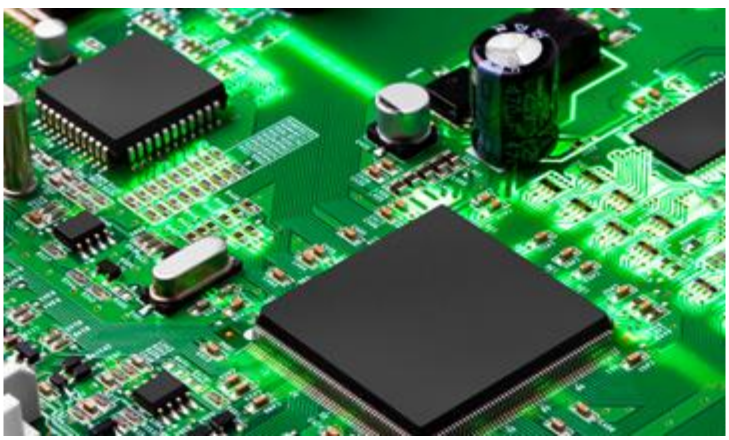
XMUT204 Electronic Design

Demo 4 - PCB Design

1. Introduction

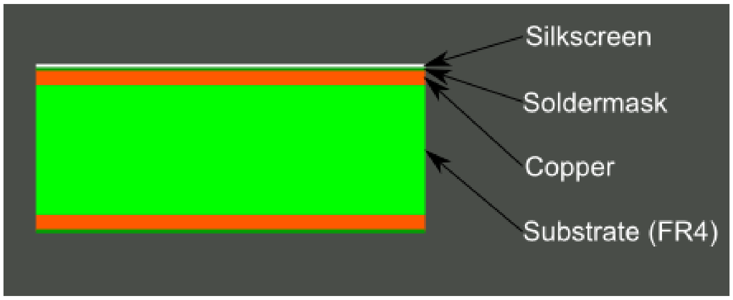
PCB is an acronym for printed circuit board. It is a board that has lines and pads that connect various points together. In the picture below, there are traces that electrically connect the various connectors and components to each other. A PCB allows signals and power to be routed between physical devices.



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.

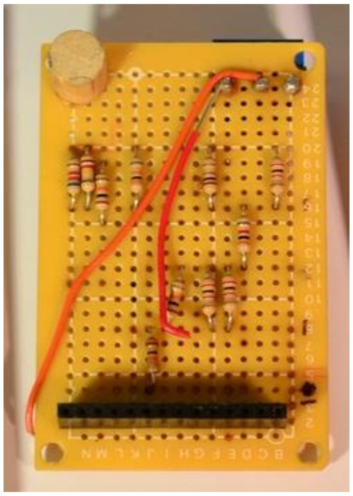
2. PCB Materials & Construction

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.



2.1. Substrate (FR4) Layer

The base material, or substrate, is usually fiberglass. Historically, the most common designator for this fiberglass is "FR4". This solid core gives the PCB its rigidity and thickness. There are also flexible PCBs built on flexible high-temperature plastic (Kapton or the equivalent).



You will find 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 (shown above) are made with other materials such as epoxies or phenolics which lack the durability of FR4 but are less expensive. You know that you have this type of PCB when you solder to it i.e. they have a very distinctive bad smell. These types of substrates are also typically found in low-end consumer electronics. Phenolics have a low thermal decomposition temperature which causes them to delaminate, smoke and char when the soldering iron is held too long on the board.

2.2. Copper Layer

The next layer is a thin copper foil, which is 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 we are referring to the number of copper layers (2) in typical given PCB. This can be as few as 1 layer or as many as 16 layers or more. The diagram 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. But, some PCBs that handle very high power may use 2 or 3 ounce copper. Each ounce per square foot translates to about 35 micrometers or 1.4 thousandths of an inch of thickness of copper.

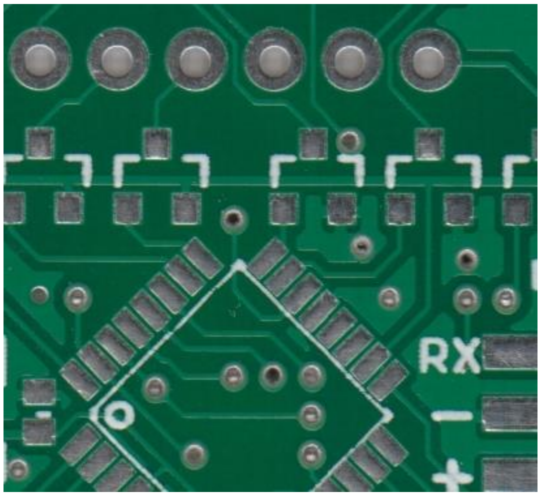
2.3. Soldermask Layer

The layer on top of the copper foil is called the soldermask layer. This layer gives 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.



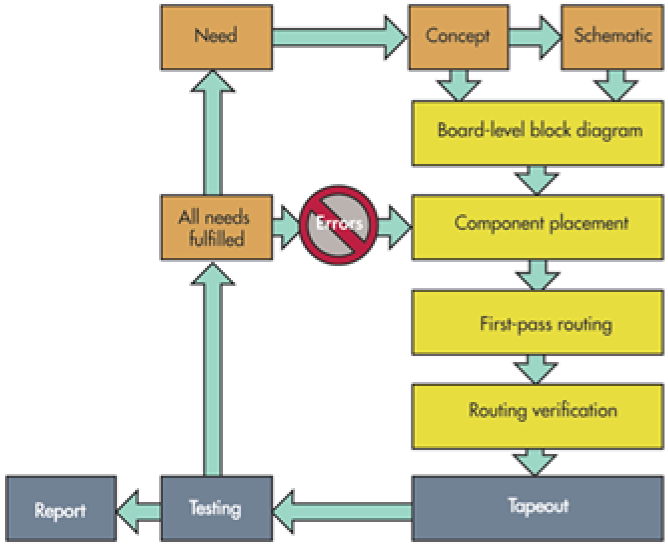
2.4. Silkscreen Layer

The white silkscreen layer is applied on top of the solder mask layer. The silkscreen 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. Black, grey, red, and even yellow silkscreen colours are widely available. It is, however, uncommon to see more than one colour on a single board.

3. The Basic PCB Design Process

The 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.

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).

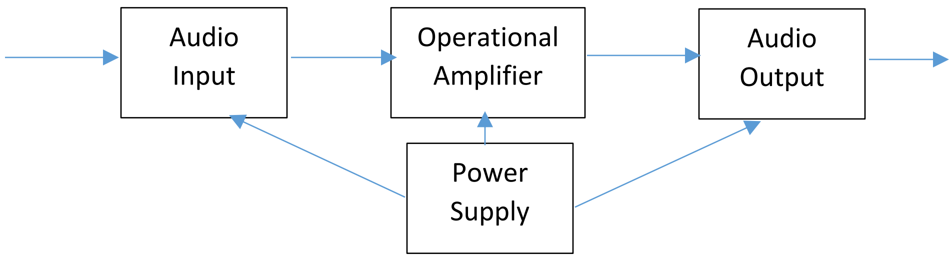
4. PCB Design Best Practices

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

4.1. Component Arrangements

Identify what each part of your circuit does, and divide the circuit into sections according to function.

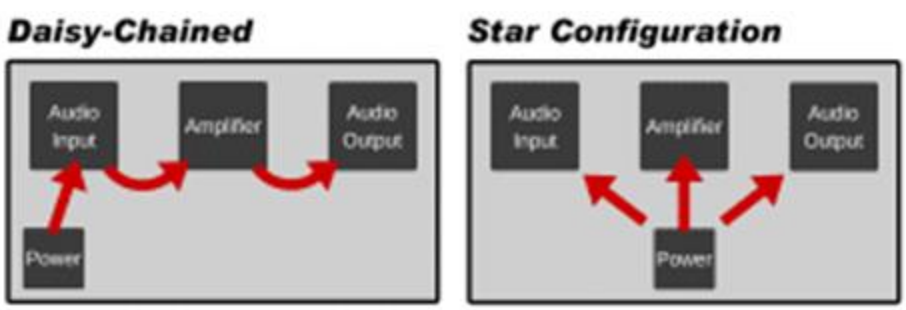
For example, an audio amplifier circuit has four main sections: a power supply, an audio input, the op amp, and an audio output. It might help to draw some diagrams at this point to help you visualize the design before you start laying it out.



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.

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.

4.2. 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.

4.3. User Experience & Interfaces

The location of components like power connections, potentiometers, LEDs, and audio jacks in your finished project will affect how your PCB is laid out.

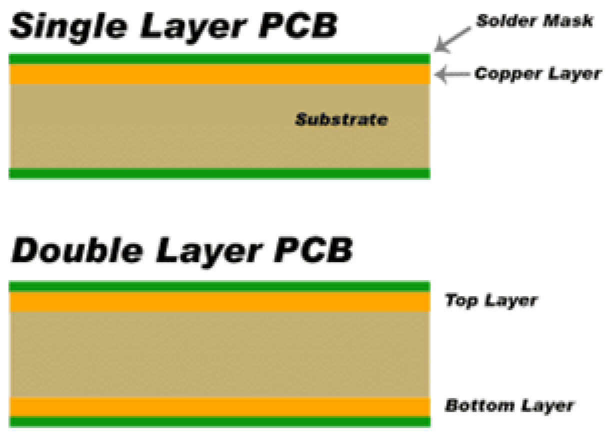
Do you need an LED near a power switch to indicate that it is on?

Do you need to put a volume potentiometer next to a gain potentiometer?

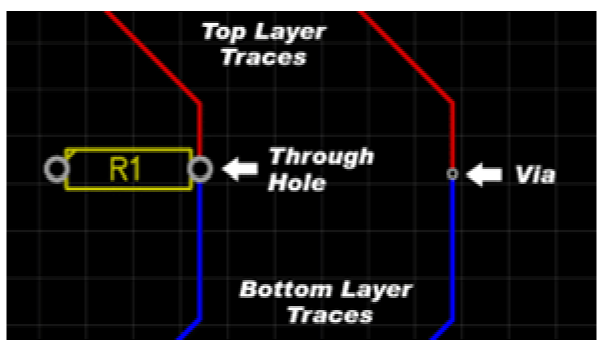
For the best user experience you might have to make some compromises and design the rest of your PCB around the locations of these components.

4.4. 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.

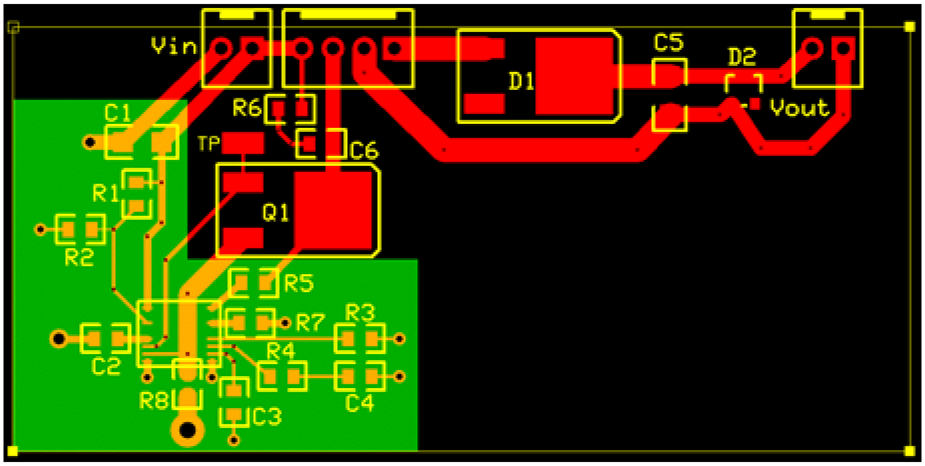


The traces on one layer can be connected to the other layer with a via. A via is a copper plated hole in the PCB that electrically connects the top layer to the bottom layer. You can also connect top and bottom traces at a component's through hole.



4.5. Ground Layers

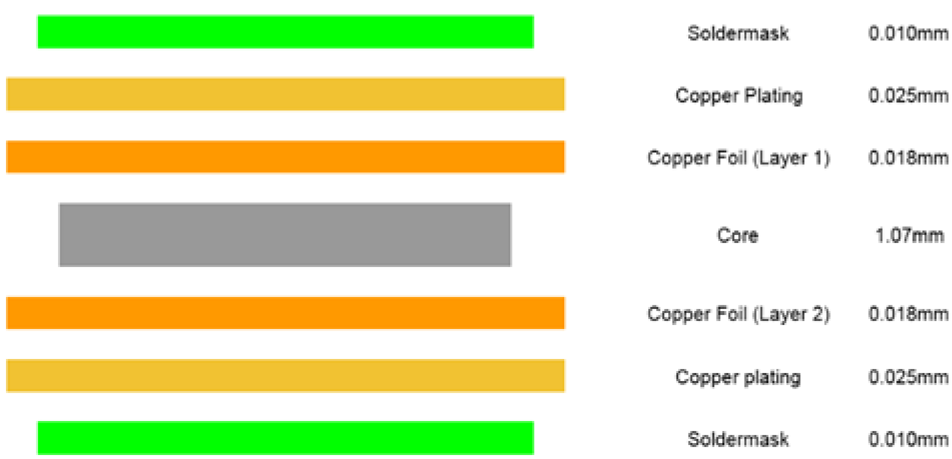
Some double layer PCBs have a ground layer, where the entire bottom layer is covered with a copper plane connected to ground. The positive traces are routed on top and connections to ground are made with through holes or vias.



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.

4.6. 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.

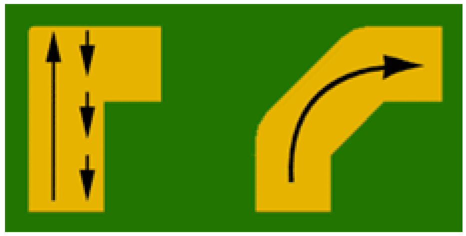


To determine safe values for thickness, you need to know the current (amperage) that will flow through the trace in question.

Trace width is another factor that affects how much current can safely flow through the circuit (discussed as follows).

4.7. PCB Traces

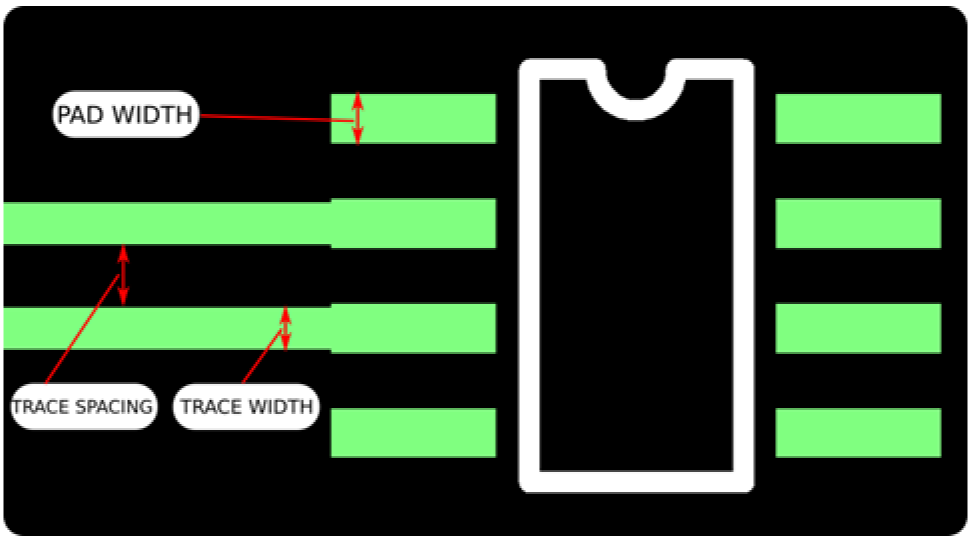
If you look at a professionally designed PCB, you will probably notice that most of the copper traces bend at 45° angles. One reason for this is that 45° angles shorten the electrical path between components compared to 90° angles. Another reason is that high-speed logic signals can get reflected off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

4.8. 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.

5. Designing Schematic of the Circuit

Before you start designing your PCB, it is a common practice to make a schematic of your circuit first. The schematic will serve as a blueprint for laying out the traces and placing the components on the PCB.

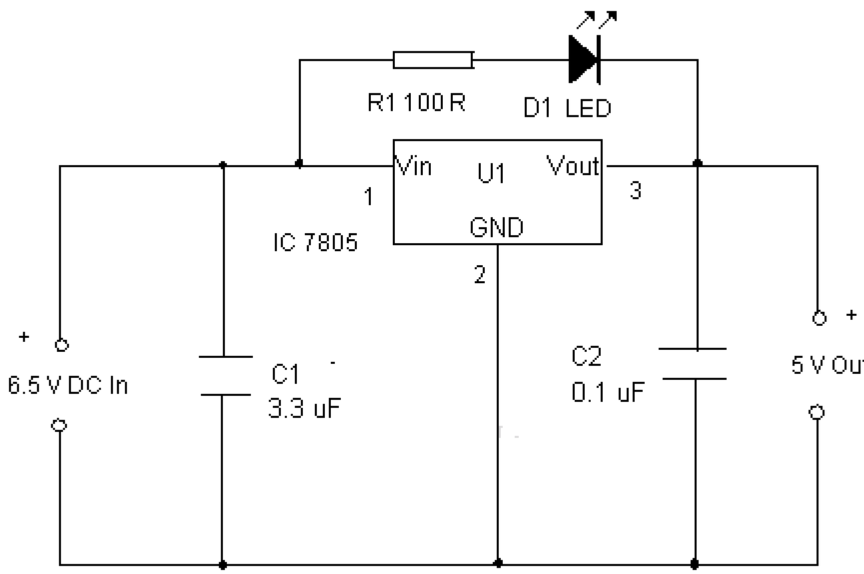
Plus, the PCB editing software can import all of the components, footprints, and wires into the PCB file, which will make the design process easier (more on this later).

Rather than using breadboard, if you do prototyping through simulation, then it is imperative to design the schematic of the circuit first.

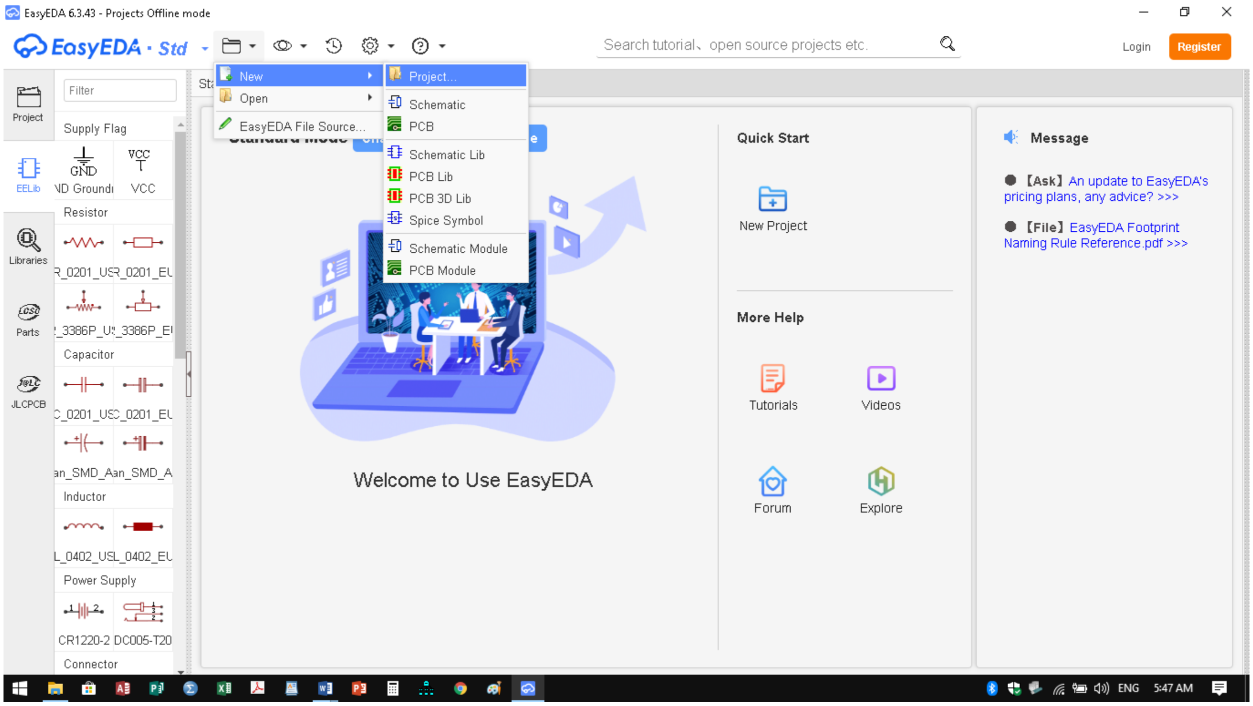
For this exercise, we are planning to design the schematic and PCB layout of a simple linear voltage regulator circuit as shown below using a free electronic computer aided design (ECAD) software called EasyEDA.

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/).

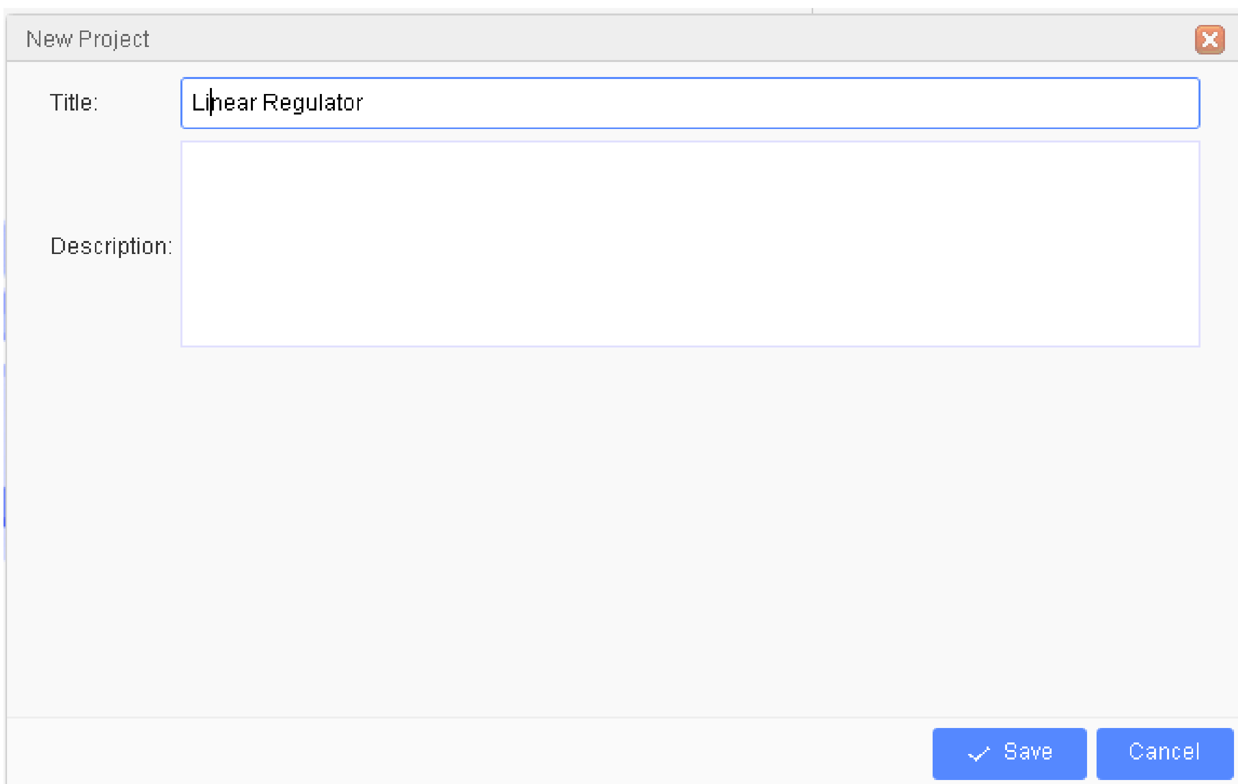
Once you install the software in your computer and chose the installation as an offline project, you can use the software for creating a schematic and a PCB layout of an example circuit given in the following figure (i.e. it is a linear voltage regulator based on the 7805 IC).



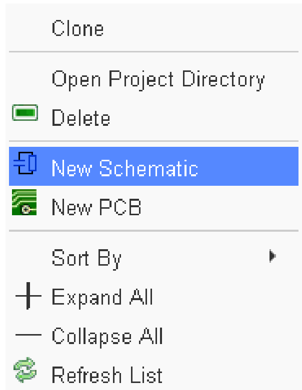
Start by logging into the EasyEDA and you will be shown with its integrated design environment (IDE) window.



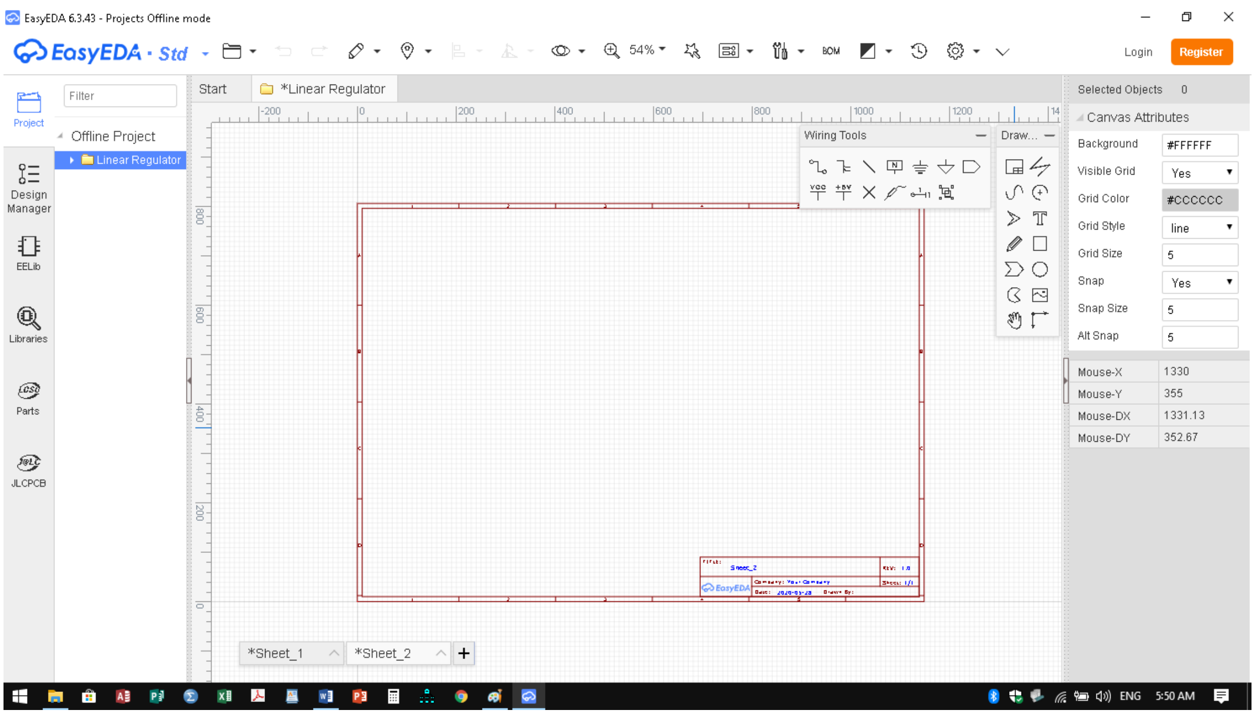
In the software create a new project and name it as “Linear Regulator” from folder menu in the toolbar.



From the folder menu in the toolbar or from the Project option on the left-hand side panel, click on the "New Schematic" option in the drop down menu.

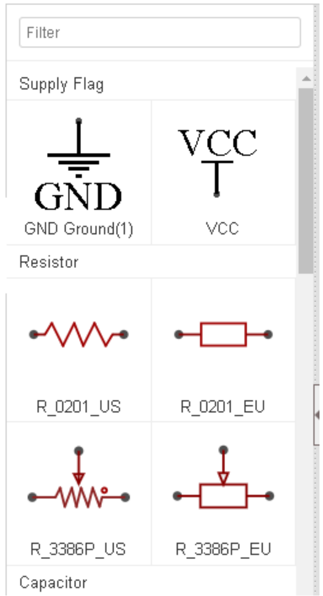


Now you will see a blank canvas where you can draw the schematic.

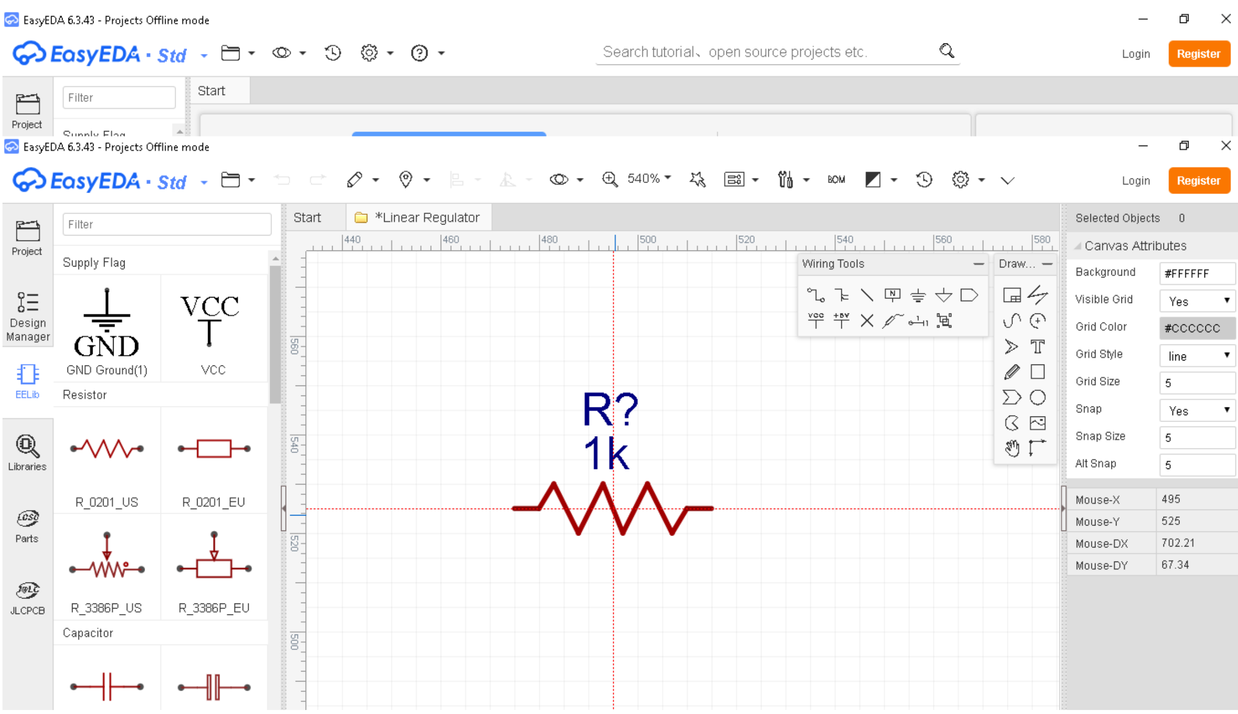


It is best to place all of your schematic symbols on the canvas before drawing any wires.

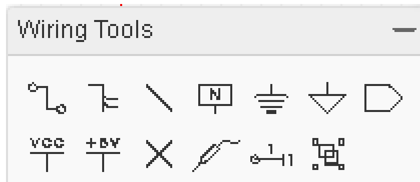
In EasyEDA, schematic symbols are located in "Libraries". The default library (i.e. EElib) has most of the common symbols, but there are also other third party libraries from a few sponsoring electronic design and manufacturing companies with lots of other symbols.

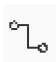


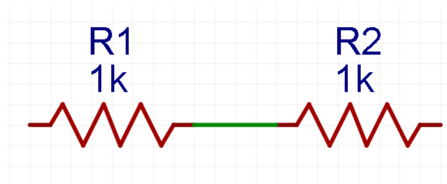
Each schematic symbol you use needs to have a PCB footprint associated with it. The PCB footprint will define the component's physical dimensions and placement of the copper pads or through holes. Now is a time to decide which components you will be using.



Once all of your symbols are placed on the schematic and you have assigned values to each symbol, it is time to start drawing the wires.



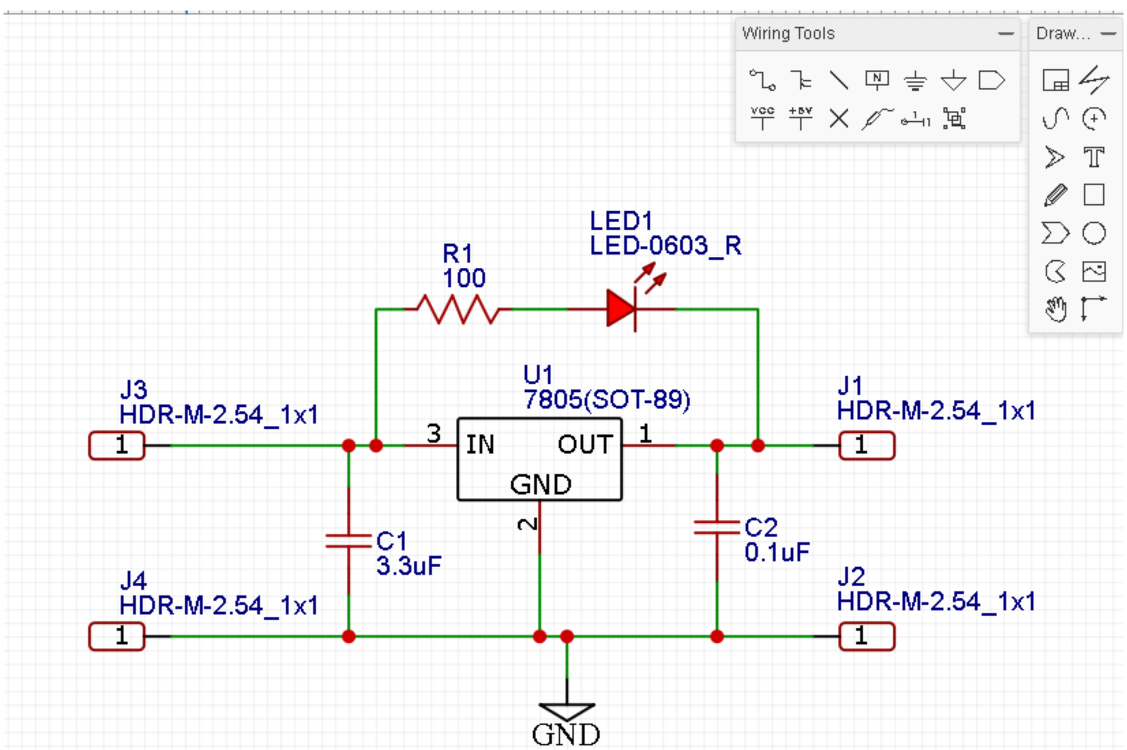
Select  in the Wiring Tools window and start connecting one component to another using your mouse.



After all the wiring is done, using text feature it is a good practice to include design text in the schematic i.e. your name or team, date, version/revision of the schematic design, etc.

The labels will be transferred over to the PCB layout and eventually be printed on the finished PCB.

Each symbol has a name (U1, D1, R1, C1, C2, etc.) and value (LM7805, LED, 100 Ω, 3.3 µF, etc.) that can be edited by clicking on the label.

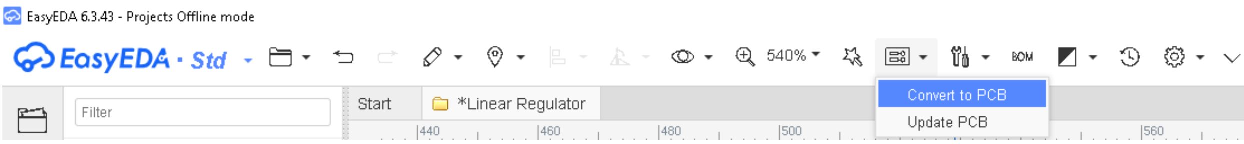


The next step is to import (i.e. front annotating) the schematic into the PCB editor.

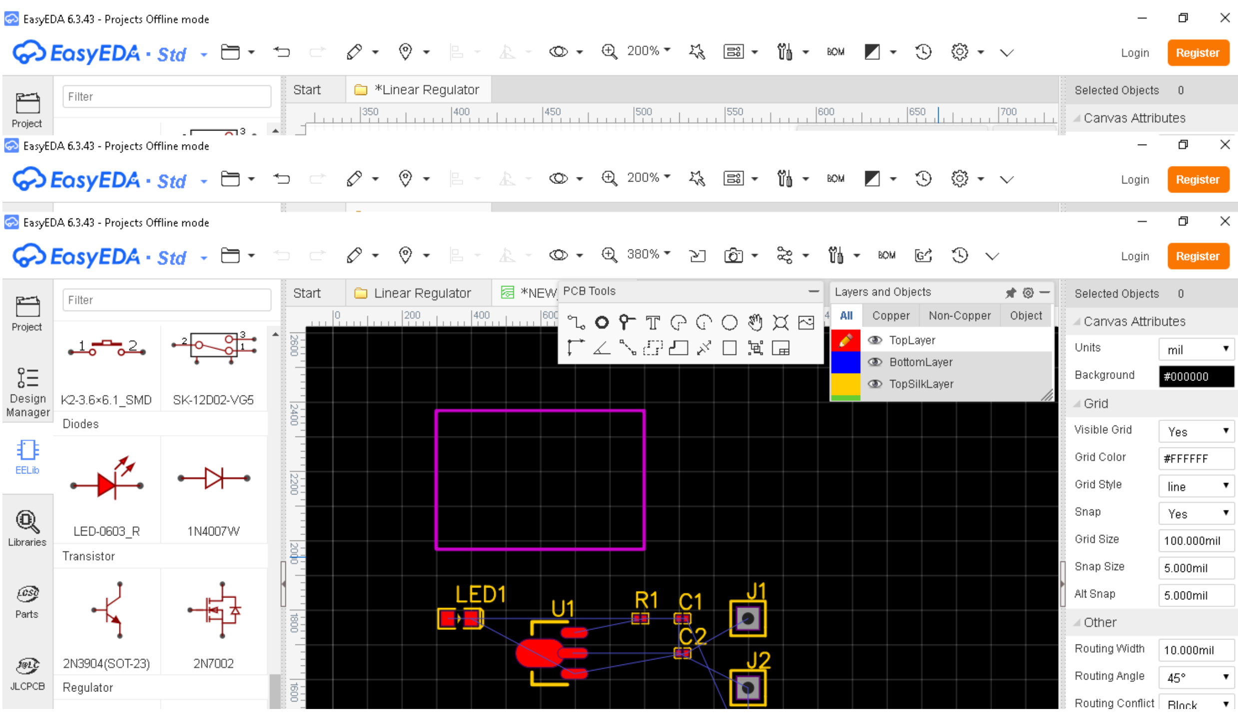
6. Designing PCB Layout

In this section we start to layout a PCB from the given schematic of the linear voltage regulator circuit in the EasyEDA software.

From the schematic in the schematic editor, click on the “Convert Project to PCB” button.

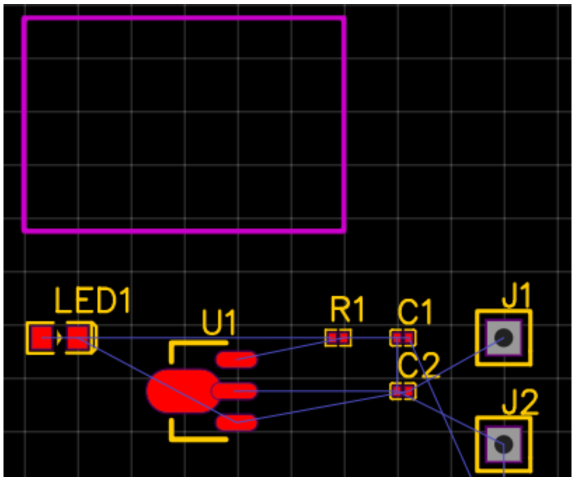


The window will be converted to PCB design editor window. All of the components in the schematic and their connections in the schematic editor window of EasyEDA are transfered to this PCB editor window. The footprints associated with each schematic symbol are also automatically generated in the PCB editor. See also the outlines of these components.



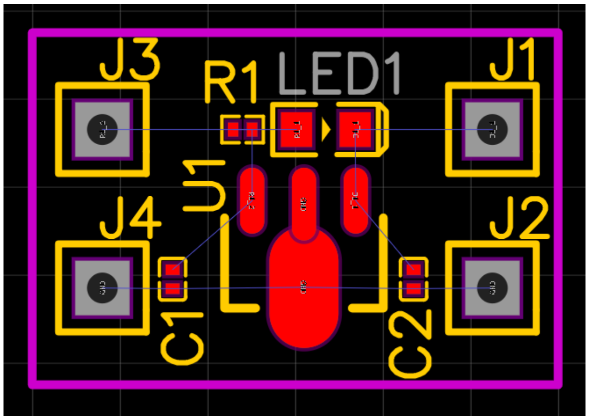
Notice the thin blue lines connecting the components. These are called ratsnest lines.

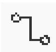
Ratsnest lines are virtual wires that represent the connections between components. They show you where you need to route the traces according to the wiring connections you created in your schematic.

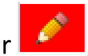


Now you can start arranging the components, keeping in mind the design tips mentioned in the previous section. You might want to do some research to find out if there are any special design requirements for your circuit. Some circuits perform better with certain components in specific locations. For example, in a linear voltage regulator circuit the decoupling capacitors need to be placed close to the chip to reduce noise.

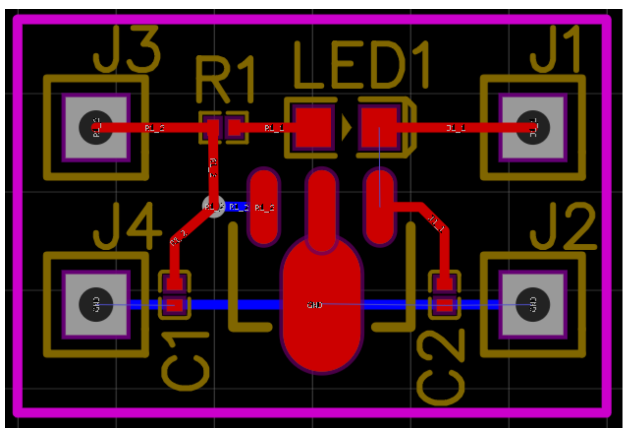
After you have arranged all of the components in the PCB outline, it is time to start drawing the traces.



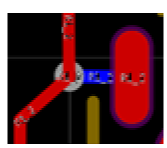
Click on  in the PCB Tools panel to change the mode of operation of the user interface of EasyEDA to trace.

Make sure that you work on the right layer by clicking on the correct layer in the layers and objects panel i.e. for two sided layer PCB (top and bottom layers), ensure that you click on top layer , or click on  to work on the bottom layer

Use the ratsnest wires as a rough guide for routing each trace. However, they would not always show you the best way to route the traces, so it is a good idea to refer back to your schematic to verify the correct connections.

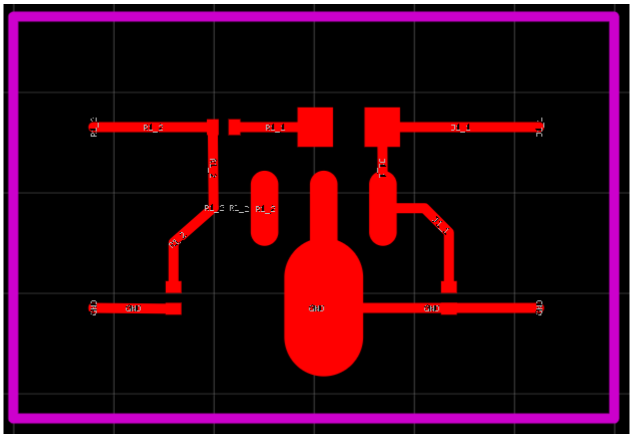


Notice that in the tracing given above the red traces are on the top layer and the blue traces are on the bottom layer. See also the via included to enable connection from the top layer to the bottom layer.

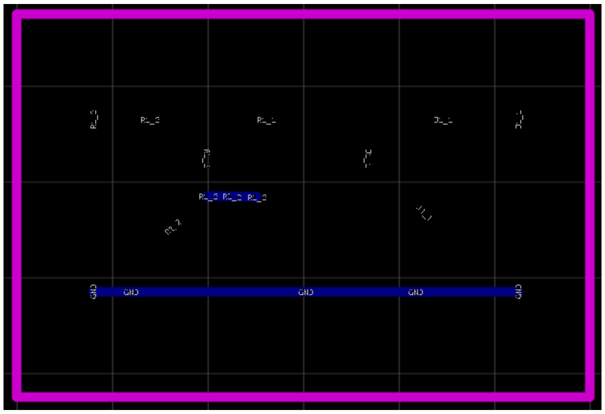


You can toggle the eye icon  inside the layers and objects panel to hide or show any particular layer of the PCB.

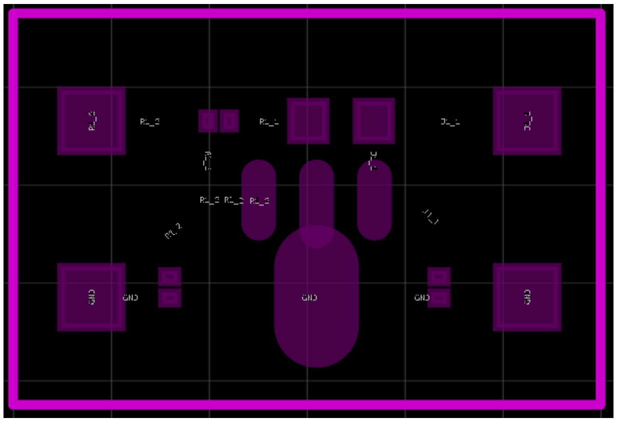
See the diagram below for displaying the top solder layer only.



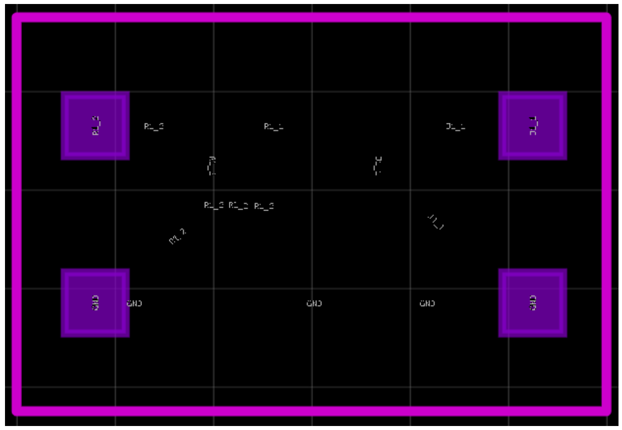
The figure below is what the bottom solder layer looks like.



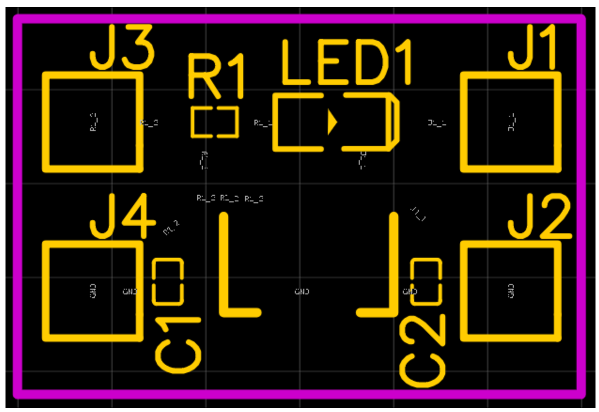
The top solder mask layer is shown in the figure below.



The bottom solder mask layer is as shown in the figure below.



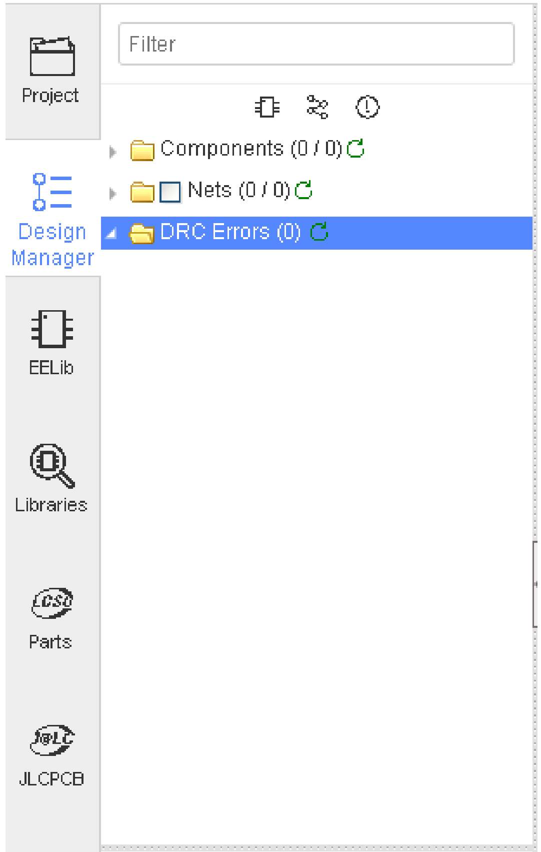
The figure shown below component outlines (silkscreen) layer of the PCB.



Add also PCB design texts to the PCB i.e. 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).

The last thing to do is to 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.

The design rule check can be found by clicking the "Design Manager" option in the left-hand side panel in the PCB editor window:



Items that fail the design rule check will be listed below the "DRC Errors" folder. If you click on one of the errors, the problem trace or component will be highlighted in the PCB view.

Finally, at this point, it is a good practice to double check your PCB layout against your schematic to make sure that everything is connected properly.

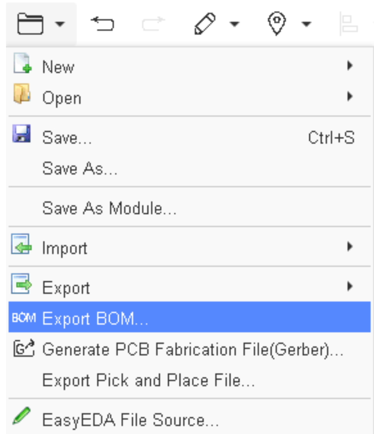
7. Design Project Documentation

Once you have finished creating the schematic and PCB of the circuit. Several project documentations are typically needed for circuit design project. These requirements are useful later on for further steps and processing in the manufacturing of the circuit i.e. construction of the PCB and component ordering.

7.1. Bill of Materials

Simultaneously with the schematic's creation, the bill of materials (BOM) should be generated. The components in the circuit should be selected by analysing the maximum operating voltages and current levels of each node of the circuit while considering tolerance criteria. With electrically satisfactory components chosen, each component should be reconsidered based on availability, budget, and size.

Click on the BOM option of the folder drop-down menu  which is available in toolbar of the EasyEDA for Excel spreadsheet of the BOM.



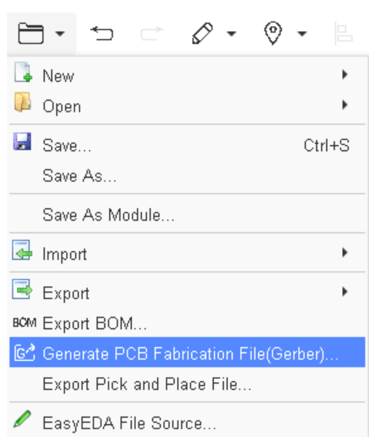
The BOM must be kept up-to-date with the schematic at all times. The BOM requires the quantity, reference designators, value (numeric value of ohms, farads, etc.), manufacturer part number, and PCB footprint for each component.

These five requirements for the BOM are critical because they define how many of each part are needed, explain identification and circuit locations while exactly describing each circuit element used for purchasing and substitution, and explain the size of each part for area estimations. Additional descriptions may be added, but it should be a condensed list describing each circuit element, and too much information can over-complicate library development and management.

7.2. PCB Documentation

The PCB's 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 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. Click on the Generate PCB Fabrication File (Gerber) option of the folder drop-down menu  in the toolbar which is available in the EasyEDA to create a zip file of all of these 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.

The special features included in the FAB outline include but are not limited to notches, cut outs, bevels, back-filled vias-in-pad (used for BGA-type IC packages that have an array of pins under the device), blind/buried vias, surface finish and levelling, hole tolerances, layer count, and more.

If you are satisfied with the result, the next step is to order the PCB.

8. Ordering the PCB

For manufacturing the PCB, you are required to send standardised Gerber file to the manufacturer of the PCB. You can create your PCB's Gerber files in the EasyEDA software before you send them to the manufacturer.

Gerber files are a set of image files that contain the patterns used to manufacture your PCB.

All of the files are compressed into a single .zip file. There is a separate file for the copper traces, silkscreen, and locations of drill holes and vias.

References

1. Cohen, Patricio, Concepts and terminology used in Printed Circuit Boards (PCB), Electrosoft Engineering, Web, May 25, 2018.

2. Mauney, Charles, Thermal Considerations for Surface Mount Layouts, Texas Instruments, Web, May 13, 2018.