

# Eagle Tutorial Day 1

Heyo, welcome to the first day of learning how to use Eagle! Today, we'll be focusing on the following things:

- Installing and setting up Eagle
- Eagle Control Panel and navigating its rly confusing menus
- Creating your first schematic
- How to use eagle command manager
- Basic electronic components and what they do
- Use of libraries; RCL, LED, other common libraries and packages
- Creating a simple circuit that lights up an LED when a button is pressed

**End of Day Goal: Know how to navigate Eagle schematic designer and libraries and create a very simple schematic**

If anyone has any questions or feedback on the format of the lessons feel free to let me know by pm @aaron on the FCA slack

## Part 1: Installing Eagle

You'll ofc need Eagle installed in order to do anything with it, here's how

Go to: <https://www.autodesk.com/education/free-software/eagle>



## EAGLE

Make anything with Autodesk EAGLE. Bring your next spark of an idea to life with powerful, easy-to-use PCB design software made for every engineer.

[System Requirements](#)

[About EAGLE for Classrooms and School Labs](#)

### STEP 1: SIGN IN OR CREATE AN ACCOUNT

I have an account

[SIGN IN](#)

I need an account

[CREATE ACCOUNT](#)

Not a student or educator? [Get the free trial](#)

### Step 2: Get access to EAGLE

[GET ACCESS](#)

[You are receiving an Educational license. See more](#)

~~Create an account first, make sure to register as a student so everything is free. Make sure to register with your school email (@edu or similar) NOT personal email in order to get approved (which might take a day or so)~~

~~Once your account has been approved as an “education” account, download and install Eagle according to the autodesk website. Once it’s open, proceed to next step~~

Thanks to Harry for figuring out that between 4am when I wrote this and 8/4, this method no longer works lol

Instead go to

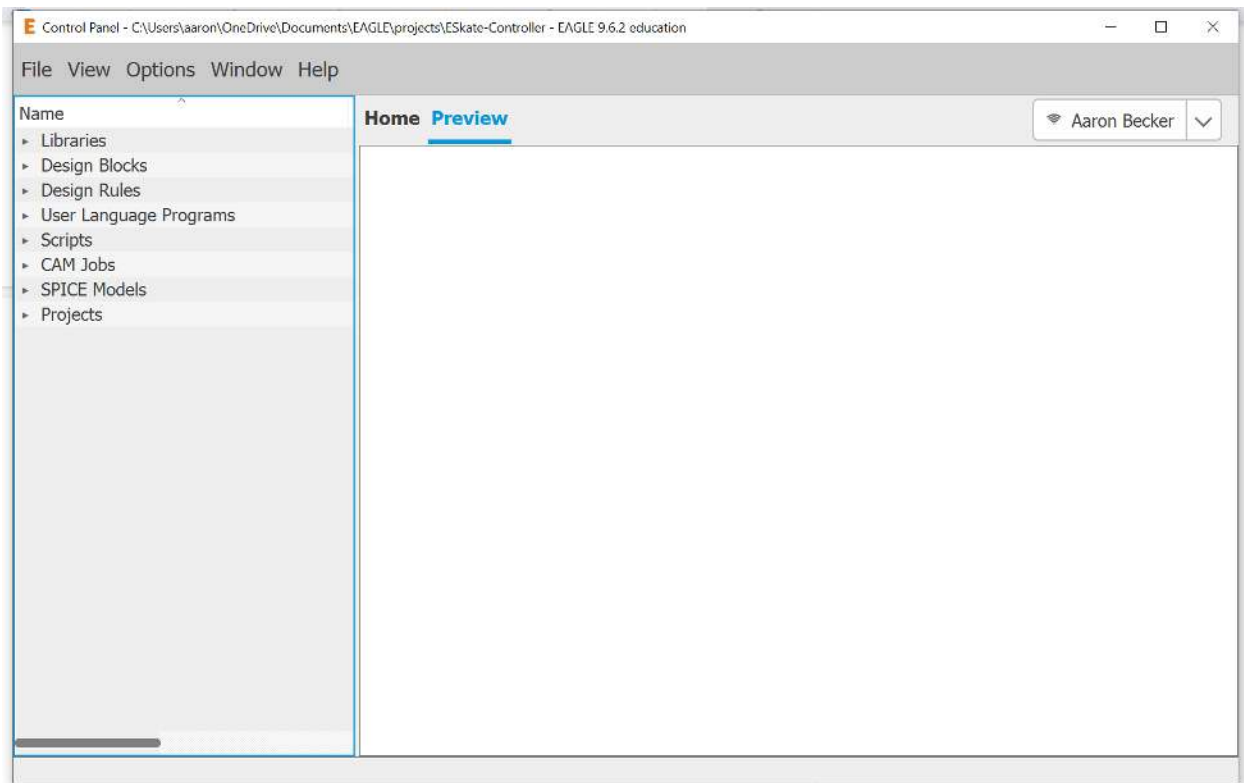
<https://www.autodesk.com/products/eagle/free-download?plc=F360&term=1-YEAR&support=ADVANCED&quantity=1>

If it will not run after download and stays on the splash screen, this will tell you how to fix it:

[https://knowledge.autodesk.com/support/eagle/troubleshooting/caas/sfdcarticles/sfdcarticles/Eagle-crashes-seconds-after-launching-splash-screen.html?\\_ga=2.171513056.1959485477.1606773650-835405581.1602797253](https://knowledge.autodesk.com/support/eagle/troubleshooting/caas/sfdcarticles/sfdcarticles/Eagle-crashes-seconds-after-launching-splash-screen.html?_ga=2.171513056.1959485477.1606773650-835405581.1602797253)

# Part 2: First Look at the Control Panel

Once you have Eagle open for the first time, you'll see something like this



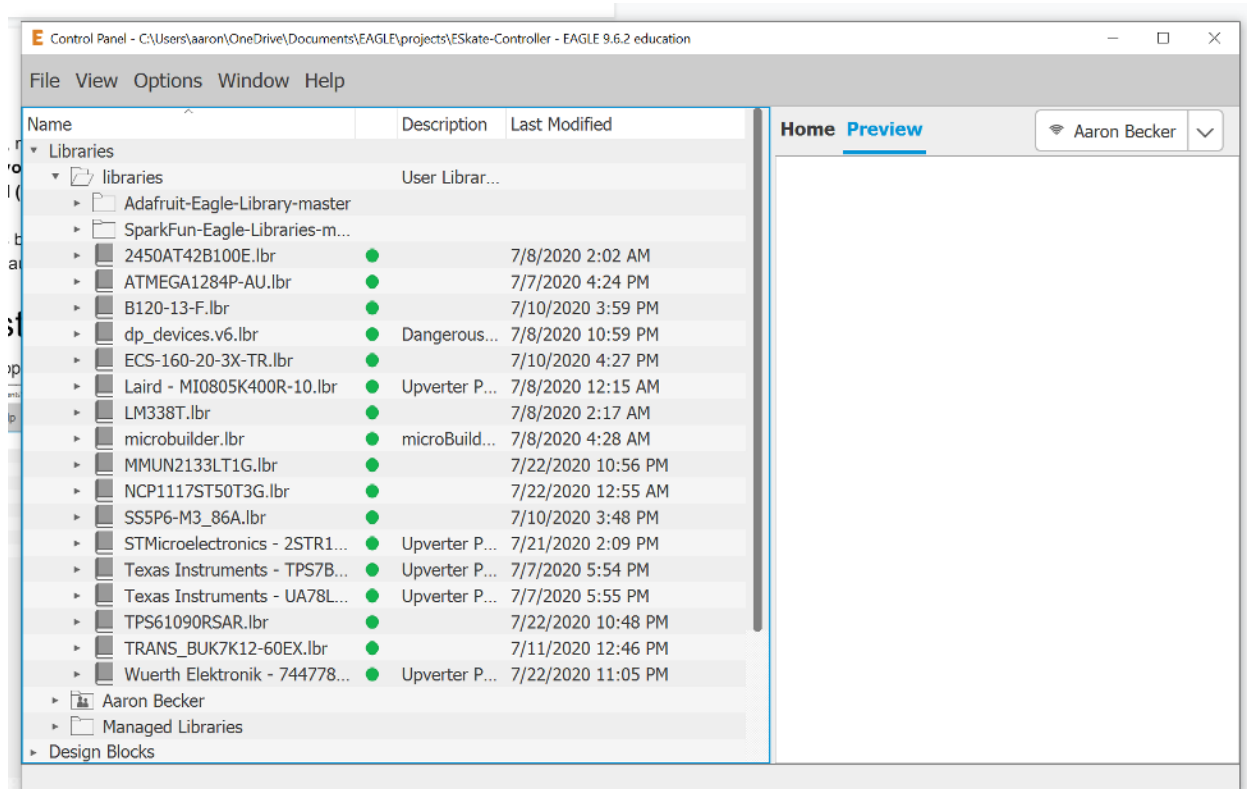
This window is called the “Control Panel”. I think of it kinda like file explorer or Finder for Eagle; you can manage your projects, install new libraries (which hold different components).

Running thru the options on the side one at a time:

## Libraries

Libraries hold “families” of components that you have installed. For example, a component is a single transistor, capacitor, resistor, or even IC. Eagle itself comes with around ~100 libraries to mess around with. The screenshot below shows my library setup (Under the “libraries” subfolder); in addition to Eagle’s basic libraries I have ~20 extra ones for parts that I

couldn't find in the default ones. We'll go over later how to download and install these libraries properly



Here's some of Eagle's default libraries:
















|   |                       |   |                                 |
|---|-----------------------|---|---------------------------------|
| ▼ | Managed Libraries     |   |                                 |
| ▼ | Eagle Pcb             |   |                                 |
| ▶ | 19inch.lbr            | ● | 19-Inch Sl... 6/8/2020 1:04 AM  |
| ▶ | 40xx.lbr              | ● | CMOS Logi... 6/8/2020 1:04 AM   |
| ▶ | 41xx.lbr              | ● | 41xx Serie... 6/8/2020 1:04 AM  |
| ▶ | 45xx.lbr              | ● | CMOS Logi... 6/8/2020 1:04 AM   |
| ▶ | 74ac-logic.lbr        | ● | TTL Logic ... 6/8/2020 1:04 AM  |
| ▶ | 74ttl-din.lbr         | ● | TTL Device... 6/8/2020 1:04 AM  |
| ▶ | 74xx-eu.lbr           | ● | TTL Device... 6/8/2020 1:04 AM  |
| ▶ | 74xx-little-de.lbr    | ● | Single and ... 6/8/2020 1:04 AM |
| ▶ | 74xx-little-us.lbr    | ● | Single and ... 6/8/2020 1:04 AM |
| ▶ | 74xx-us.lbr           | ● | TTL Device... 6/8/2020 1:04 AM  |
| ▶ | 751xx.lbr             | ● | 75xxx Seri... 6/8/2020 1:04 AM  |
| ▶ | allegro.lbr           | ● | Allegro Mic... 6/8/2020 1:04 AM |
| ▶ | altera-cyclone-II.lbr | ● | ALTERA Cy... 6/8/2020 1:04 AM   |
| ▶ | altera-stratix-iv.lbr | ● | Altera Stra... 6/8/2020 1:04 AM |
| ▶ | altera.lbr            | ● | Altera Prog... 6/8/2020 1:04 AM |
| ▶ | am29-memory.lbr       | ● | Advanced ... 6/8/2020 1:04 AM   |
| ▶ | amd-mach.lbr          | ● | AMD MAC... 6/8/2020 1:04 AM     |
| ▶ | analog-devices.lbr    | ● | Analog De... 6/8/2020 1:04 AM   |
| ▶ | atmel.lbr             | ● | AVR Devices 6/8/2020 1:04 AM    |
| ▶ | avago.lbr             | ● | AVAGO Te... 6/8/2020 1:04 AM    |
| ▶ | battery.lbr           | ● | Lithium Ba... 6/8/2020 1:04 AM  |

You can find these by going to “Managed Libraries” -> “Eagle Pcb” under the Libraries tab.

The green or gray dot indicates whether the library is “enabled” or not, in my case I’ve disabled many of the ones I’m not using to free clutter, but don’t worry about this for now.

## Design Blocks

Think of design blocks like smaller pieces of a larger circuit. I personally don’t really use design blocks that often but they’re useful if there’s a part of a circuit that’s used across multiple designs (for example, a filtration circuit for a fluctuating power supply).

|  |                |                   |
|--|----------------|-------------------|
| ▼ Design Blocks  |                |                   |
| ▶ design blocks  | User Desig...  |                   |
| ▼ examples   | Example D...   |                   |
| ▶ Adafruit   |                |                   |
| ▶ Nordic   |                |                   |
| ▶ SparkFun   |                |                   |
| ▶ Timer  |                |                   |
| ▶ USB to UART  |                |                   |
|  2N3904 NPN Transistor.dbl      | General Pu...  | 5/19/2020 1:45 PM |
|  3V3-Voltage-Regulator_LM...    | 3-Terminal...  | 5/19/2020 1:45 PM |
|  5V-Voltage-Regulator_LM3...    | 3-Terminal...  | 5/19/2020 1:45 PM |
|  12V-Voltage-Regulator_LM...    | 3-Terminal...  | 5/19/2020 1:45 PM |
|  24V-Voltage-Regulator_LM...    | 3-Terminal...  | 5/19/2020 1:45 PM |
|  Adjustable-Voltage-Regulat...  | 3-Terminal...  | 5/19/2020 1:45 PM |
|  Audio-Amplifier-with-contro... | Low Voltag...  | 5/19/2020 1:45 PM |
|  Audio-Amplifier-with-maxi...   | Low Voltag...  | 5/19/2020 1:45 PM |
|  Constant-current-source.dbl    | Constant c...  | 5/19/2020 1:45 PM |
|  Current-mirror.dbl             | Current mi...  | 5/19/2020 1:45 PM |
|  Darlington Transistor.dbl     | Darlington ... | 5/19/2020 1:45 PM |
|  DCPL-VCC.dbl                 | VCC Decou...   | 5/19/2020 1:45 PM |
|  DCPL-VDD.dbl                 | VDD Deco...    | 5/19/2020 1:45 PM |
|  ESP-LAUNCHER_test-modul...   | ESP-LAUN...    | 5/19/2020 1:45 PM |
|  ESP8266EX_as-slave-devic...  | ESP8266E...    | 5/19/2020 1:45 PM |

Under examples, you can see things like a voltage regulator circuit, if you click it once a preview will show up on the right if you want to take a quick peek. Honestly I have no idea how most of these work lol and because they're examples they only apply to a specific circuit. But useful to know about

## Design Rules

The design rules, or "DRC" as you'll hear it called (stands for Design Rule Checker) is a part of Eagle to make sure you don't make any mistakes with your design (like place wires too close together or make them too thin). It's very important when doing layouts, which I'll get to in another tutorial, but not really important for today.

## User Language Programs and Scripts

You can write custom scripts that do things like import images onto your PCB, again not really useful right now

## CAM Jobs

A CAM job is used when you're finished with your design and want to send it to a company to make it for you; it exports your design into a format that represents what kind of holes, wires, pads the company producing the boards should make. Again, not important for today

## SPICE Models

Spice modelling is pretty cool; basically, you can simulate a whole circuit to see how it'll respond to certain inputs etc. Unfortunately, in the student version of Eagle, it's really hard to use because most libraries don't come with the correct data to allow spice modelling (since in order to model a component, you need to know how it performs, which requires a lot of testing that companies don't want to give away for free sad boi hrs)

## Projects

Managing your projects, if you want to check out some examples feel free to try that, otherwise we'll move on with the tutorial

## Part 3: Creating your first project



Open the Projects dropdown and right click on the lowercase “projects” subfolder, then select “New Project”



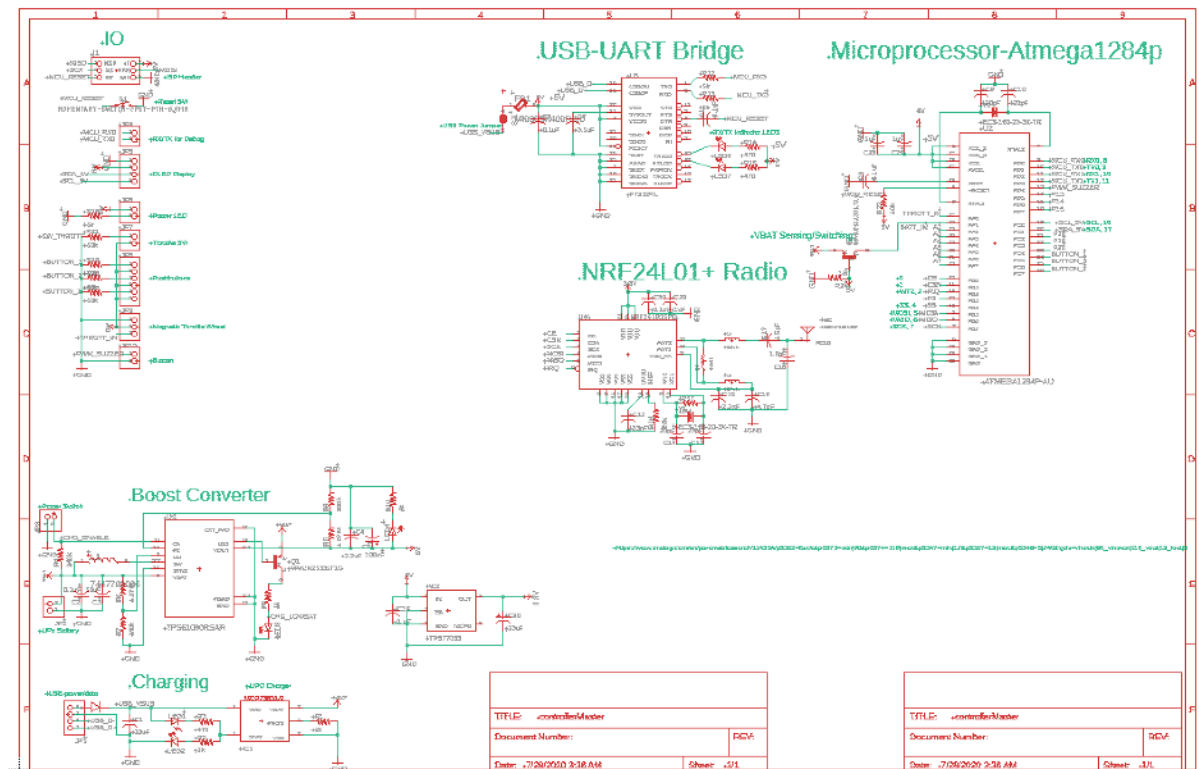
I created a new project called “TestProject”, but sadly it’s still empty

For today, we’re going to be starting with the schematic. Quick explanation of the different parts of a PCB project:

- A schematic is a graphic representation of what **components** go on a project. It doesn’t necessarily model the physical layout of the project at all, just what things are connected to what. Here’s what a

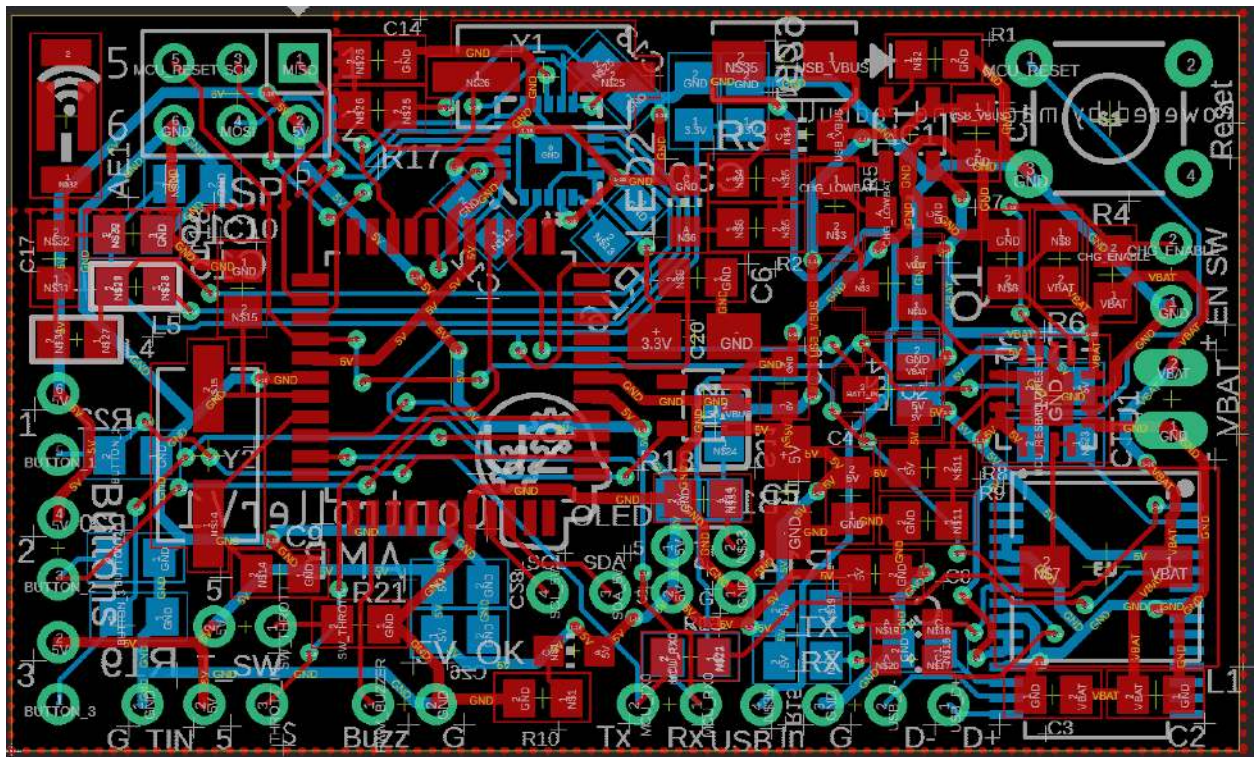


schematic looks like:

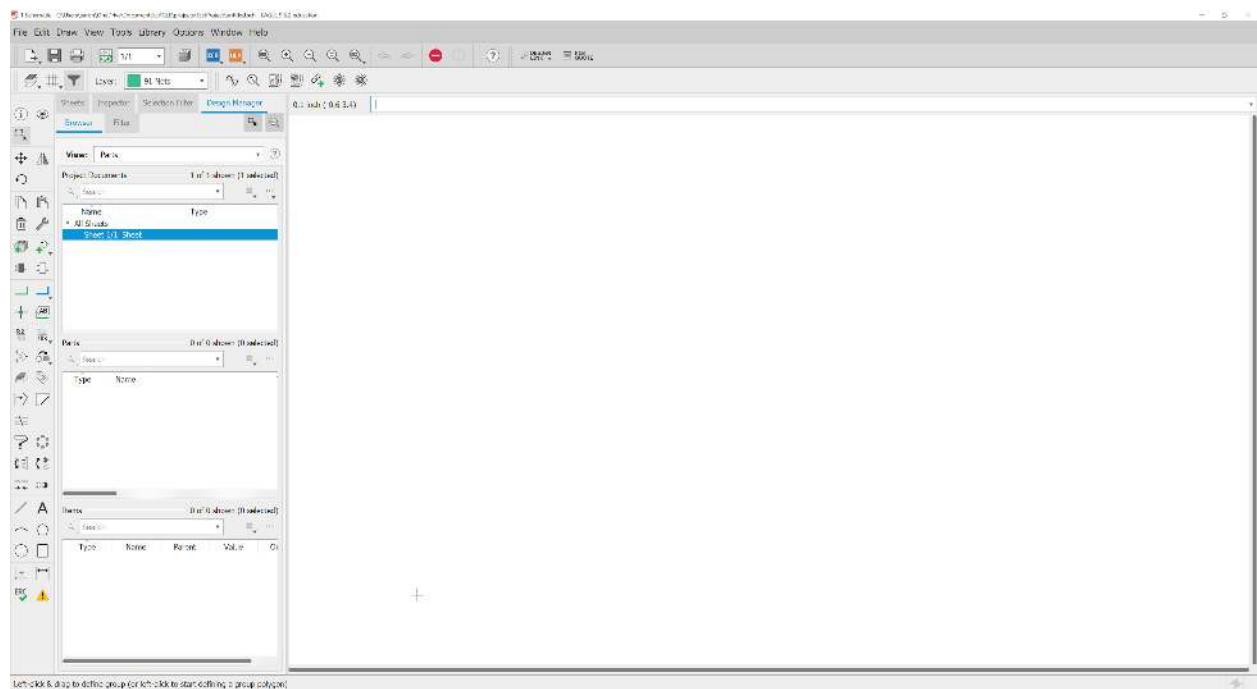


- In this schematic, I've organized the different parts of the project into different areas to make it easier to look at and understand (and ofc review because you'll probably make a mistake somewhere!)
- Once you've finished the schematic, you then make a layout, which takes all of the components from the schematic and organizes them while connecting all the pins that need to be connected together.

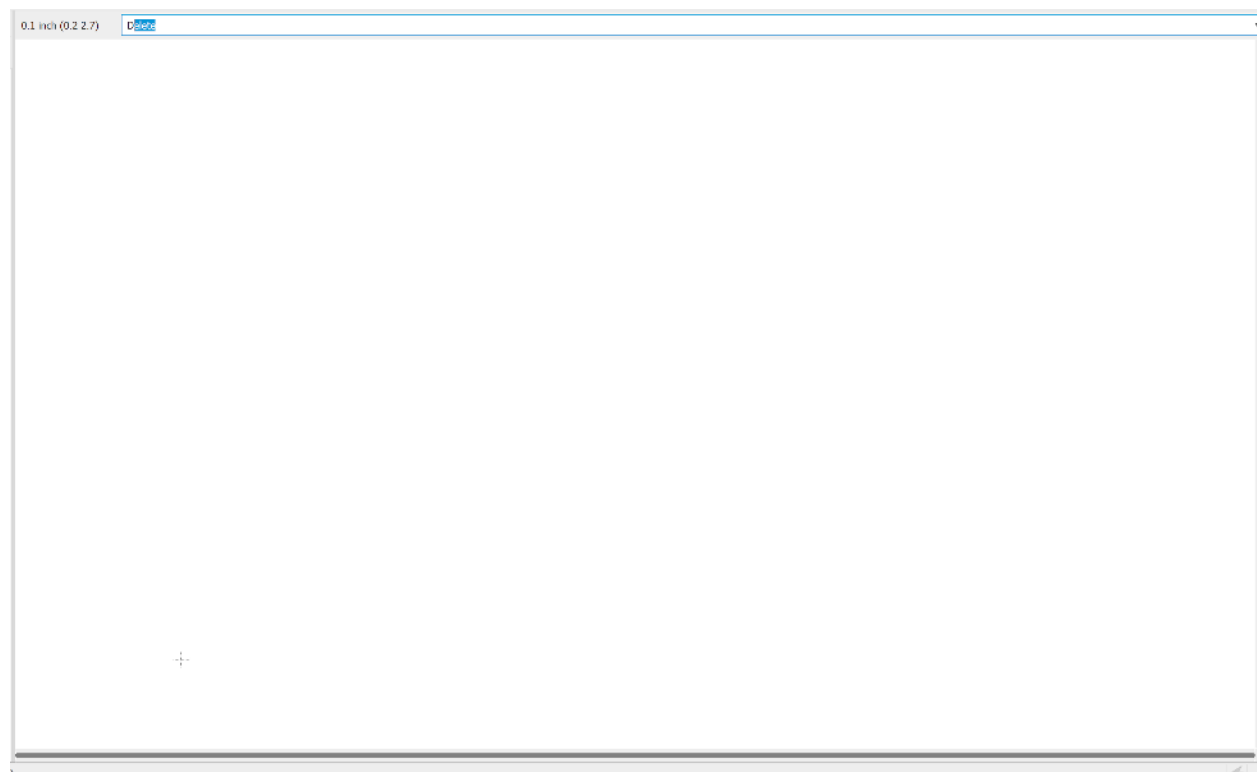
Here's what it looks like:



Okay, now that you know what a layout vs a schematic is, let's start by creating a schematic. Right click on your project that you created and go to "New" -> "Schematic"



Here's what you'll see when you first open it.



This area in the middle is for placing your components.



The area on the left holds all the tools; we'll get to these in a sec

Up on top is the Command Line, which can be accessed by pressing Ctrl+L. I find it super helpful to access tools and switch display modes faster than clicking through a bunch of buttons/menus. I'll go over some useful commands for this in a sec

Useful controls:

Mouse wheel to zoom in/out

Middle mouse+drag to pan

It's a 2d view, so no rotate

## Part 4: Adding your first part

Alright, now that you know where everything is, it's time to get to adding parts into the schematic. Remember that the schematic is only a high-level "picture" of the circuit, so it's okay for physically impossible things like overlapping wires to happen.

Alright, let's start by adding a frame to define the workspace

To do this, you'll need to add a part. You can do this by:



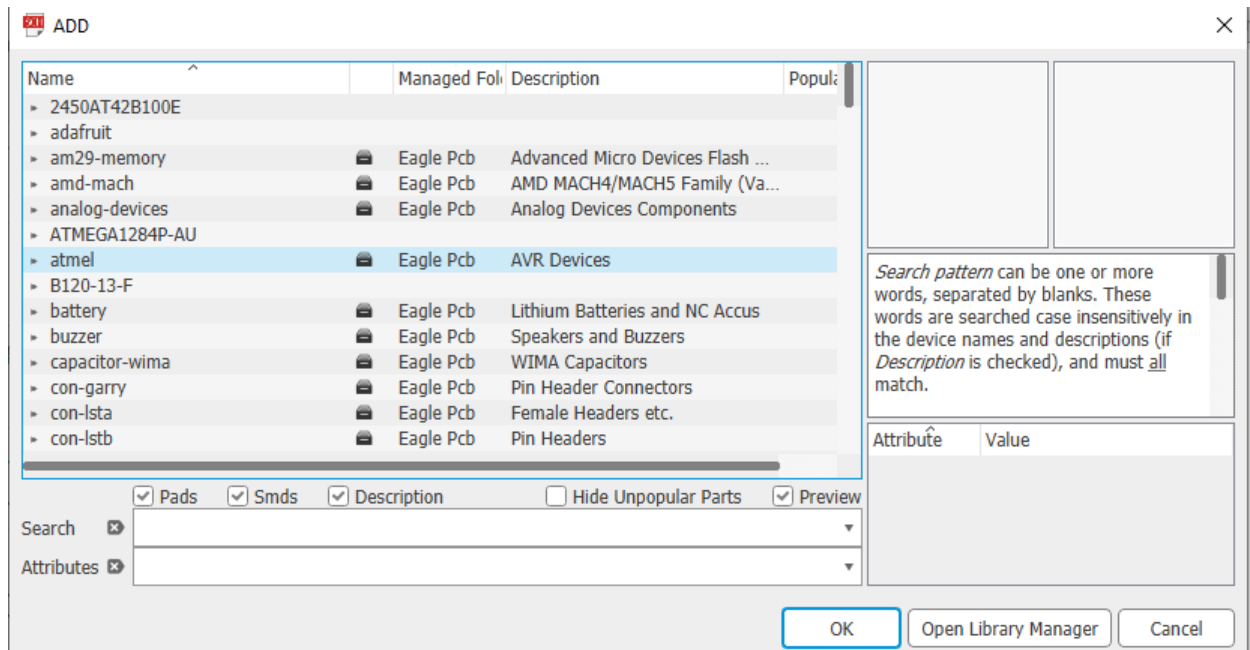
Clicking on the "add part" button

OR

Entering into command line "add"



Once you do this, you'll see a new menu pop up:



Here, you can pick what component you want by either scrolling through the libraries in the menu or by searching for it.

For today, we'll be building a simple battery-powered LED with a pushbutton to control whether it's on or off.

So, let's start by finding the components we need. These come from a few libraries:

## Frames Library

When designing a schematic, it's a good idea to use a frame to constrain the size of your design a bit.

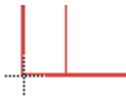
Start by opening the add component menu and then scroll until you find "frames"

| frames            | Eagle Pcb | Frames for Sheet and Layout |
|-------------------|-----------|-----------------------------|
| A3L-LOC           |           | FRAME                       |
| A3P-LOC           |           | A3 Portrait Location        |
| A4-35SC           |           | FRAME                       |
| A4-35SCP          |           | A4 35SCP                    |
| A4-S35CP          |           | FRAME                       |
| A4-SMALL-DOCFIELD |           | FRAME                       |
| A4L-LOC           |           | FRAME                       |
| A4P-LOC           |           | A4P LOC                     |
| A5L-LOC           |           | A5L LOC                     |
| A5P-LOC           |           | A5P LOC                     |
| DINA-DOC          |           | DINA DOC                    |
| DINA3_L           |           | FRAME                       |
| DINA3_P           |           | FRAME                       |

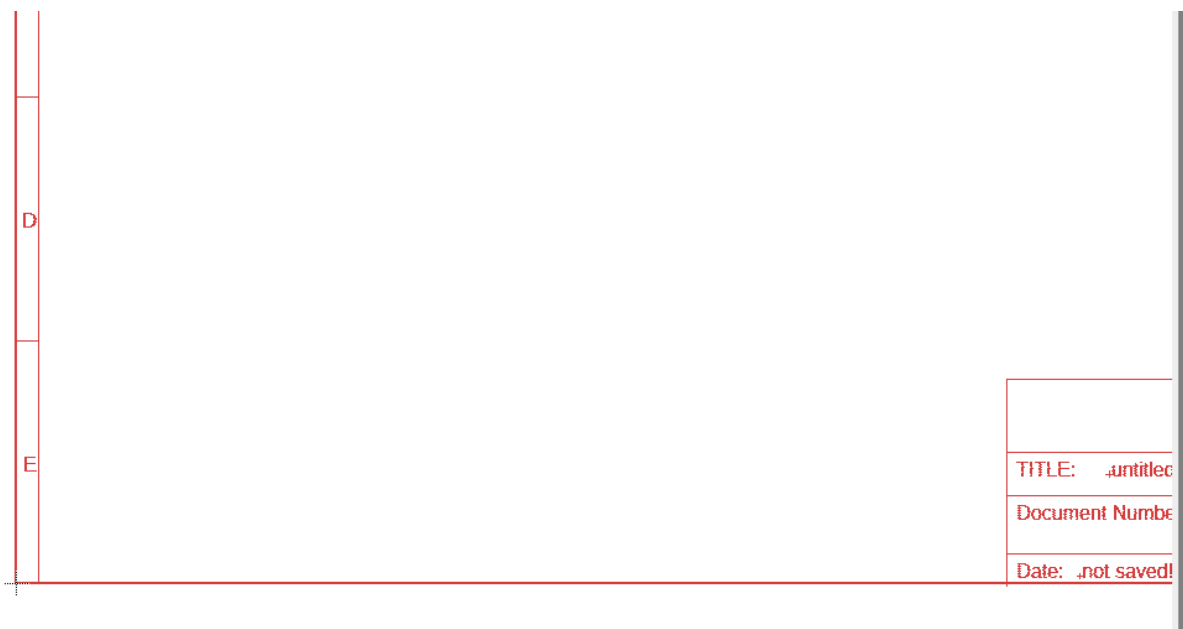
Here you can see the many choices available. We're looking for the standard FRAME\_A\_L, so scroll until you find it

|           |                                   |
|-----------|-----------------------------------|
| DOCSMAL   | DOCSMAL                           |
| FRAME_A_L | FRAME A Size , 8 1/2 x 11 INC...  |
| FRAME B L | FRAME B Size . 11 x 17 INCH. L... |

Once you've found it, double click to select and then click to place over the small origin X in the bottom left of the main screen.



Here's part of the placed frame:

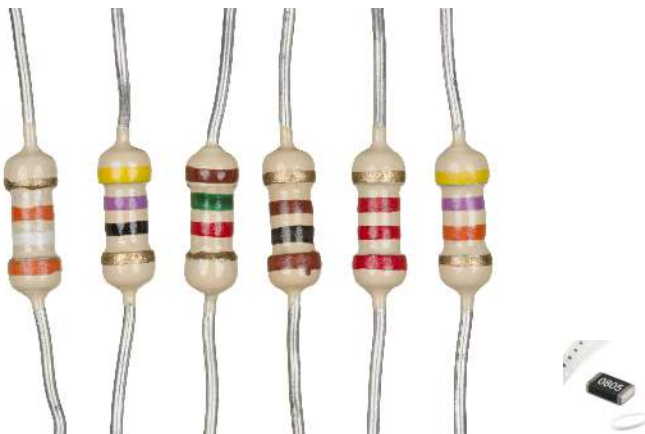


## RCL Library

|                  |           |                                  |
|------------------|-----------|----------------------------------|
| ▼ rcl            | Eagle Pcb | Resistors, Capacitors, Inductors |
| ▶ C-EU           |           | CAPACITOR, European symbol       |
| ▶ C-TRIMM        |           | Trimm capacitor                  |
| ▶ C-US           |           | CAPACITOR, American symbol       |
| ▶ CPOL-EU        |           | POLARIZED CAPACITOR, Europ...    |
| ▶ CPOL-US        |           | POLARIZED CAPACITOR, Ameri...    |
| ▶ CX             |           | X CAPACITOR                      |
| ▶ CY             |           | Y CAPACITOR                      |
| ▶ EL-            |           | BIPOLAR ELECTROLYTIC CAPA...     |
| ▶ L-EU           |           | INDUCTOR, European symbol        |
| ▶ L-US           |           | INDUCTOR, American symbol        |
| ▶ POTENTIOMETER_ |           | Potentiometer                    |
| ▶ R-EU_          |           | RESISTOR, European symbol        |
| ▶ R-TRIMM        |           | Trimm resistor                   |

The RCL (stands for Resistor, Capacitor, L for inductor) library contains most common packages for resistors, capacitors, and inductors.

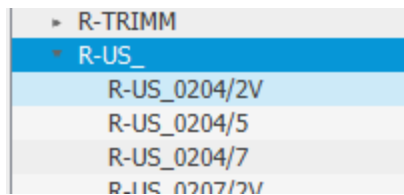
Remember that a package is the physical layout of the component. The two resistors below can have the same value, but different packages; one is through-hole, while one is surface-mount 0805 (width and height in mm) package (0805 is incredibly small compared to through hole component)



The relative size of these two images is roughly accurate to real life, as you can see the surface mount packages are far smaller

Anyways, let's start by adding a 1k resistor. A resistor's resistance is measured in Ohms, so 1k means 1000 ohms of resistance (if this is confusing, check out the intro links I put in the doc)

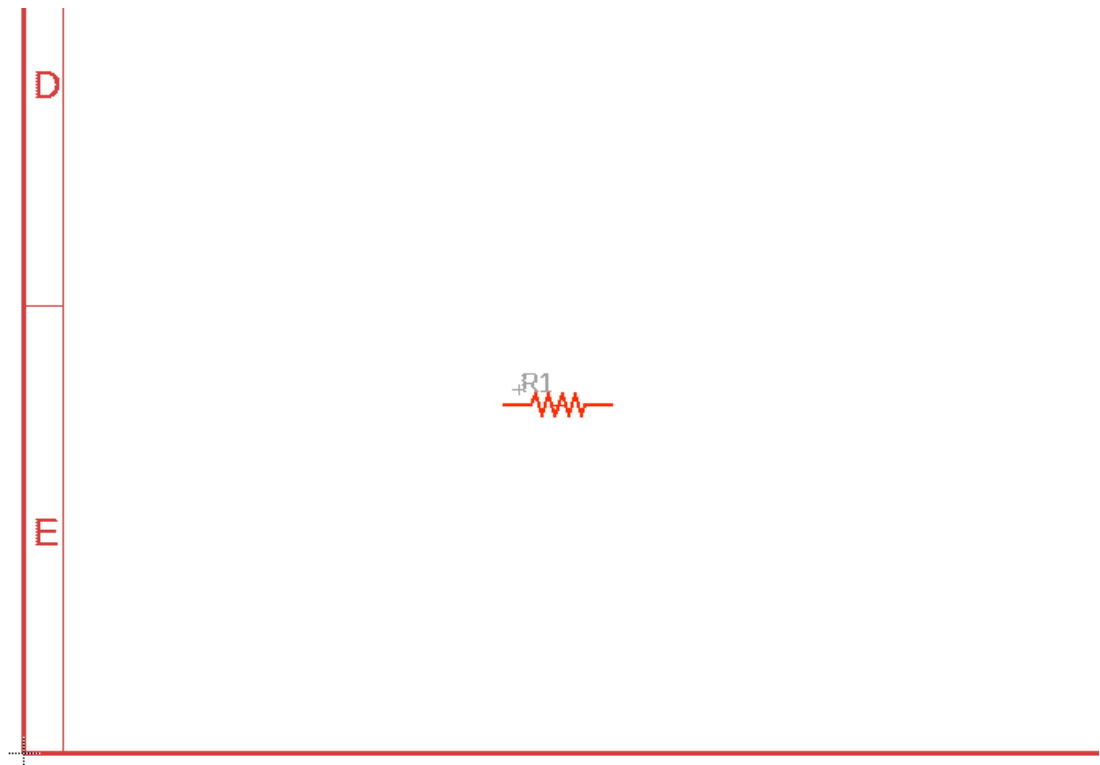




Find the R\_US dropdown and click it; you'll see literally 30-40 options for resistor packages. Scroll down until you find the R-US\_R0805 option



And click on it once to select and then again within the frame you placed to place it in the schematic. Then press ESC twice to escape the menu. You should have this:



Congrats, you've just placed your first part! And it only took 16 pages of this tutorial lol

However, although the resistor is placed, it does not yet have a value to specify its resistance (extremely important)

To add this, click on the value tool

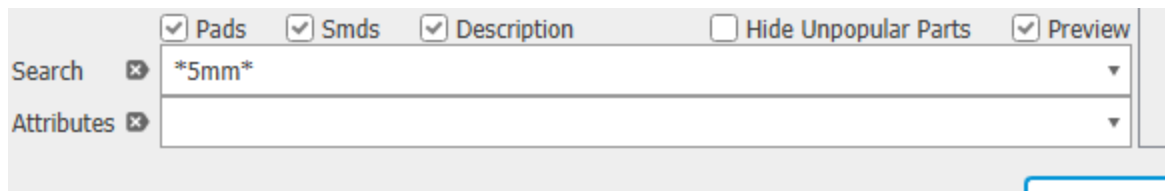


Then on the resistor, and enter 1k for the value. Then press ESC to escape the value tool

## LED Library

Next, we'll add the LED; but we'll do it by a faster method. So far, we've been manually looking up the components. However, using search can make the process go faster. Type in `"*5mm"` to search for standard 5mm components.

Note the use of asterisks before and after the search term - this is important as it means we don't care about what comes before or after the term we're looking for in the part.

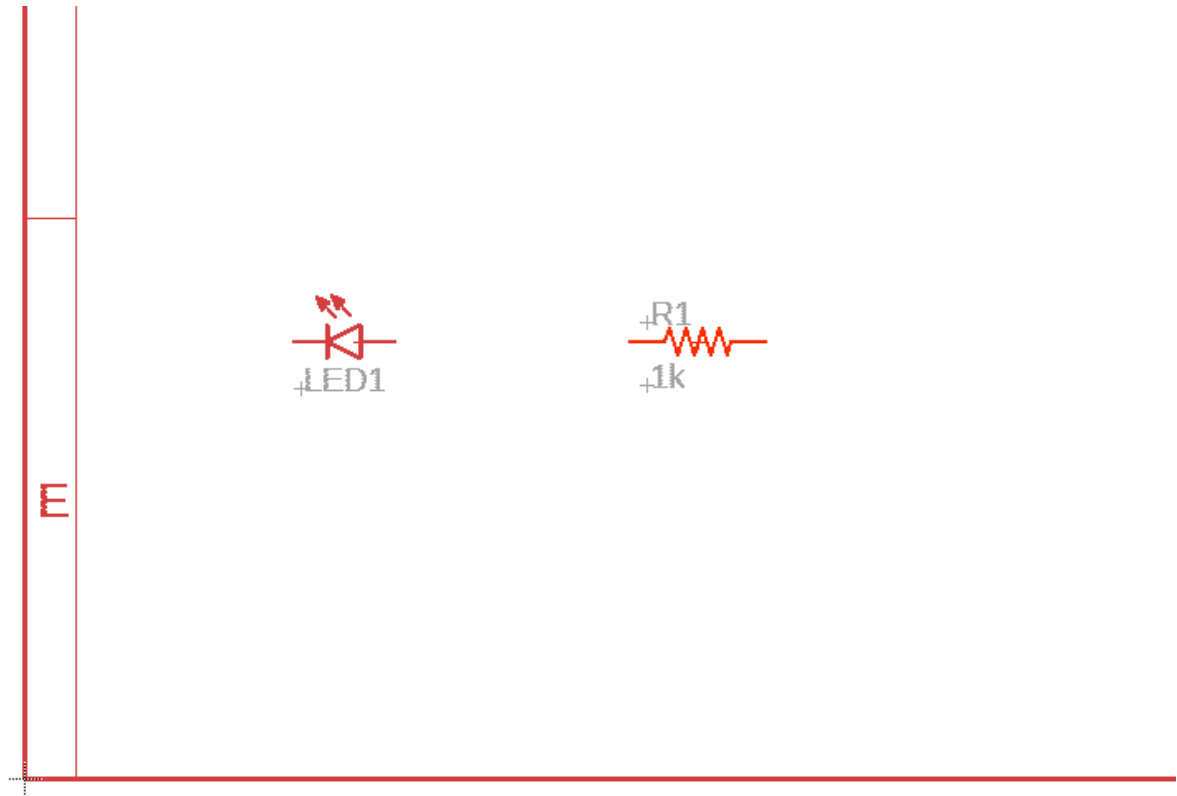


You'll see a lot of results, but from the LED library the results have been narrowed a lot - making it a lot easier to search

| led        |  | Eagle Pcb | LEDs                           |
|------------|--|-----------|--------------------------------|
| DUOLED5MM  |  |           | DUO LED                        |
| L-59EGW-CA |  |           | T-1 3/4 (5mm) BI-COLOR INDI... |
| LED        |  |           | LED                            |
| LED5MM     |  |           | LED5MM                         |

And here is the 5mm LED we needed, easy as that!

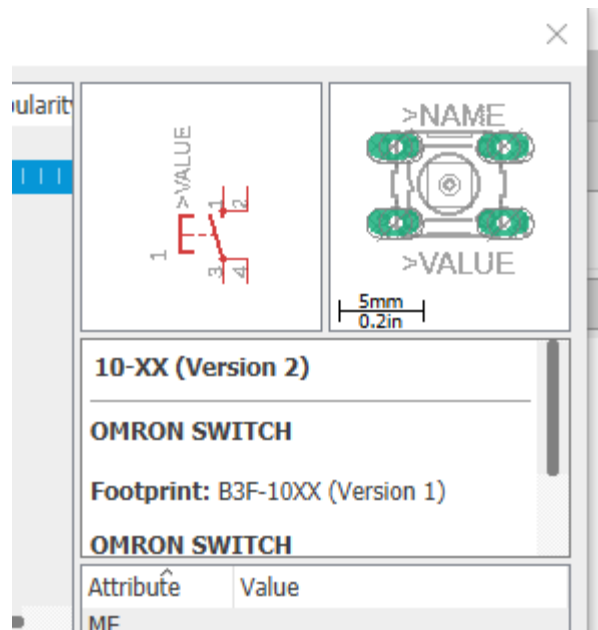
Add it to the schematic. When placing components, Left click+drag to move and right click to rotate



Here's what my schematic looks like. Again, it's only a representation of the circuit, not what it'll look like in real life, so the components use special symbols. In this case, the LED will only work if it's facing the correct direction, so that's indicated by the arrows pointing towards a certain way.

## Switch-Omron Library

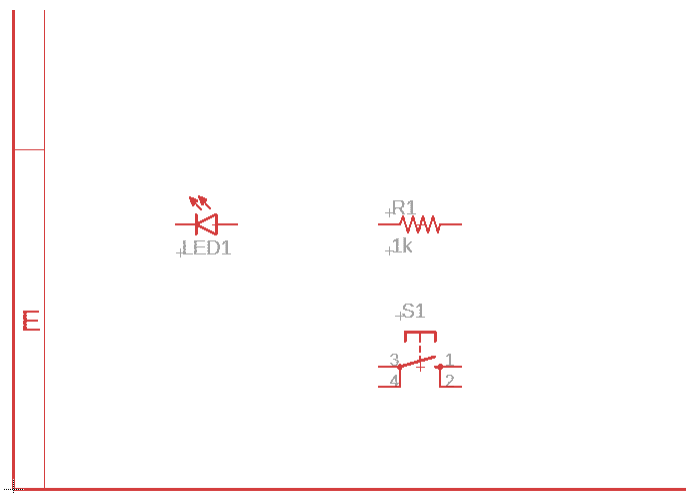
For adding the pushbutton, search “\*10-xx\*” and select the result. Notice



that it'll show two previews, like this:

The first preview is for the schematic, the second is for the physical layout. It'll even show a 3d model if one is available.

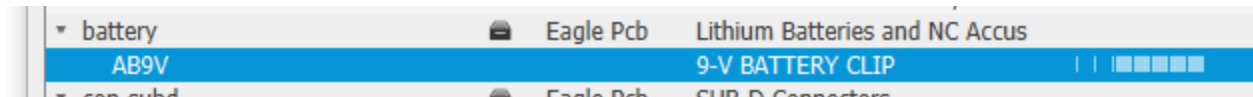
Here's the schematic again:



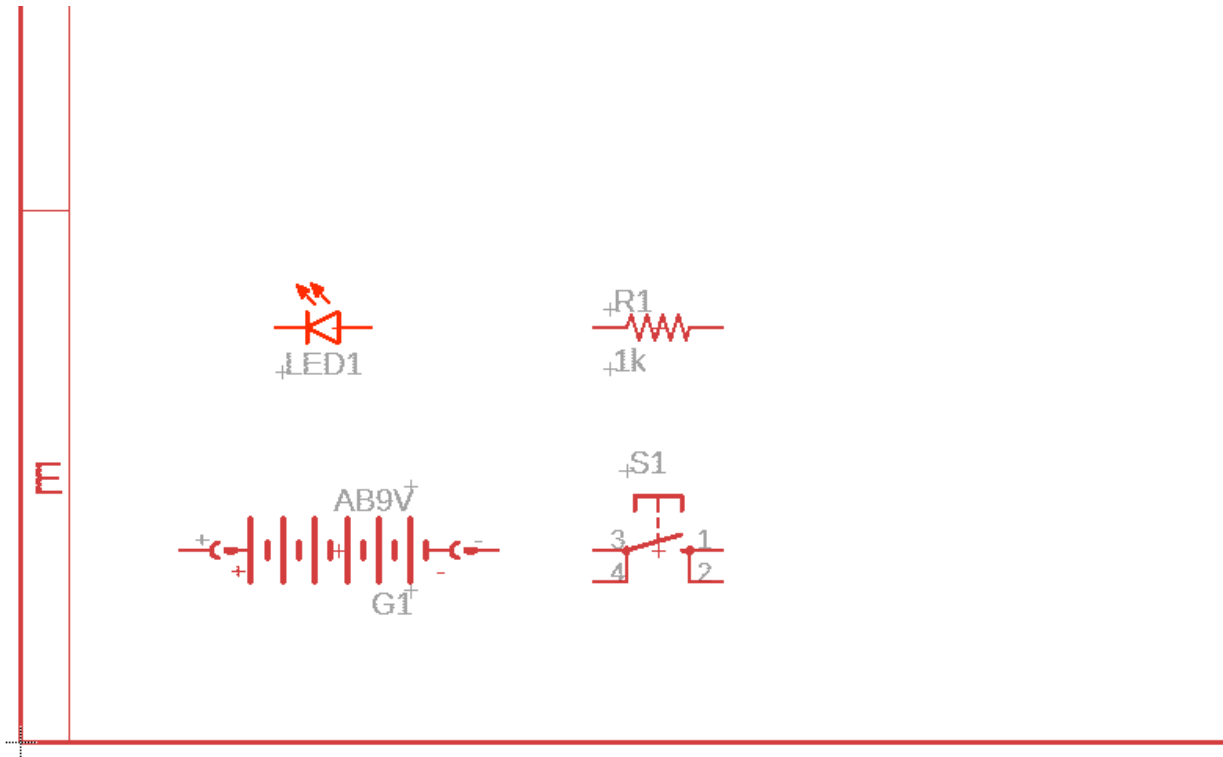
Last component to add is a battery.

## Battery Library

The last thing that we have to add is a battery, in this case a 9v battery. Search for “\*9v\*” and select the 9v battery clip AB9V



Alright, now all the components are added!



Here's the final schematic I have (note that I rotated a bunch of components)

Alright, that's it for adding stuff!

## Part 5: Connecting everything together!

Now that all the components have been added into the schematic, it's time to connect them together using virtual wires. Again, these don't need to be realistic; they can overlap (although it's a bad design practice, so try to avoid it if you can)

In order to connect everything, you need a new tool: Nets

Here's the Net tool

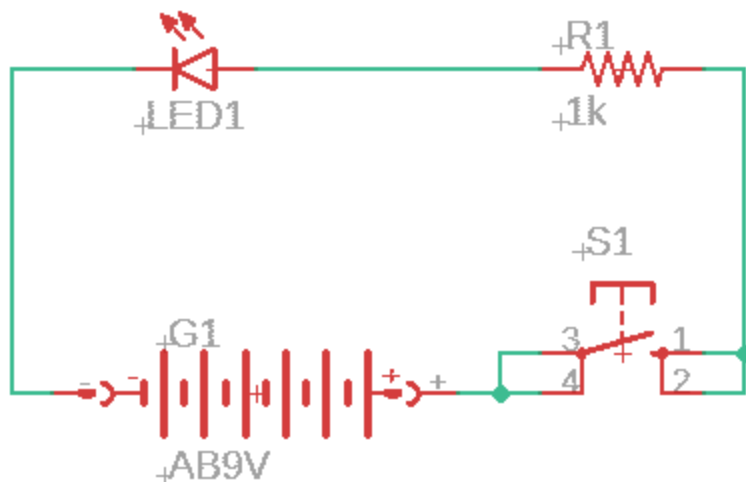


Click on it, then you can click on one of the “outputs” from the components to connect everything together



Here I have connected the anode (positive side) of the LED to the resistor

Now, finish wiring up the circuit as shown (note that the battery was flipped in previous photos. The side marked with a +, aka positive, must be oriented towards the side of the LED without the “bar”)



Here's the completed circuit! This diagram represents an LED that can be turned on and off with a pushbutton.

# Conclusion

That wraps up day 1! Hope y'all liked it. Please post a screenshot of the circuit you've created in the thread that this document was linked from. Make sure to save what you have, as it'll be used in next week's tutorial.