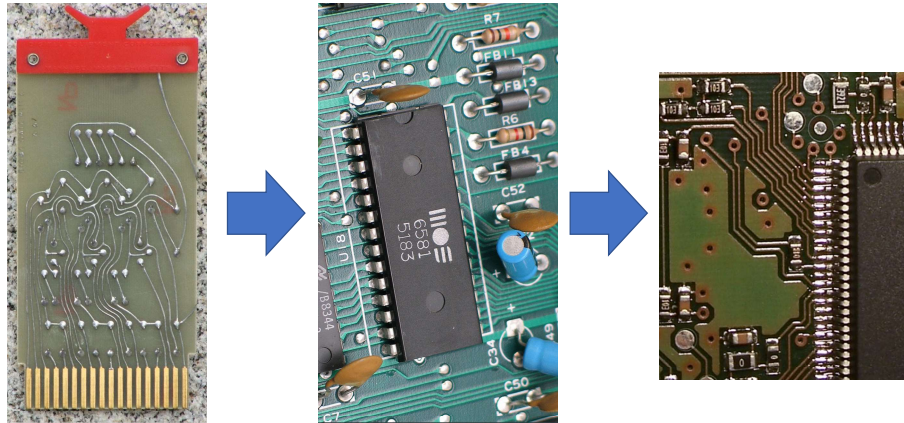




PCB Design 101

What is a PCB?

A printed circuit board (PCB) **mechanically supports** and **electrically connects** electronic components or electrical components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. (Wikipedia)



PCB 101

A printed circuit board (PCB) mechanically supports and electrically connects electronic components or electrical components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. (Wikipedia)

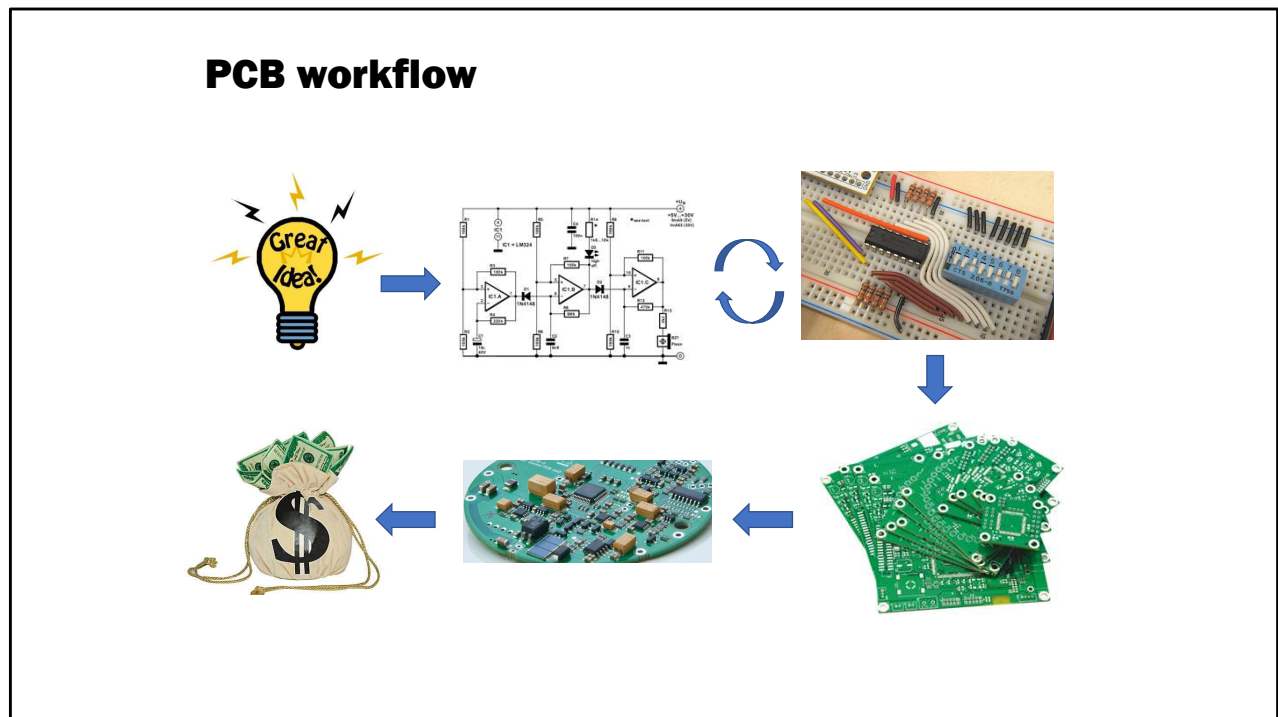
We've seen incredible progress in the PCB fabrication techniques, specifically in terms of providing industry grade PCBs fabrication affordable for makers like ourselves!

Reference:

<https://learn.sparkfun.com/tutorials/pcb-basics>

https://en.wikipedia.org/wiki/Printed_circuit_board

<http://www.circuitbasics.com/make-custom-pcb/>



PCB Workflow

While this is not a definitive guide to PCB workflow it should give you a good sense of what are the most common scenarios for a PCB creation.

Following the picture above we have:

Idea: this is when you envision your new product

Circuit: your idea gain life through the careful design of your product. This is where learning how to use the components will help.

Breadboard: you can prototype your circuit using a breadboard much like the one we're using today at the workshop, learn, adjust and iterate. Expect to go back and forth within steps 2 and 3 several times.

PCB (printed circuit board): once you have the circuit design complete, you can create a board to match the circuit and have it fabricated.

Assembly: add components to your board

Monetize: the final step in this loop is when you monetize your project.

A more detailed view of steps 2 and 4:

1. Create the schematics

Add components

Create connections

Define part footprint (sizes and types)

Add annotations and comments (specially those around technical decisions and tradeoffs, so you'll remember them next time)

Check nets for loose components and connections, and fix them

Review it all once again

2. Design the PCB

PCB size: define your PCB size, smaller designs can be trickier to create and solder.

Part Packages: Choice of part packages will heavily influence the board size (using 0402 SMD components save tons of space, rather than larger ones)

PCB layers: define how many layers are needed given your size constraints

PCB shape: using a PCB software will give you the flexibility to create boards in any shape you need.

Connections to external components: many components may live outside of the PCB (potentiometers, inputs/outputs, panel LEDs, speakers, etc) so you have to think how will these components be connected to the PCB.

PCB mounting: Define if mounting hole will be needed, how many, their position and hole diameter

Review

Connections and connectors check for usability: Consider how the final board will look like, where the connectors will be and if the orientation positioning of the parts make sense and if they work.

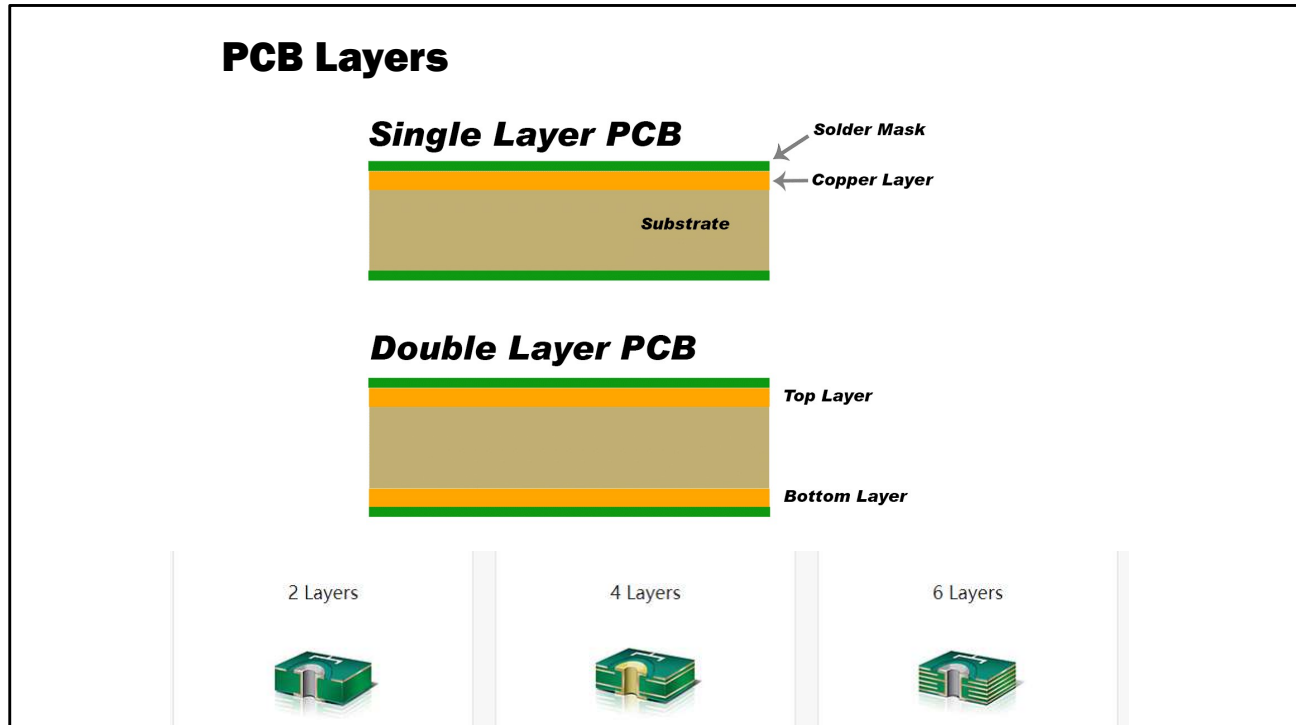
Nets: check nets for loose components and connections, and fix them

Print a full size version of your design: this is a crucial step to give you a good sense of how the final board will look like and allow for a more detailed review of your design.

Order

This can be replaced by export to GRBL so that you can fabricate the board yourself or get it fabricated by a 3rd party

Ordering from the Easy EDA website will take you to JLCPCB website



PCB Layers

Larger circuits can be difficult to design on a single layer PCB because it's 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.

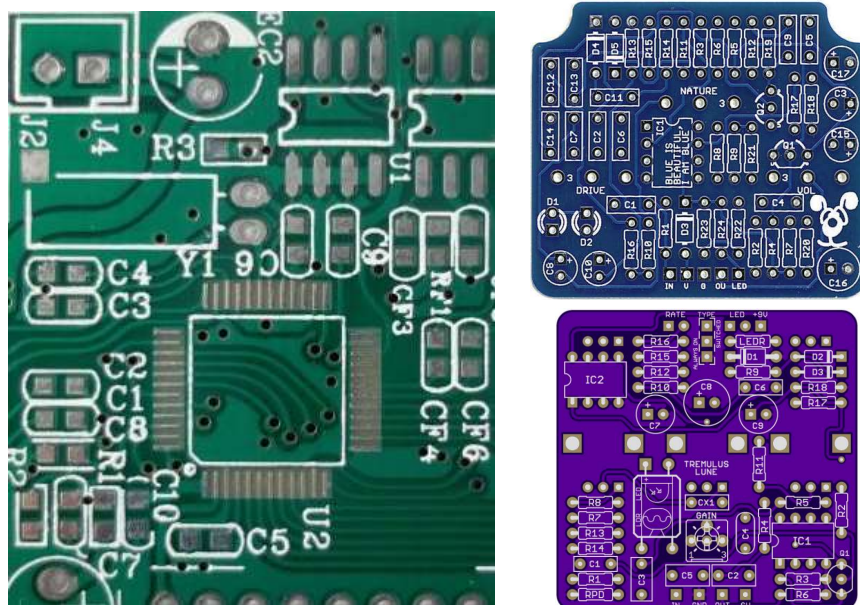
The traces on one layer can be connected to the other layer with a via. A via is a copper plated hole in the PCB that electrically connects the top layer to the bottom layer. You can also connect top and bottom traces at a component's through hole.

Most of the time, 2 layers will suffice for most of our projects. Normally the upper layer will have a Ground plane, which is a continuous block of copper defined by the borders of the board. It is called ground plane because this copper area will be connected to Ground. In a 2-layer board, the bottom layer will have a VCC plane.

4 layer boards will have both ground and VCC planes in the two inner layers, leaving both upper and lower layers for circuit paths.

In this workshop we'll only cover the two-layer model.

PCB Solder Mask / Silkscreen



PCB Solder Mask

The layer on top of the copper foil is called the solder mask layer. This layer gives the PCB its (usually) green color. 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 above, the green solder mask is applied to most of the PCB, covering up the small traces but leaving the silver rings and SMD pads exposed so they can be soldered to. As you can see, even though the green color is the most common one for PCBs, it is not the only option: PCBs can have solder masks in a wide variety of colors. Note that non-green color might have a slight increase in the fabrication price.

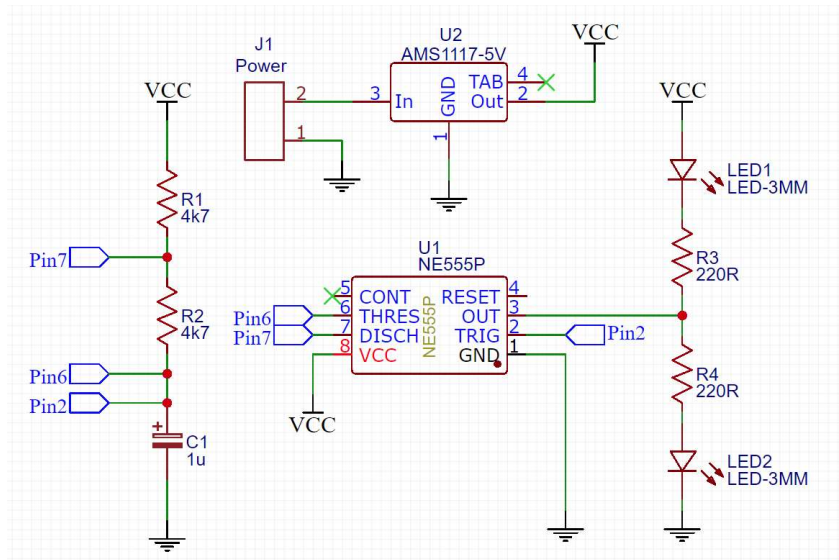
PCB Silk Screen

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. We often use silkscreen labels to indicate what the function of each pin or LED.

During the PCB design phase, you can add, modify and remove text and symbols from the silk screen layer, as well as adding images to be printed in your PCB. We can have silk screen on both layers of the PCB.

Source: <https://learn.sparkfun.com/tutorials/pcb-basics>

PCB: It all starts with a Schematic



PCB Schematics

Before you start designing your PCB, you need to have a schematic of your circuit.

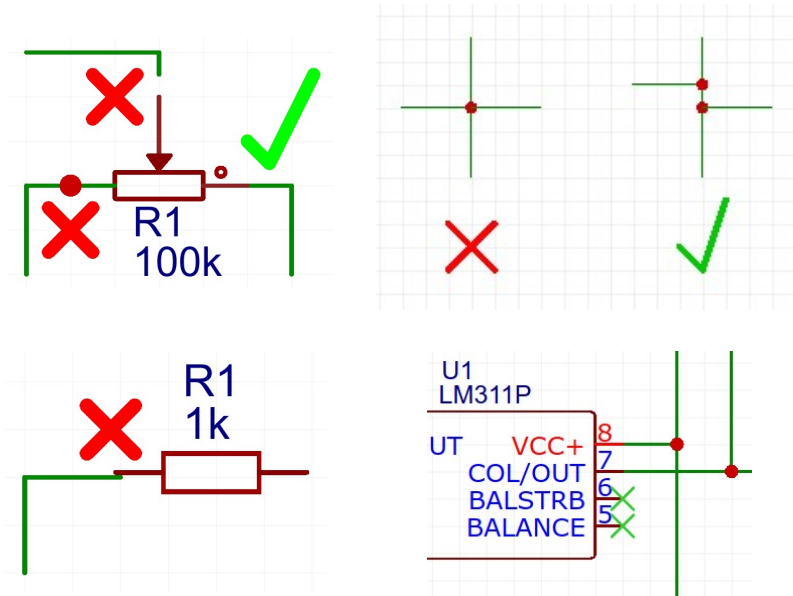
The schematic will serve as a blueprint for laying out the traces and placing the components on the PCB. Plus, the PCB editing software can import all of the components, footprints, and wires into the PCB file, which will make the design process easier.

PCB Good Practices

- Keep your traces as short as possible
- Protect your circuit: Voltage regulators, reverse polarity protection via shottky diodes
- Avoid noise: Add decoupling capacitors, filtering capacitors, no 90 degrees curves in your traces
- Add test points to facilitate later validation (pogo pins for high volume)
- Add breakpoints for voltage lines if you need to insulate and test them (especially useful for high voltage lines)

- Remember to enlarge trace width for power lines!!
- Think about mounting for your final board and add holes as needed.
- Print your PCB to a 1:1 scale for validation (layout and hole sizes)

Schematic good practices



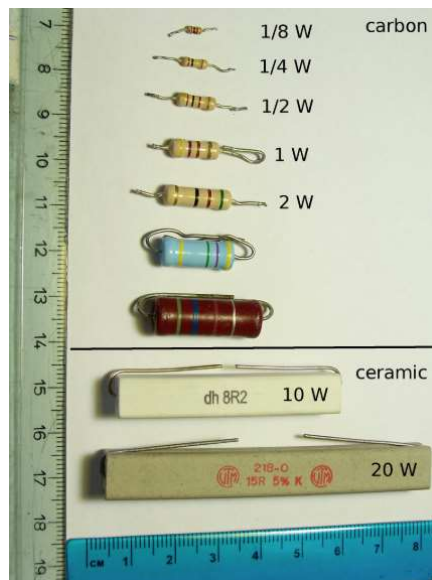
Schematics good practices

Name all your components correctly, including their values

For more complex circuits consider adding text explaining what each part of the circuit does

Use Net Ports to connect points of the circuit without having to draw long and crossing lines

Parts footprint



Metric code	Imperial code
0402	01005
0603	0201
1005	0402
1608	0603
2012	0805
2520	1008
3216	1206
3225	1210
4516	1806
4532	1812
5025	2010
6332	2512

Part footprint

Every component you add to your schematics is actually divided into two parts: the schematics representation and the physical representation.





















A resistor could be represented by either the US symbol or the ISO one. So all resistors in the schematic would pretty much look like the same. As the board designer it is up to you to define which component size you need for that circuit.

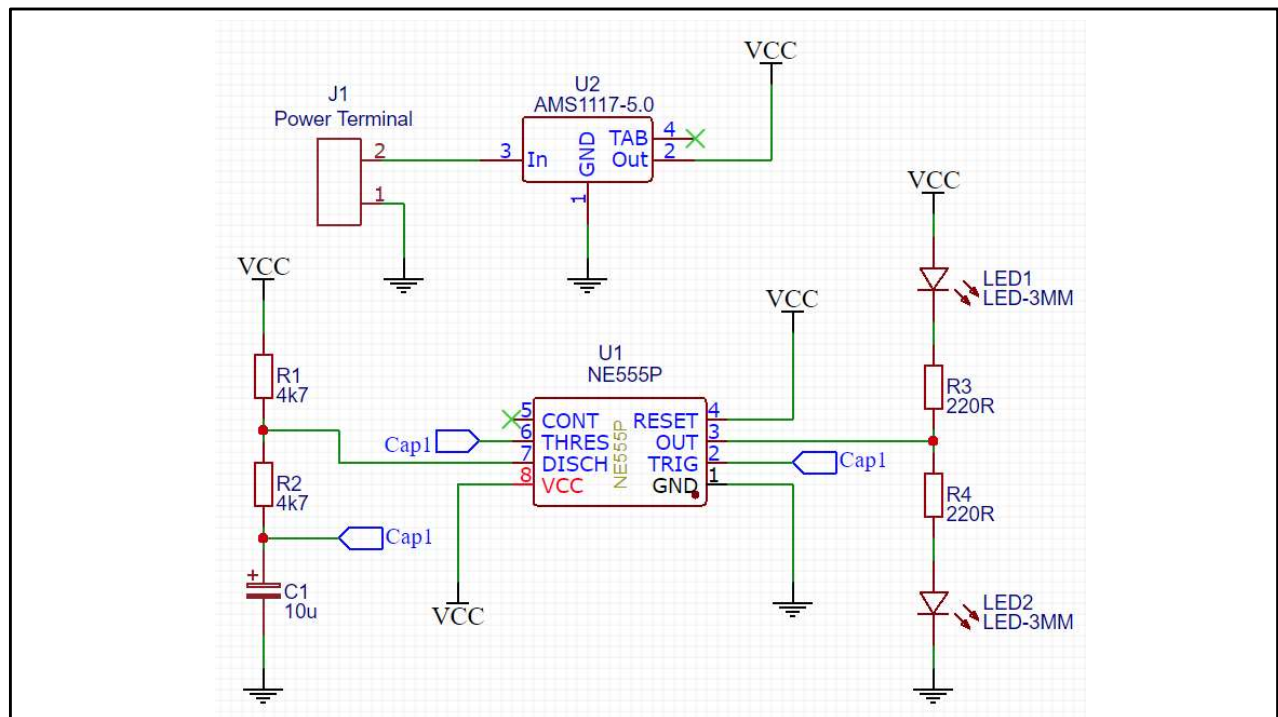
Considering the resistor example, as the picture above shows, we have two major categories:

THT - through hole technology refers to the mounting scheme used for electronic components that involves the use of leads on the components that are inserted into holes drilled in printed circuit boards (PCB) and soldered to pads on the opposite side

SMT - Surface-mount technology is a method for producing electronic circuits in which the components are mounted or placed directly onto the surface of PCBs. An electronic device so made is called a surface-mount device (SMD).

Keyboard Shortcuts

Schematics	 Wire	 +  VCC	
	 Text	 +  GND	
ALL	 Rotate	 Zoom +	 Flip Hor.
	 Fit Window	 Zoom -	 Flip Vert
	 +  Save		 Change Attributes
PCB	 Bottom Layer	 Draw Copper Area	
	 Top Layer	 Text	 Via



Initial steps

Log in to <https://easyeda.com> and create an account.

Open the Editor, click New  Project (top left menu), New Schematic

EELIB to the left menu has the most commonly used parts, but not all of them.

Libraries will give you a catalog of virtually all part you'll ever need (probably an exaggeration)

Adding parts to your project

- Go to Libraries and search for “LM555”. Explore the different packages available.
- Get the DIP-8 (Dual in Line Package), hit “Place”.
- Drop it into the schematic canvas. The component will remain selected in case you need to drop multiple items. Hit “esc” on your keyboard if you only need one.
- Zoom in and out to get used to the interface. Zoom occurs towards the mouse pointer. Try it!

- Shortcut to try: “R” will rotate the part. Do that until pin 8 of the LM555 is in the lower left corner, as the picture above shows.
- Hit Save (CTRL+S) or click File Save often.
- Go back to the Libraries and search for “AMS1117-5”, SOT223, the one that looks like in the picture above, and place it.
- Same thing for a “Screw terminal” get a 1x2, 5x8mm and place it.
- Remaining components are common, and you’ll find them in the EELib menu.
- Select resistor (your choice of American or European style – the picture above shows the European one). The little arrow at the bottom right of each component allow you to select the package. In this case we want Axial 0.3mm.
- Place 4 resistors in the canvas, rotate them if needed (hit R).
- Select Electrolytic Capacitor (picture above shows EU style), TH 3.0 and place one in the schematics.
- Select LED, TH 5mm, place 2 of them. Rotate if needed: you can rotate parts even before placing them in the design. You can always rotate parts later by clicking the part and hitting R.
- Select GND (CTRL+G also works) and place 5 of them according to the picture above.
- Select VCC (CTRL+Q also works) and place 4 of them, one will be rotated.
- From the floating menu “Wiring Tools”, select “Net Port” and place 3 of them in the schematics.
- CTRL + S to save.
- Go to the Design Manager menu on the left menu and expand Nets. Hit refresh (the little icon to the right of Nets) to see if there are any component not connected.
- Since we didn’t connect the components it will show them all as errors or warnings.
- Organize the parts as above (by clicking and dragging the parts, you can also click and use the arrows to move them around). Rename all components to match the schematics above.

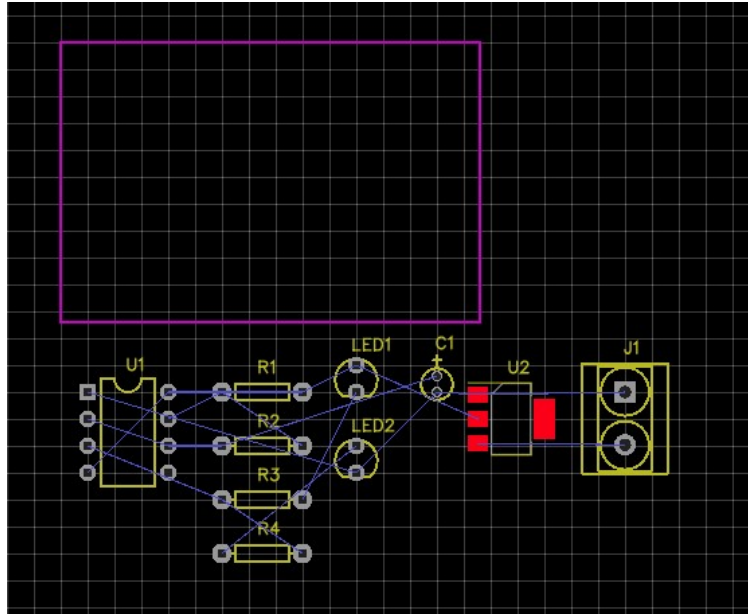
Connecting parts and testing the Schematic

- Connect them! Click W to draw a wire and connect all parts as shown above.
- From the floating menu to the right, select “No connect flag” which is an X, and place it on pin 5 of the LM555, and another one on the Pin 4 of the voltage

regulator, to indicate that these pins won't be connected at all.

- Check Design Manager again by clicking refresh (the little icon to the right of Nets). It should show no errors.
- Change the value for resistors and the capacitors by clicking on them and changing their name on the top right panel.
- Also change the Net ports to "Cap1" all three need to have the same name as they should all be connected.
- Review the schematics and check Design Manager one last time.
- If you don't find any errors in the Nets, you can proceed!
- CTRL + S to save.
- Last step is to click on "Convert to PCB" on the top menu.

Easy EDA: PCB Workflow



Creating your PCB from the Schematics

After clicking on “convert to PCB”:

Change Canvas attributes – Units to mm.

Go to menu “Set board outline”:

- Rectangular
- Width = 60mm
- Height = 40mm
- Start X = 0
- Start Y = 0

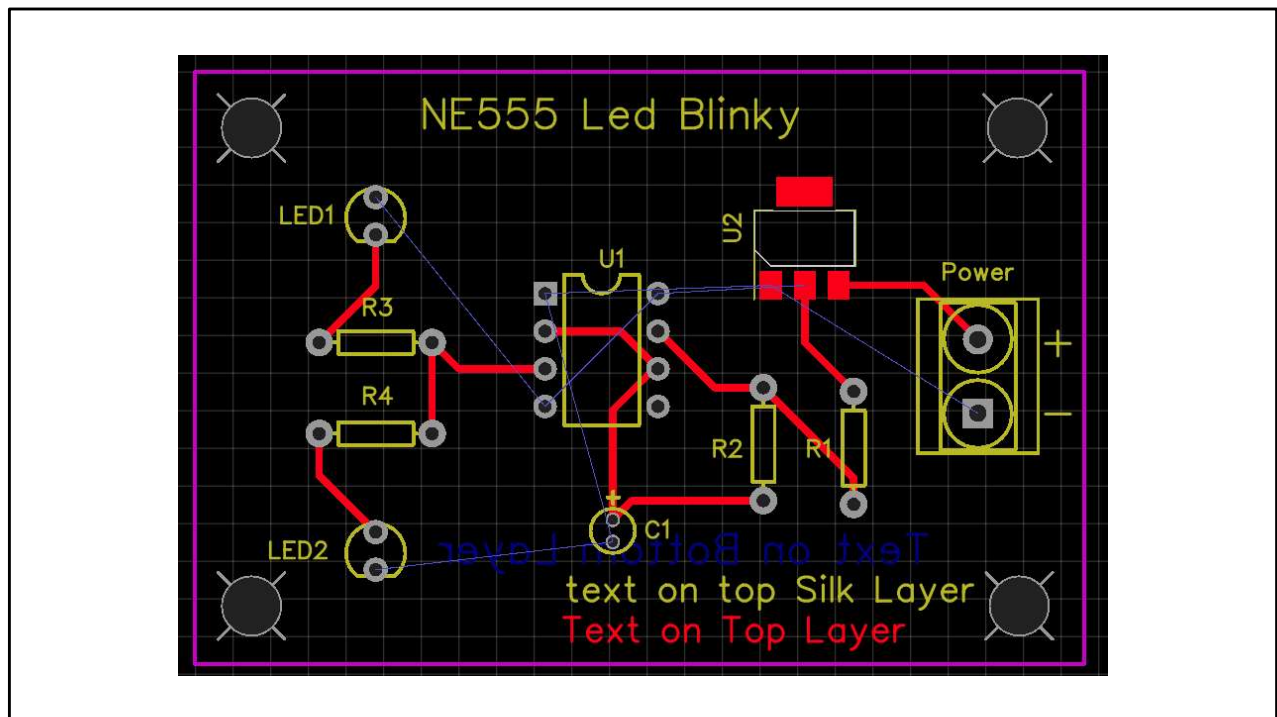
All these configuration items can be changed later if needed.

Once you select them you get a blank board, all components and their connections (blue lines known as ratsnest) as the picture above shows.

CTRL+S to save.

The floating “Layers and Objects” menu shows all available layers you can work with. During this workshop we’ll work only with the following:

- Top Layer
- Bottom Layer
- Top silk Layer
- Bottom Silk Layer



Organizing components in the board

Make sure you're in the Top Layer (the pencil shows in the Red square, which indicates you're editing the Top Layer).

- Drag and drop the component in the board as shown above. Components can be dragged once you hover the mouse pointer and their lines become all white. You can also drag and reposition only the part name.
- Rotate parts (R) to create the shortest path on the blue lines (ratsnest).
- Zooming in helps a lot in this process. If you zoom in enough times, you'll see the name of each connection in the part pins.
- Place all parts.
- On the floating left menu, select T (text) and type "Text on Top Layer". Adjust text size by changing its height.

- Still on the floating menu, select Hole and place 1 of them to the board corners. Change it diameter to 3mm, center X at 3mm and center Y at -3mm.
- Hole 2 will be at 57mm at -3mm
- Hole 3 will be at 3mm and -37mm
- Hole 4 will be at 57mm and -37mm

Move to the Top Silk Layer (pencil will be placed at the yellow square):

- On the floating left menu, select T (text) and type “Text on Top Silk Layer”. Adjust text size by changing its height.
- Add 4 new text that will say “Power”, “NE555 Led Blinky”, “+” and “-” and position them according to the picture above.

Move to the Bottom Silk Layer:

- On the floating left menu, select T (text) and type “Text on Bottom Silk Layer”. Adjust text size by changing its height.

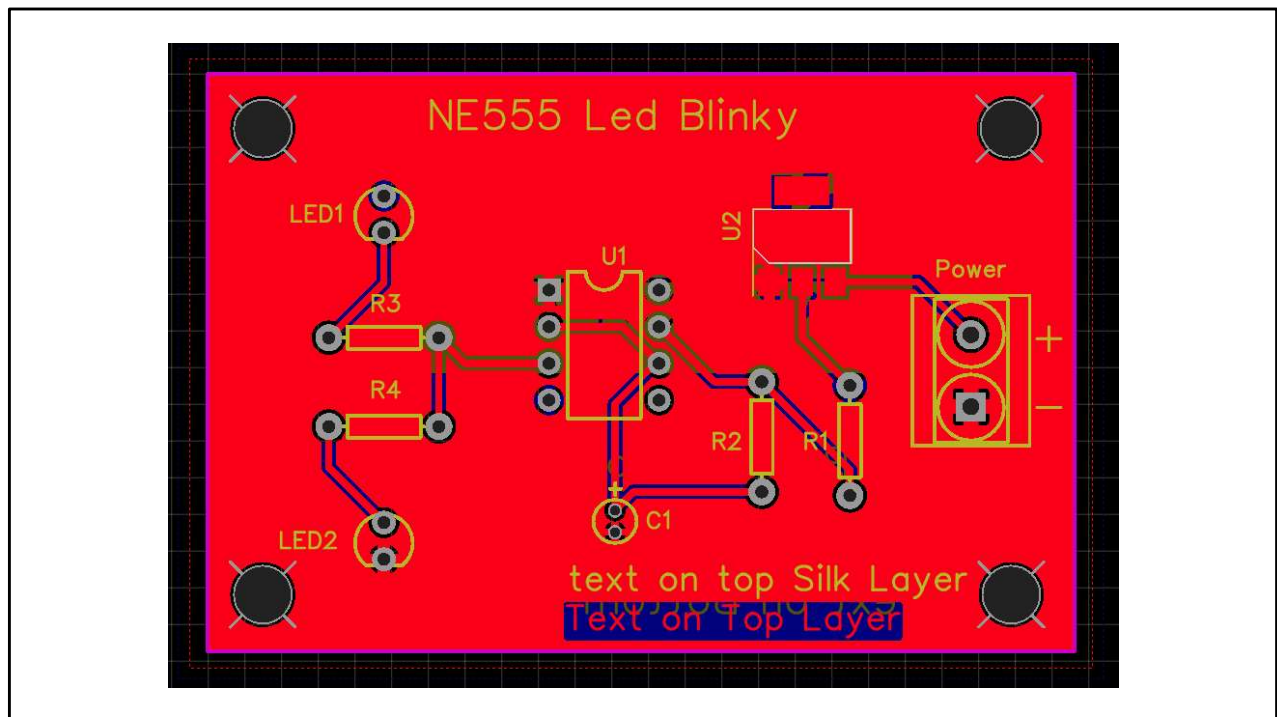
Move to the Bottom Layer:

- On the floating left menu, select T (text) and type “Text on Bottom Layer”. Adjust text size by changing its height.

Go back to the top layer:

- Hit “W” to start drawing the traces in the board. Zoom in will make this task easier: if you zoom enough, you’ll see the names of each segment.
- Connect parts following the blue lines (ratsnest). Hit ESC once you’re finished with a segment.
- Don’t connect any VCC or GND lines. w
- Go to the Design Manager menu on the left menu and expand Nets. Hit refresh (the little icon to the right of Nets) to see if there are any component not connected.
- At this point you should see some errors on the VCC and GND.

- Connect pin 2 (the middle one) on the voltage regulator to R1, this is a VCC line. Change the width of this line from the standard 0.254mm to 0.5mm.
- Power lines usually need to be wider than normal signal lines. Specially the entry points like in this case.



Creating GND and VCC planes

Creating the top GND plane (the copper area on top of the board that is connected to GND):

- Change the view to the top layer.
- On the floating left menu, select “Copper area” and draw a rectangle around the board.
- As soon as you release the mouse button it will automatically create the GND plane as shown in the picture above.
- You can change this ground plane by clicking the dotted line of the rectangle to select it. It also can be removed.
- Note that when you select this rectangle the Net = GND. This tells Easy EDA that this plane should be connected to the GND on the board.

- If you add a new component or change any position or trace in the design, the ground plane will need to be rebuilt. Just select the rectangle and click “Rebuild Copper Area”

Creating the bottom VCC plane (the copper area at the bottom of the board that is connected to VCC):

- Change the view to the bottom layer.
- On the floating left menu, select “Copper area” and draw a rectangle around the board.
- Click the dotted line of the rectangle to select it and change its Net from GND to VCC and then click “Rebuild Copper Area”.
- Check Design Manager again by clicking refresh (the little icon to the right of Nets). It should show no errors.

Review the board: on the top menu try Photo View and 3D view.

At this point the board is ready for fabrication!

Go back to board view, on the top menu select “generate fabrication file (Gerber)”.