Process



- Ideal PCB design starts with the need for its design and construction and continues through the final production boards as shown in the figure below.
- After determining why the PCB is needed, the product's final concept should be decided.
- The concept includes the design's features, the functions the PCB must have and perform, interconnection with other circuits, placement, and the approximate final dimensions.
- The ideal PCB design flow begins when designers recognize a need that must be fulfilled i.e. the PCB is for mass production in case of consumer product or for manufacturing purpose if it is intended for specific client requests, and it does not end until testing verifies that the design can meet those needs.

The Basic PCB Design Process

- Ambient temperature range and concerns regarding the operating environment should be addressed and used to specify the materials selected for the PCB.
- Components and PCB materials must be selected to guarantee operation under all expected and potential forms of duress the board may be exposed to during its lifetime.
- The circuit schematic is drawn based on the concept that meets the need for designing the PCB. The schematic shows the electrical implementation of each function of the PCB.
- With the schematic drawn, a realistic drawing of the final PCB dimensions should be completed with areas designated for each of the circuit's schematic blocks (groups of components closely connected for electrical reasons or constraints).

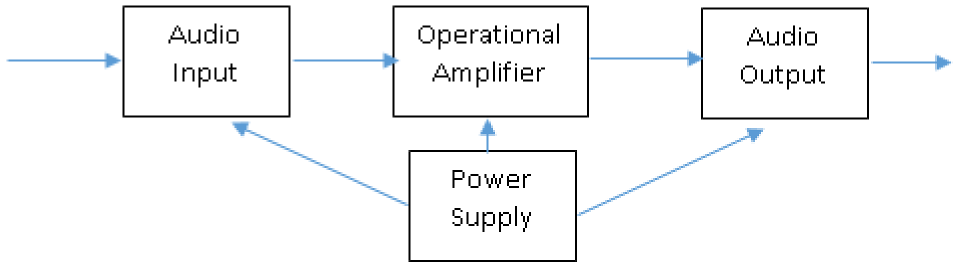
PCB Design Best Practices

From what we have covered so far, this section outlines some of the practical tips for enhancing the design of PCB process.

- Component arrangement
- PCB shape and size.
- User experience and interfaces.
- PCB layers.
- Ground layers.
- Layer thickness.
- PCB traces.
- Trace width.

Component Arrangements

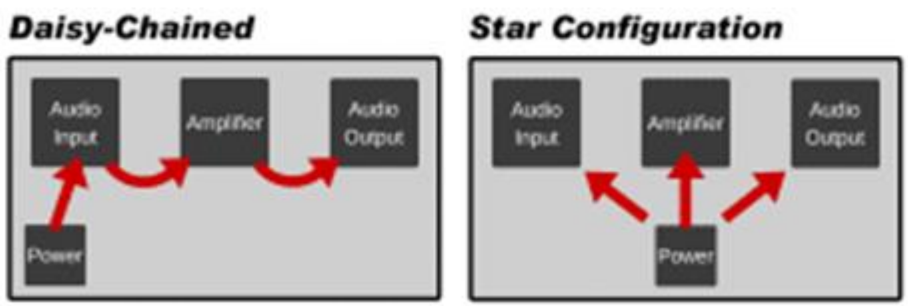
- Identify what each part of your circuit does, and divide the circuit into sections according to function.
- For example, an audio amplifier circuit has four main sections: a power supply, an audio input, the op amp, and an audio output. It might help to draw some diagrams at this point to help you visualize the design before you start laying it out.



- Keep the components in each section grouped together in the same area of the PCB to keep the conductive traces short. Long traces can pick up electromagnetic radiation from other sources, which can cause interference and noise.

Component Arrangements

- The different sections of your circuit should be arranged so the path of electrical current is as linear as possible. The signals in your circuit should flow in a direct path from one section to another, which will keep the traces shorter.
- Each section of the circuit should be supplied power with separate traces of equal length. This is called a star configuration, and it ensures that each section gets an equal supply voltage.



- If sections are connected in a daisy-chain configuration, the current drawn from sections closer to the supply will create a voltage drop and result in lower voltages at sections further from the supply.

PCB Shape and Size

- It is not uncommon to see round, triangular, or other interesting PCB shapes. Most PCBs are designed to be as small as possible, but that is not necessary if your application does not require it.

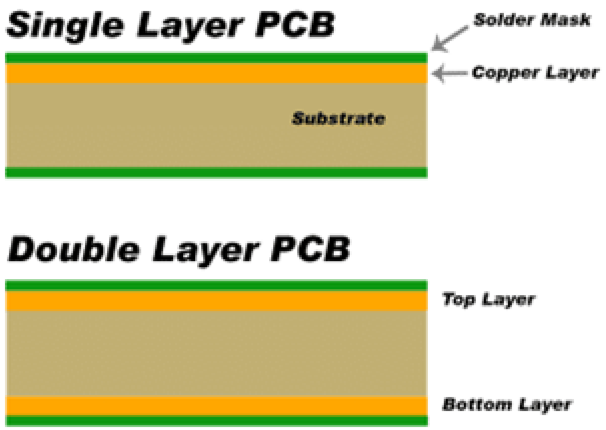
- If you plan on putting the PCB into an enclosure, the dimensions may be limited by the size of the housing. In that case, you will need to know the enclosure's dimensions before laying out the PCB so that everything fits inside.
- The components you use will also have an effect on the size of the finished PCB. For instance, surface mounted components are small and have a low profile, so you will be able to make the PCB smaller. Through-hole components are larger, but they are often easier to find and easier to solder.

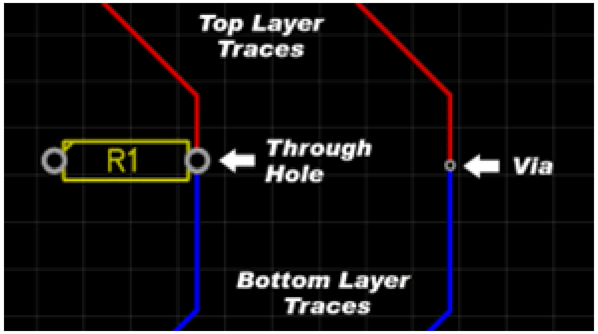
User Experience & Interfaces

- The location of components like power connections, potentiometers, LEDs, and audio jacks in your finished project i.e. an audio amplifier circuit will affect how your PCB is laid out.
- Do you need an LED near a power switch to indicate that it is on?
- Do you need to put a volume potentiometer next to a gain potentiometer?
- For the best user experience you might have to make some compromises and design the rest of your PCB around the locations of these components.

PCB Layers

- Larger circuits can be difficult to design on a single layer PCB because it is hard to route the traces without intersecting one another. You might need to use two copper layers, with traces routed on both sides of the PCB.

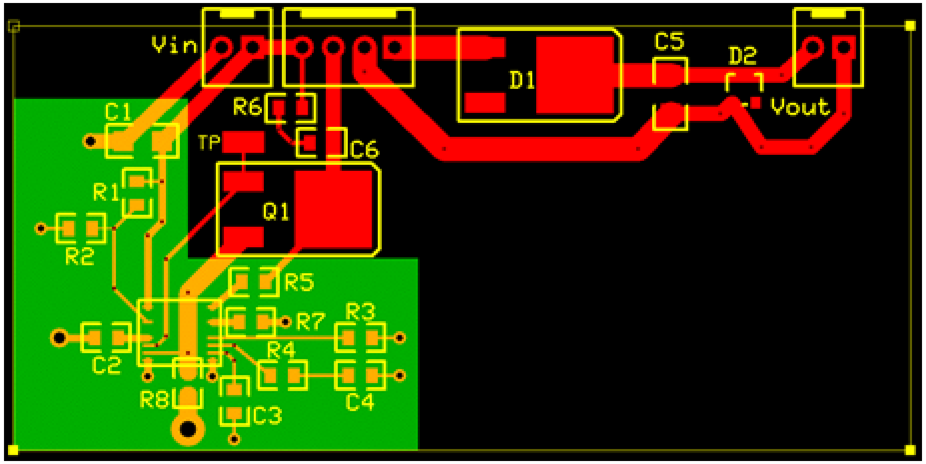




- The traces on one layer can be connected to the other layer with a via (left figure above). A via is a copper plated hole in the PCB that electrically connects the top layer to the bottom layer.
- You can also connect top and bottom traces at a component's through hole.

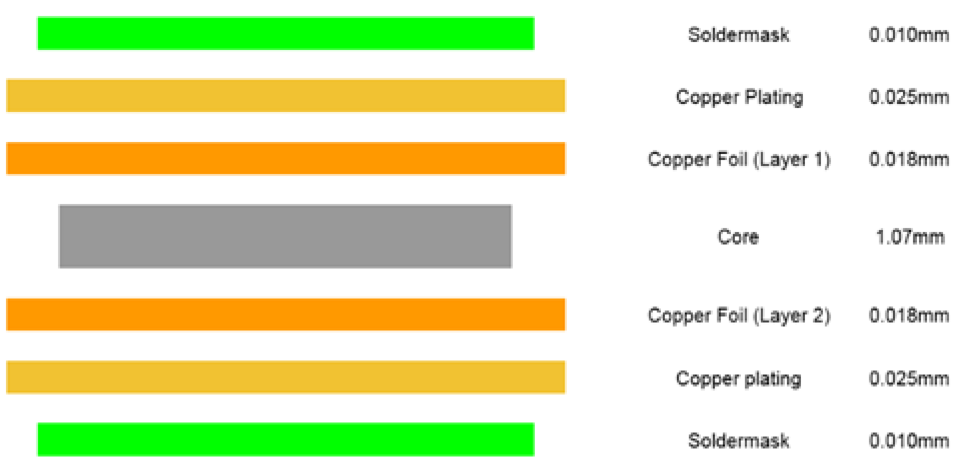
Ground Layers

- Some double layer PCBs have a ground layer, where the entire bottom layer is covered with a copper plane connected to ground. The positive traces are routed on top and connections to ground are made with through holes or vias.
- Ground layers are good for circuits that are prone to interference, because the large area of copper acts as a shield against electromagnetic fields. They also help dissipate the heat generated by the components.



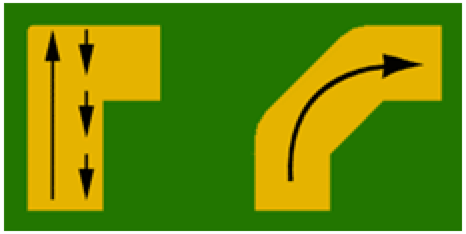
Layer Thickness

- Most PCB manufacturers will let you order different layer thicknesses. Copper weight is the term manufacturers use to describe the layer thickness, and it is measured in ounces. The thickness of a layer will affect how much current can flow through the circuit without damaging the traces.
- To determine safe values for thickness, you need to know the current (amperage) that will flow through the trace in question.
- Trace width is another factor that affects how much current can safely flow through the circuit (discussed as follows).



PCB Traces

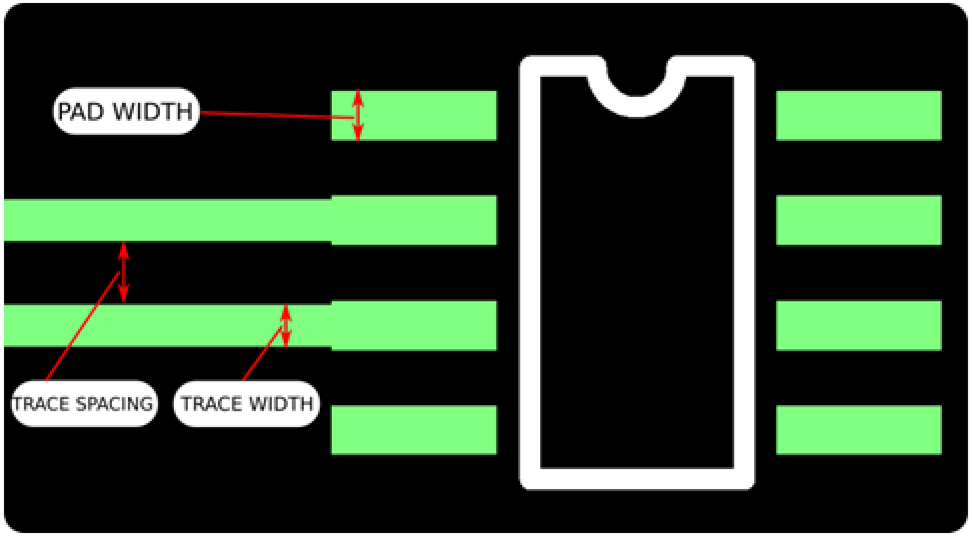
- If you look at a professionally designed PCB, you will probably notice that most of the copper traces bend at 45° angles. One reason for this is that 45° angles shorten the electrical path between components compared to 90° angles. Another reason is that high-speed logic signals can get reflected off the back of the angle, causing interference.



- If your project uses digital logic or high-speed communication protocols above 200 MHz, you should probably avoid 90° angles and vias in your traces. For slower speed circuits, 90° traces would not have much of an effect on the performance of your circuit.

Trace Width

- Like layer thickness, the width of your traces will affect how much current can flow through your circuit without damaging the circuit.
- The proximity of traces to components and adjacent traces will also determine how wide your traces can be.
- If you are designing a small PCB with lots of traces and components, you might need to make the traces narrow for everything to fit.

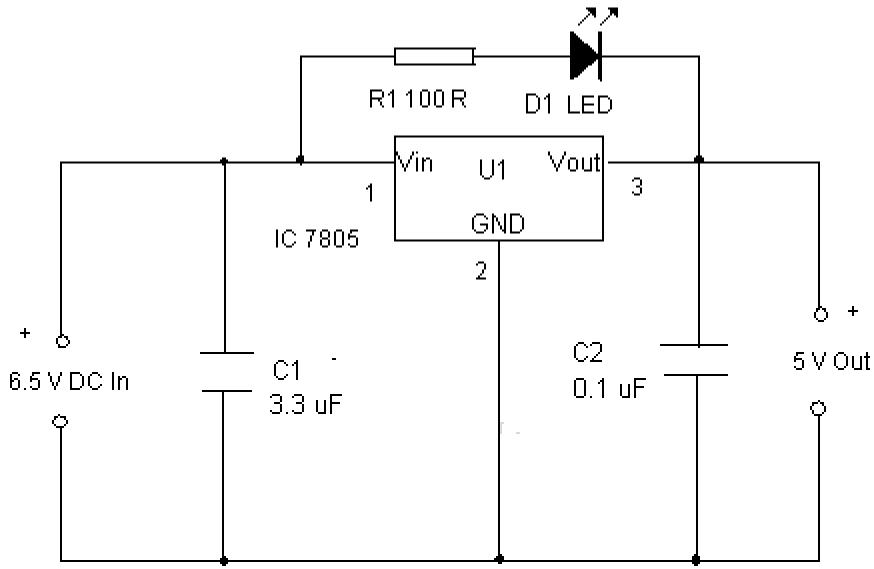


Designing Schematic of the Circuit

- For this exercise, we are planning to design the schematic and PCB layout of a simple linear voltage regulator circuit using a free electronic computer aided design (ECAD) software called EasyEDA.
- The software is available in Windows, MacOS and Linux versions at the VUW wiki website and it is also downloadable from its manufacturer website: https://easyeda.com/).

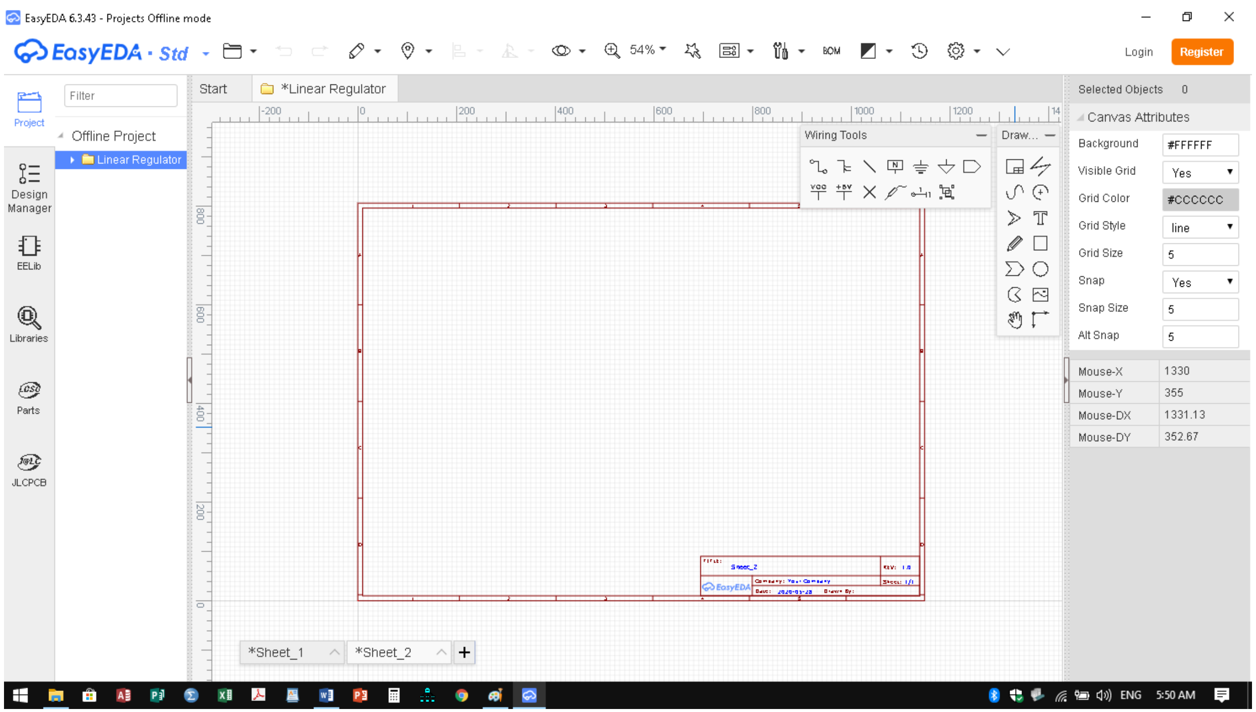
Designing Schematic of the Circuit

Once you install the software in your computer and chose the installation as an offline project, you can use the software for creating a schematic and a PCB layout of an example circuit given in the following figure (i.e. it is a linear voltage regulator based on the 7805 IC)



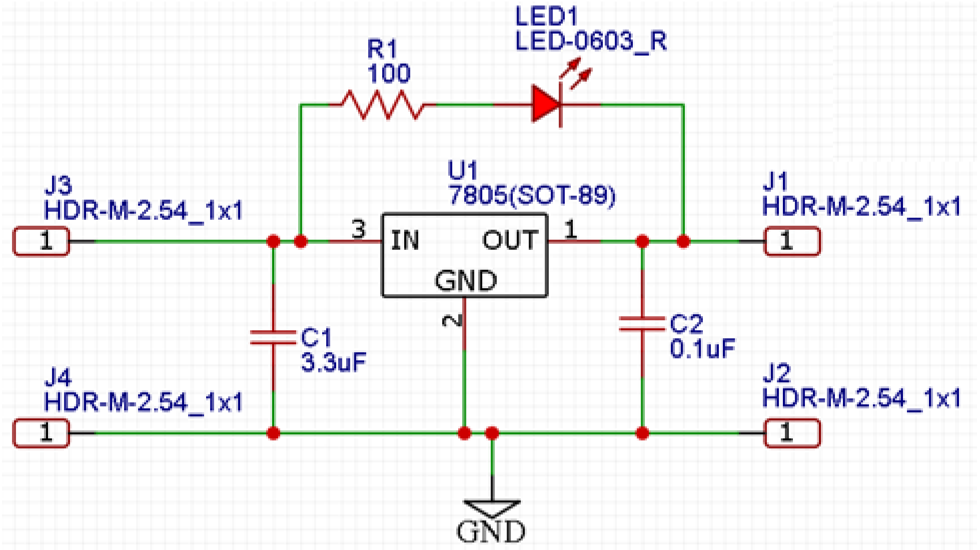
Designing Schematic of the Circuit

Create the schematic and a PCB layout of the voltage regulator circuit in EasyEDA software



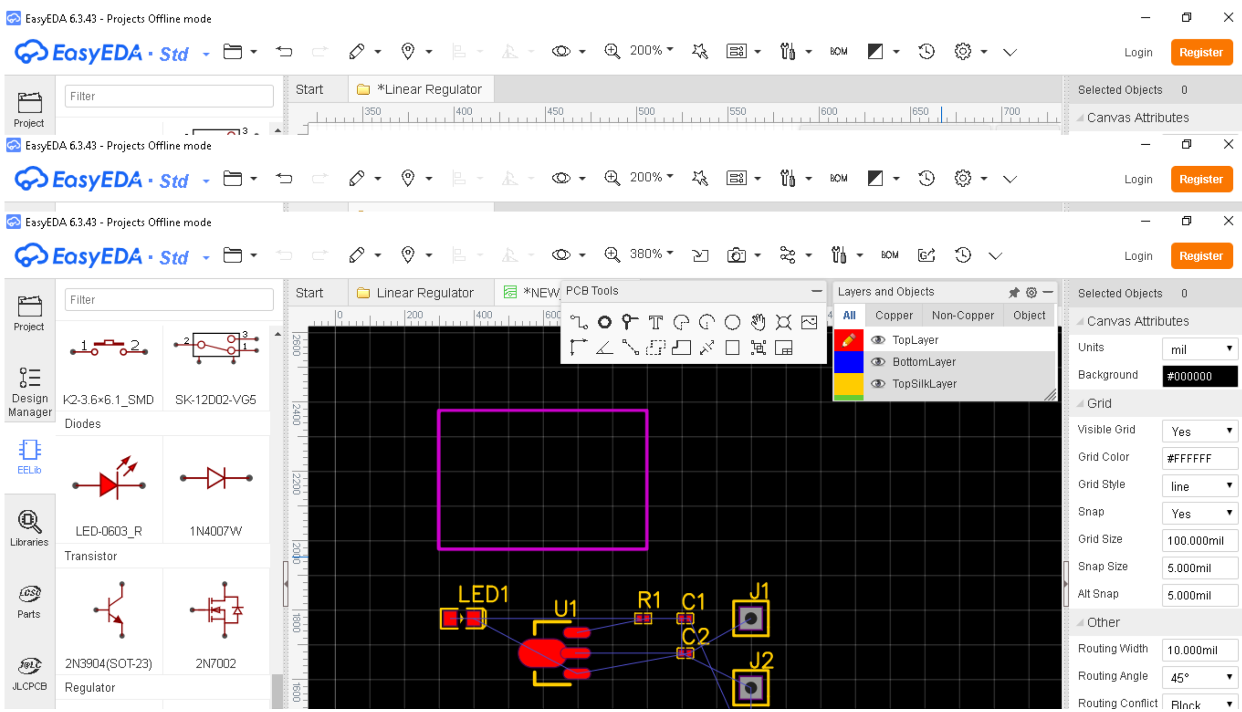
Designing Schematic of the Circuit

The resulting schematic diagram of the voltage regulator circuit.



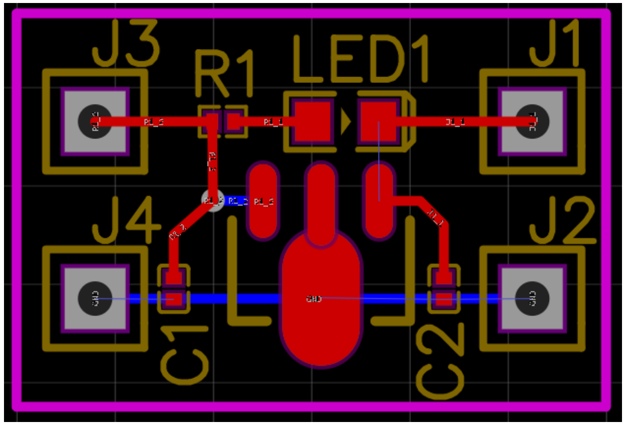
Designing PCB Layout of the Circuit

Create the PCB layout of the voltage regulator circuit in EasyEDA software

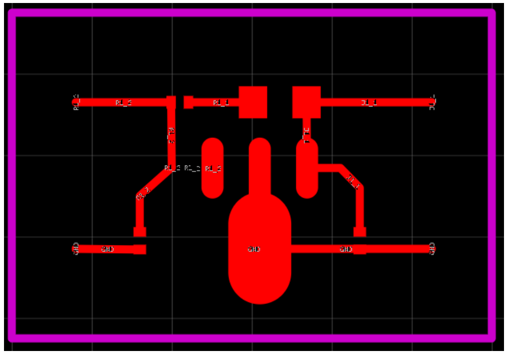


Designing PCB Layout of the Circuit

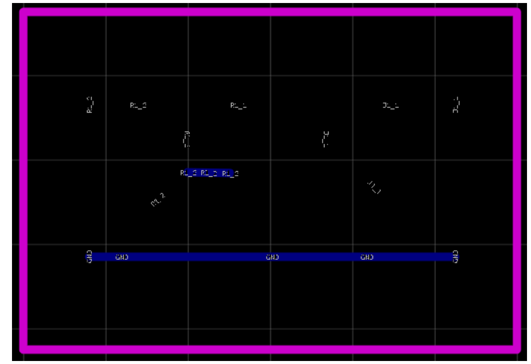
The resulting PCB layout diagram of the voltage regulator circuit.



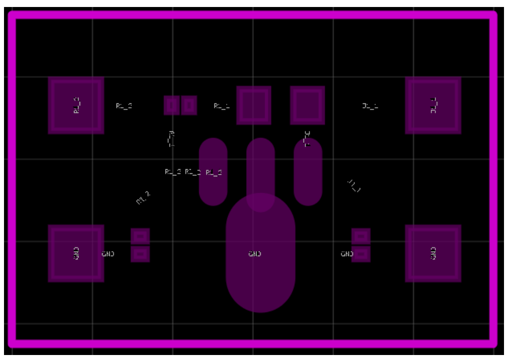
Designing PCB Layout of the Circuit



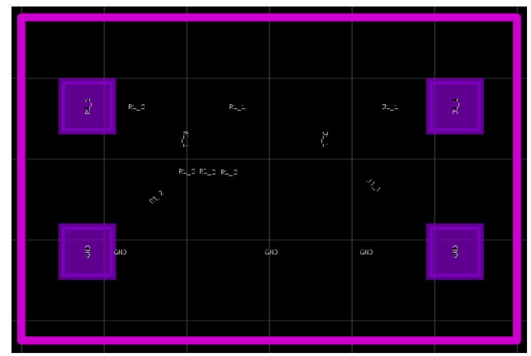
Top copper layer



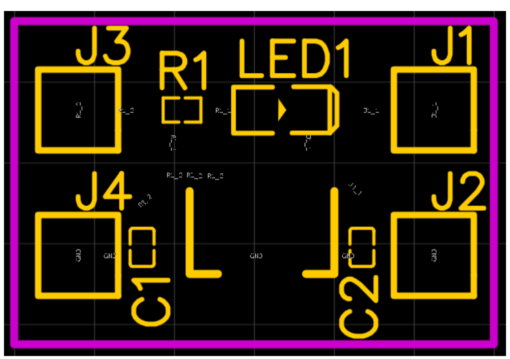
Bottom copper layer



Top soldermask layer



Bottom soldermask layer



Silkscreen (top) layer

Design Final Touches

- Add also PCB design texts to the PCB:
  - your name or team,
  - date,
  - version/revision,
  - etc.
  - These texts could be attached on the top or bottom copper layer or on the silkscreen layer.
- Carry out a design rule check. A design rule check will tell you if any components overlap or if traces are routed too close together.

Design Project Documentation (Bill of Materials)

The BOM requires the following information:

- the quantity,
- reference designators,
- value (numeric value of ohms, farads, etc.),
- manufacturer part number, and
- PCB footprint for each component.

Design Project Documentation (PCB Documentation)

The PCB's manufacturing documents should include:

- the hardware dimensional drawings,
- schematic,
- BOM,
- layout file,
- component placement file,
- assembly drawings and instructions, and
- Gerber file set.
- User guides also are useful but are typically not required.

The Gerber file set is a PCB jargon for the output files of the layout that are used by PCB manufacturers to create the PCB.

Design Project Documentation (Gerber Files)

A complete set of Gerber files includes output files generated from the board layout file:

- Silkscreen top and bottom.
- Solder mask top and bottom.
- All metal layers.
- Paste mask top and bottom.
- Component map (X-Y coordinates).
- Assembly drawing top and bottom.
- Drill file.
- Drill legend.
- FAB outline (dimensions, special features).
- Netlist file.