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

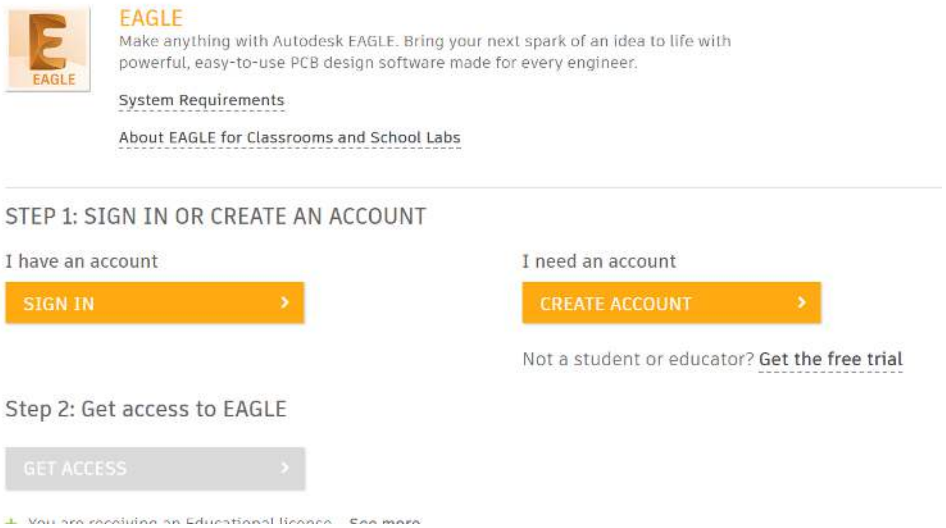
End of Day Goal: Know how to navigate Eagle schematic designer and libraries and create a very simple schematic

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

I need an account

Not a student or educator? Get the free trial

Step 2: Get access to EAGLE

+ 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)**~~

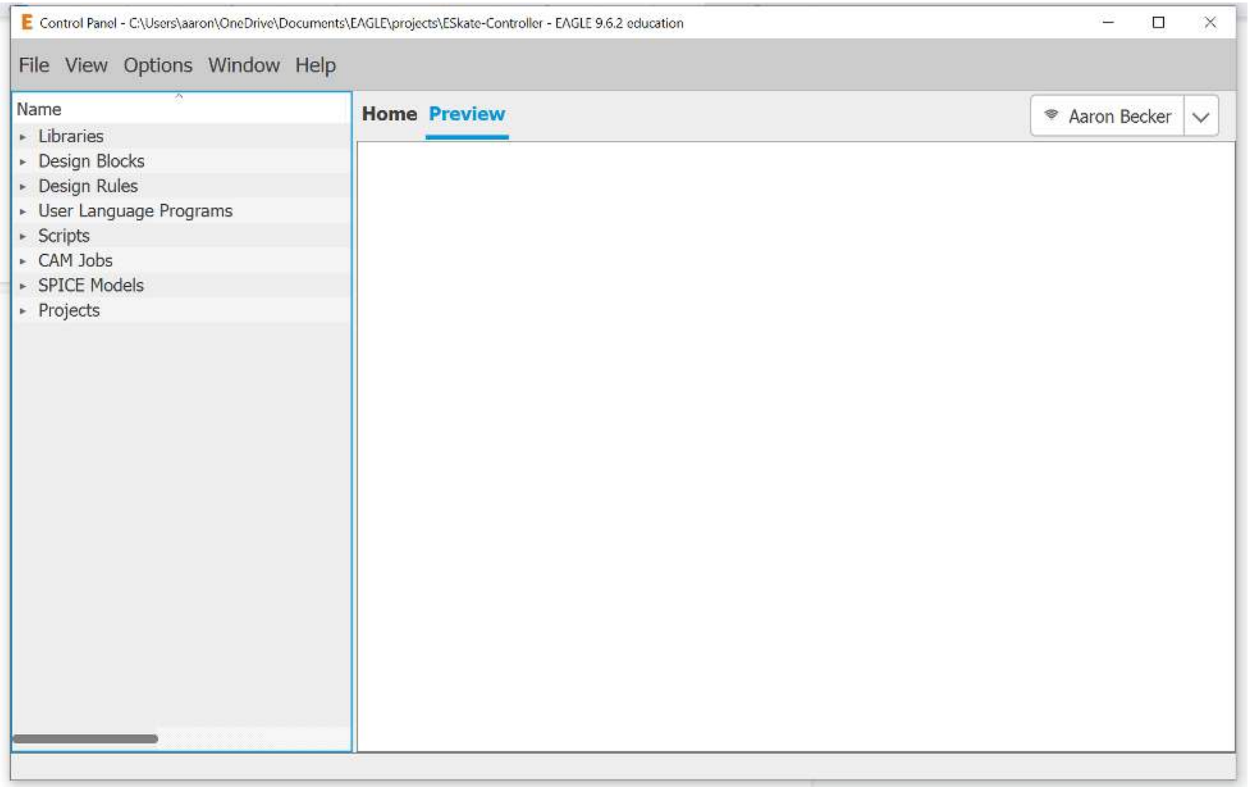
~~Once your account has been approved as an "education" account, download and install Eagle according to the autodesk website. Once it's open, proceed to next step~~

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

Part 2: First Look at the Control Panel

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

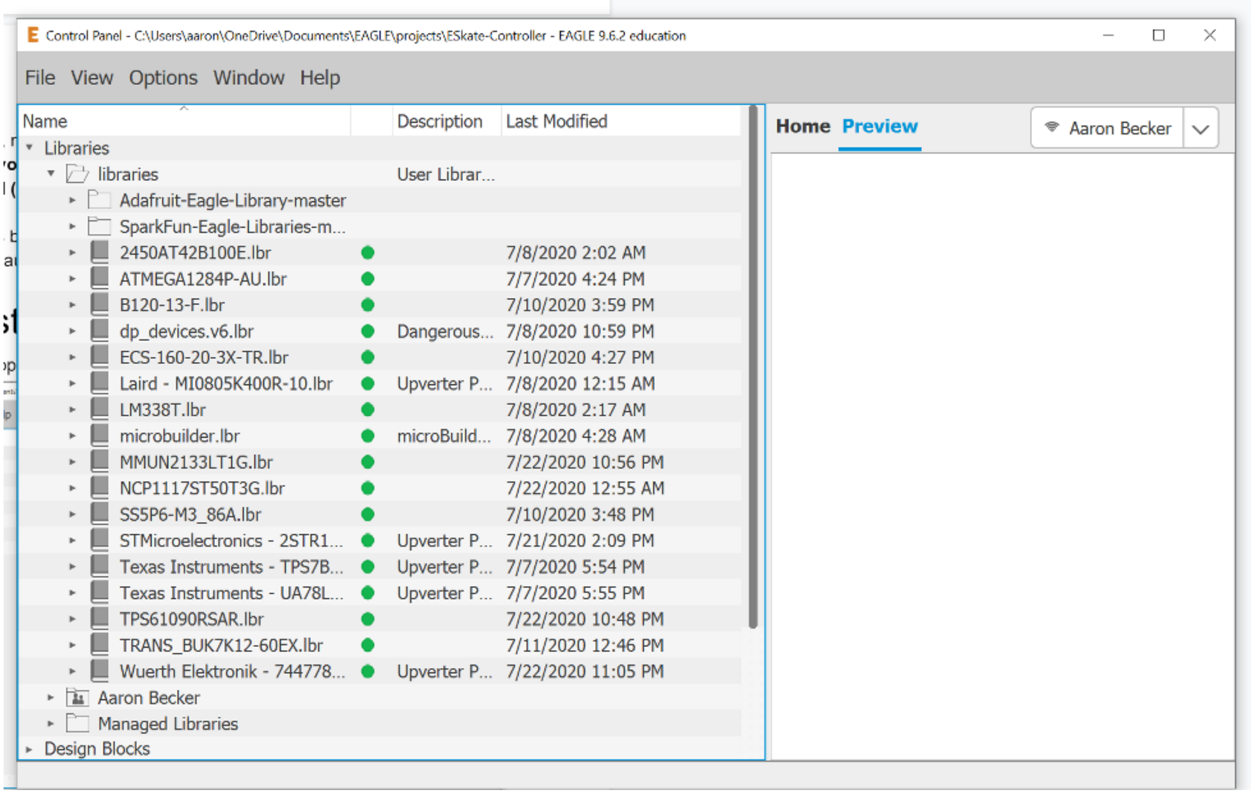


This window is called the "Control Panel". I think of it kinda like file explorer or Finder for Eagle; you can manage your projects, install new libraries (which hold different components).

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

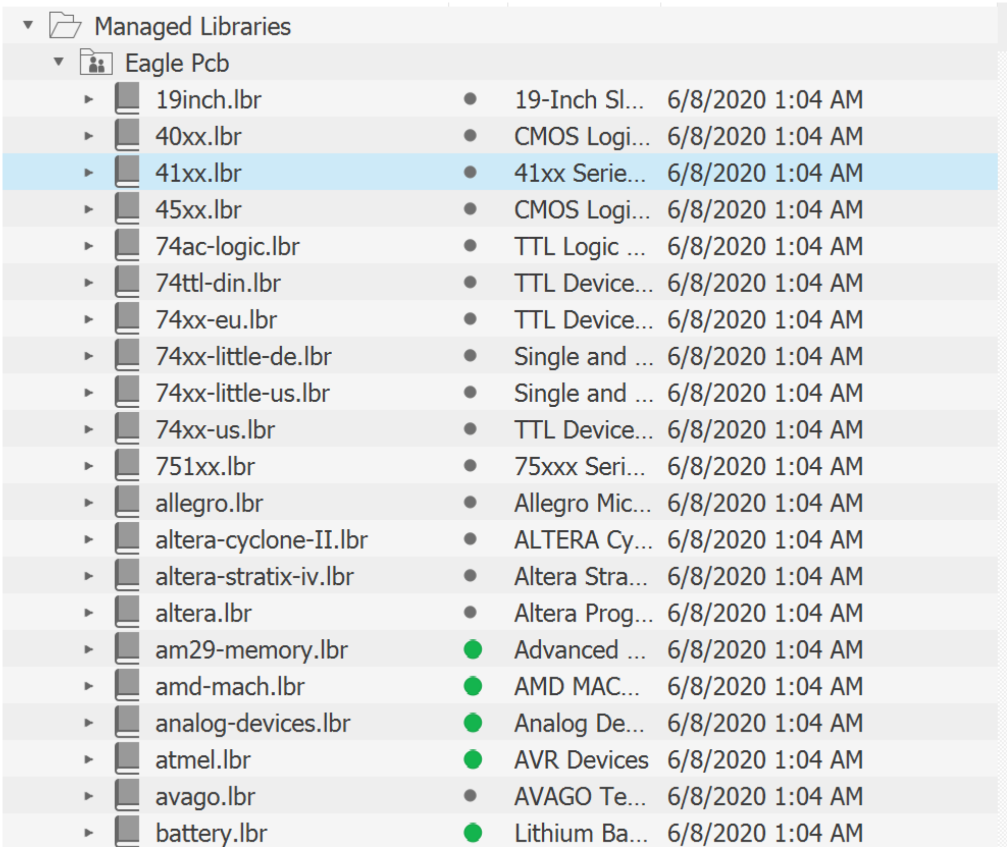
Libraries

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


Control Panel - C:\Users\aaron\OneDrive\Documents\EAGLE\projects\ESkate-Controller - EAGLE 9.6.2 education
File
View
Options
Window
Help
Home
Preview
Aaron Becker

Name		Description	Last Modified
Libraries			
libraries		User Librar...	
Adafruit-Eagle-Library-master			
SparkFun-Eagle-Libraries-m...			
2450AT42B100E.lbr	●		7/8/2020 2:02 AM
ATMEGA1284P-AU.lbr	●		7/7/2020 4:24 PM
B120-13-F.lbr	●		7/10/2020 3:59 PM
dp_devices.v6.lbr	●	Dangerous...	7/8/2020 10:59 PM
ECS-160-20-3X-TR.lbr	●		7/10/2020 4:27 PM
Laird - MI0805K400R-10.lbr	●	Upverter P...	7/8/2020 12:15 AM
LM338T.lbr	●		7/8/2020 2:17 AM
microbuilder.lbr	●	microBuild...	7/8/2020 4:28 AM
MMUN2133LT1G.lbr	●		7/22/2020 10:56 PM
NCP1117ST50T3G.lbr	●		7/22/2020 12:55 AM
SS5P6-M3_86A.lbr	●		7/10/2020 3:48 PM
STMicroelectronics - 2STR1...	●	Upverter P...	7/21/2020 2:09 PM
Texas Instruments - TPS7B...	●	Upverter P...	7/7/2020 5:54 PM
Texas Instruments - UA78L...	●	Upverter P...	7/7/2020 5:55 PM
TPS61090RSAR.lbr	●		7/22/2020 10:48 PM
TRANS_BUK7K12-60EX.lbr	●		7/11/2020 12:46 PM
Wuerth Elektronik - 744778...	●	Upverter P...	7/22/2020 11:05 PM
Aaron Becker			
Managed Libraries			
Design Blocks			

Here’s some of Eagle’s default libraries:



▾ Managed Libraries

▾ Eagle Pcb

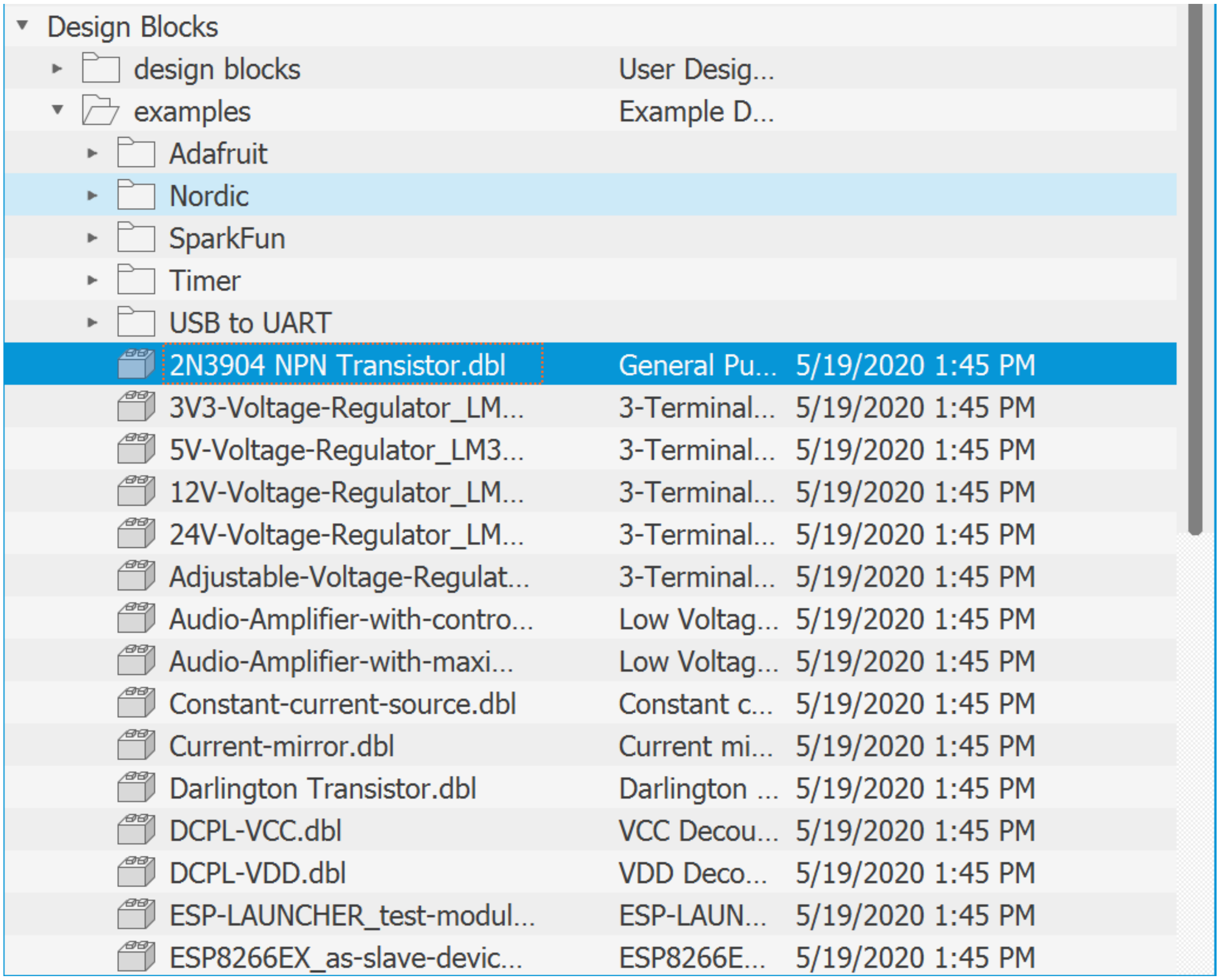
▸ 19inch.lbr	●	19-Inch Sl...	6/8/2020 1:04 AM
▸ 40xx.lbr	●	CMOS Logi...	6/8/2020 1:04 AM
▸ 41xx.lbr	●	41xx Serie...	6/8/2020 1:04 AM
▸ 45xx.lbr	●	CMOS Logi...	6/8/2020 1:04 AM
▸ 74ac-logic.lbr	●	TTL Logic ...	6/8/2020 1:04 AM
▸ 74ttl-din.lbr	●	TTL Device...	6/8/2020 1:04 AM
▸ 74xx-eu.lbr	●	TTL Device...	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

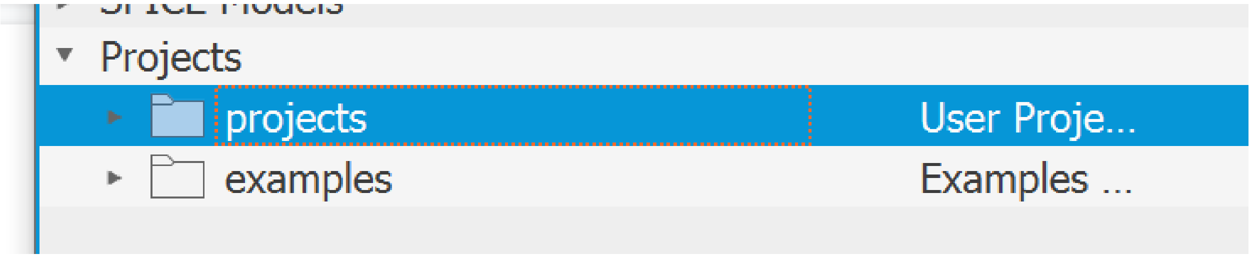
SPICE Models

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)

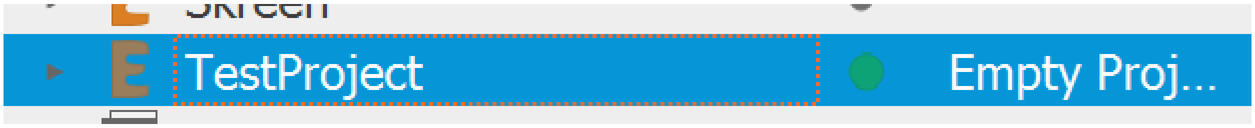
Projects

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



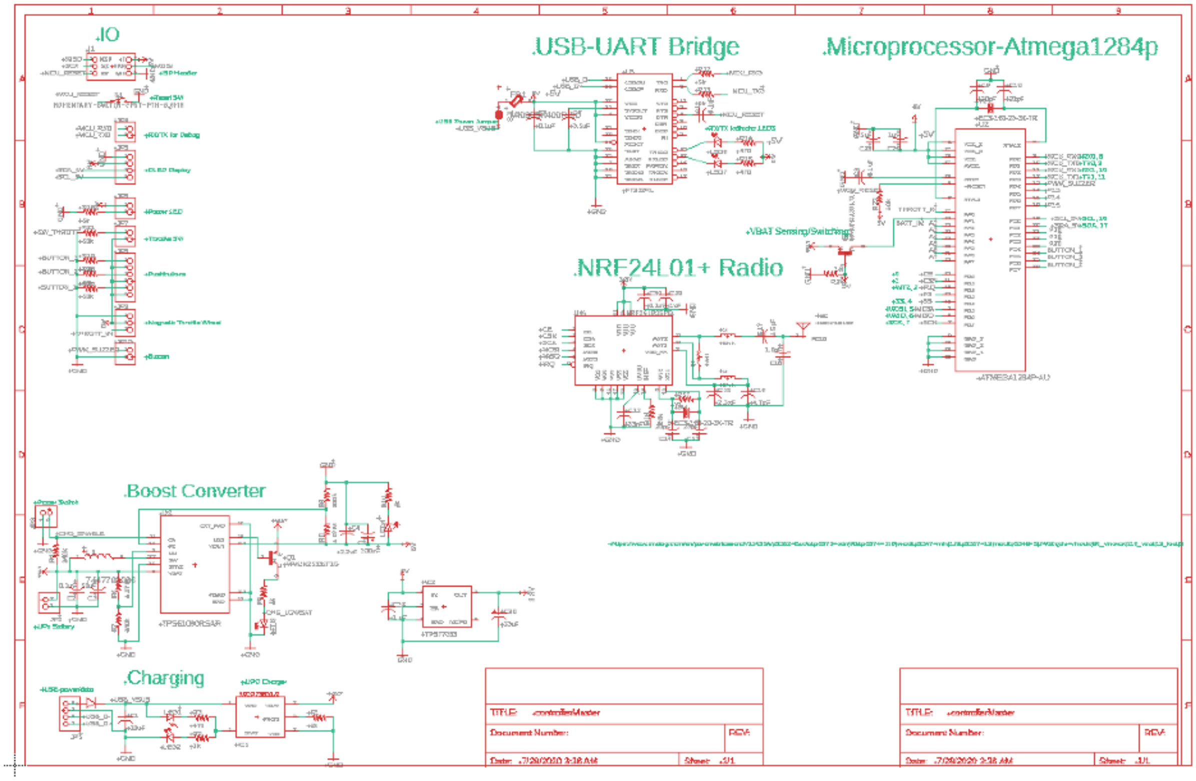
Open the Projects dropdown and right click on the lowercase “projects” subfolder, then select “New Project”



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

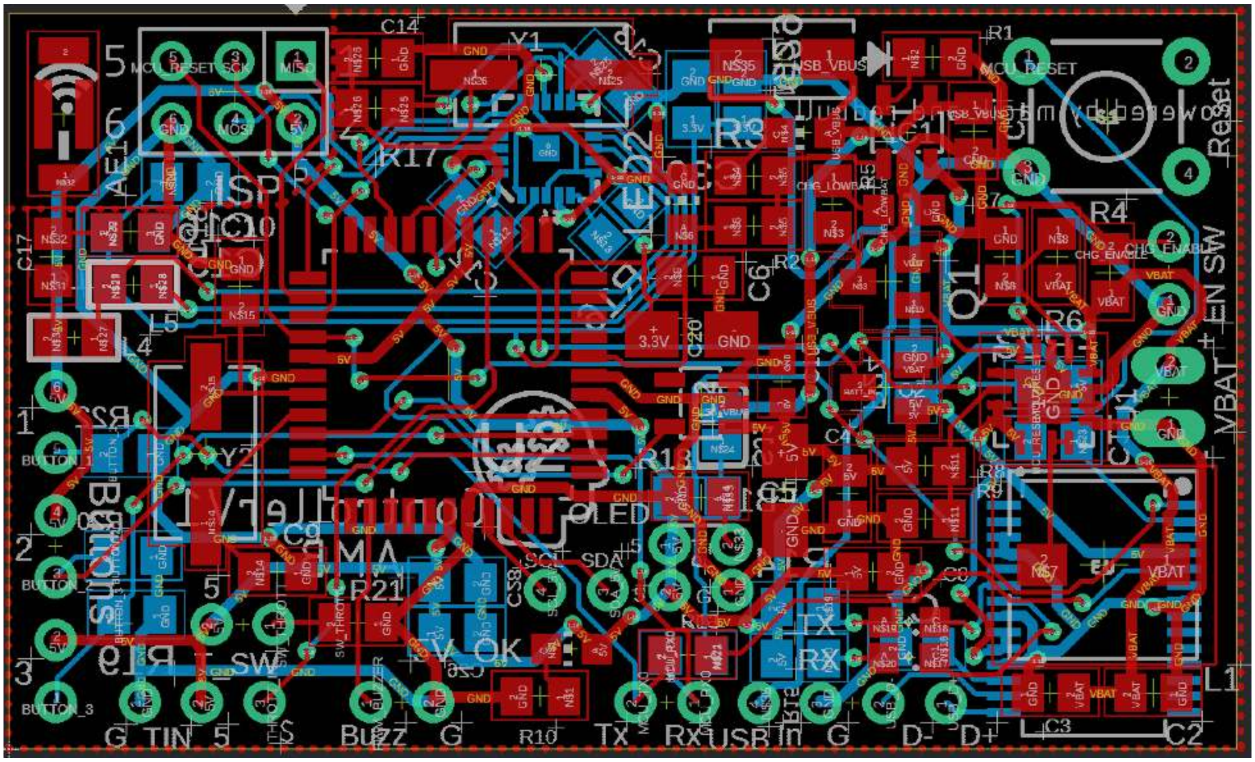
For today, we’re going to be starting with the schematic. Quick explanation of the different parts of a PCB project:

- 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:



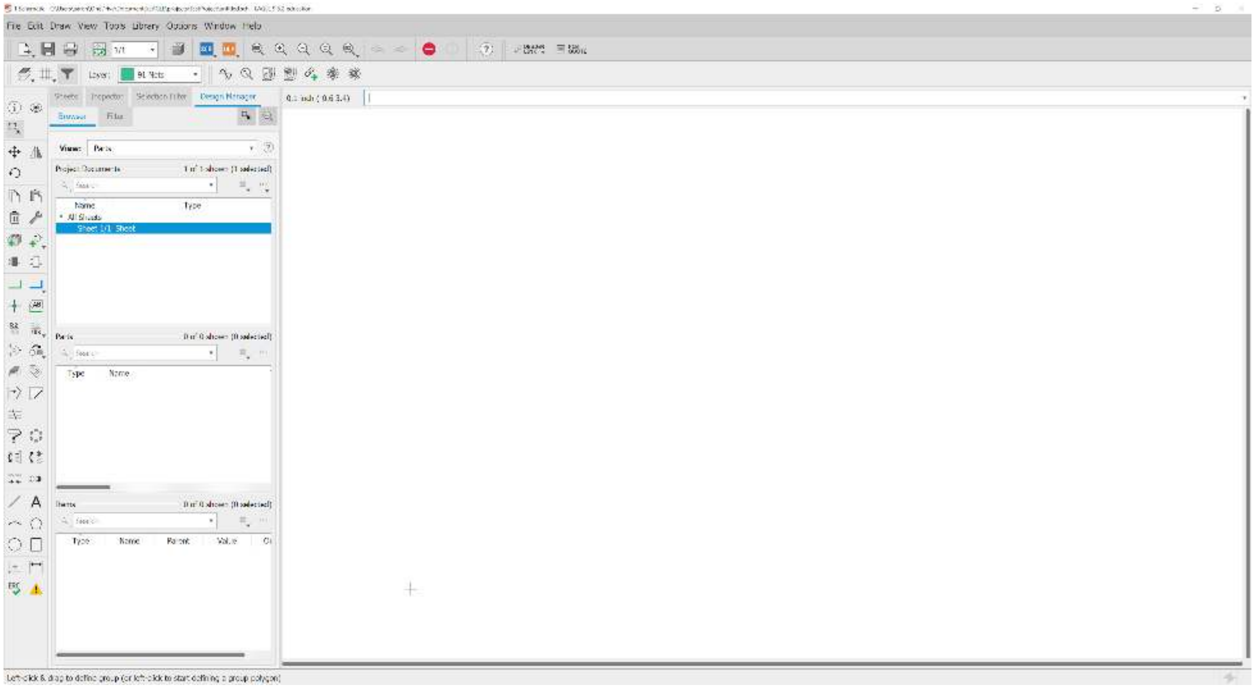
- In this schematic, I've organized the different parts of the project into different areas to make it easier to look at and understand (and ofc review because you'll probably make a mistake somewhere!)
- Once you've finished the schematic, you then make a layout, which takes all of the components from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

Here’s what it looks like:

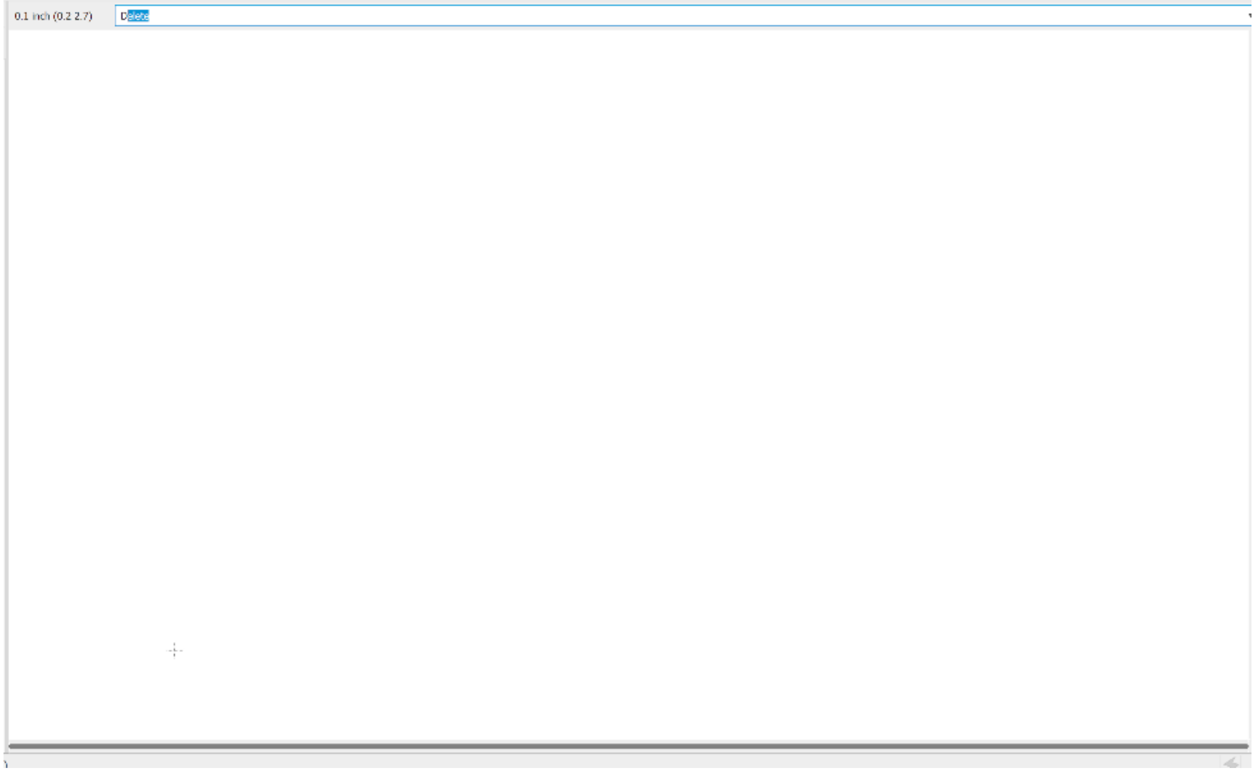


- In this particular layout, you can see a number of important features like holes for the components and even smaller green holes called vias which I’ll get into when we learn layouts. For now, all of the red pads (regular flat areas that are used to attach a component) are on the top side of the board, and all of the blue is on the bottom. In this way, you have both sides to make your design work (in standard PCB manufacturing at least)

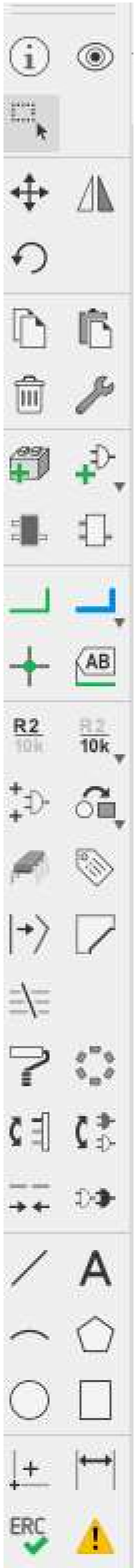
Okay, now that you know what a layout vs a schematic is, let’s start by creating a schematic. Right click on your project that you created and go to “New” -> “Schematic”


File Edit Draw View Tools Library Options Window Help
Layer: 91 Nets
Sheets Inspector Selection Filter Design Manager
Browser Filter
View: Parts
Project Documents
Name Type
All Sheets
Sheet 1/1 Sheet
Parts
Type Name
Type Name Parent Value
Left-click & drag to define group (or left-click to start defining a group polygon)

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

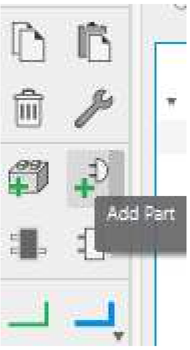
Useful controls:
Mouse wheel to zoom in/out
Middle mouse+drag to pan
It's a 2d view, so no rotate

Part 4: Adding your first part

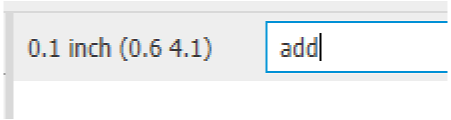
Alright, now that you know where everything is, it's time to get to adding parts into the schematic. Remember that the schematic is only a high-level "picture" of the circuit, so it's okay for physically impossible things like overlapping wires to happen.

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

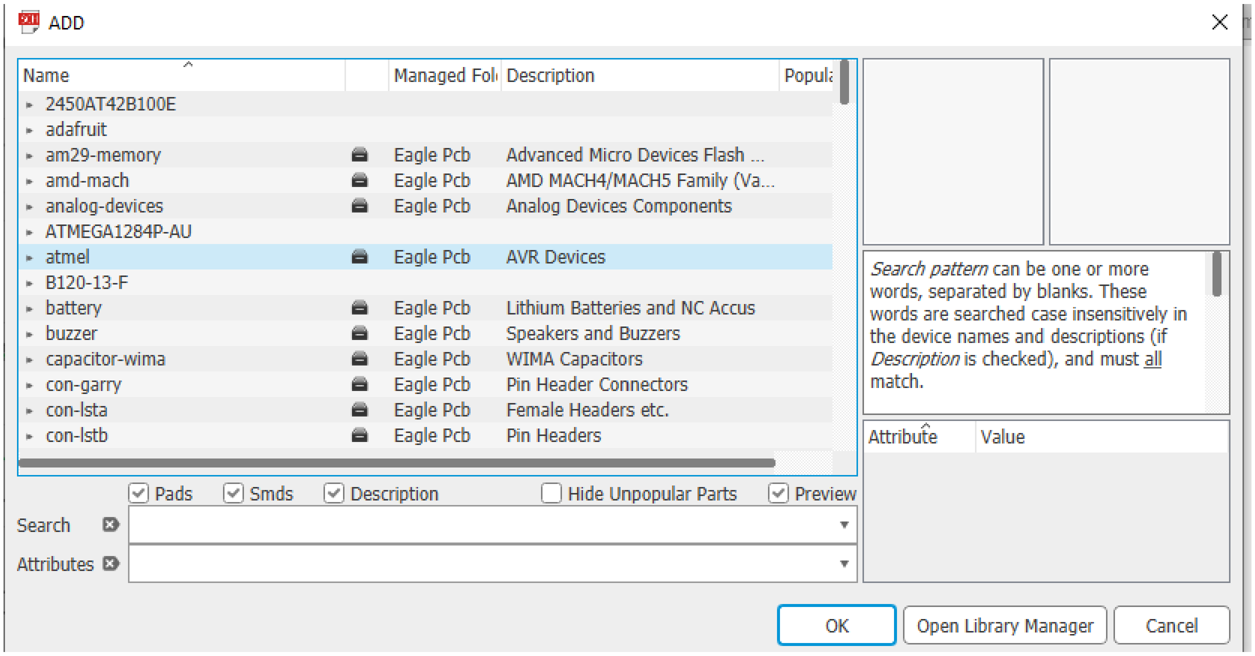
To do this, you'll need to add a part. You can do this by:



Clicking on the "add part" button
OR
Entering into command line "add"


0.1 inch (0.6 4.1)
add

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



Name	Managed Fol	Description	Popul
2450AT42B100E			
adafruit			
am29-memory	Eagle Pcb	Advanced Micro Devices Flash ...	
amd-mach	Eagle Pcb	AMD MACH4/MACH5 Family (Va...	
analog-devices	Eagle Pcb	Analog Devices Components	
ATMEGA1284P-AU			
atmel	Eagle Pcb	AVR Devices	
B120-13-F			
battery	Eagle Pcb	Lithium Batteries and NC Accus	
buzzer	Eagle Pcb	Speakers and Buzzers	
capacitor-wima	Eagle Pcb	WIMA Capacitors	
con-garry	Eagle Pcb	Pin Header Connectors	
con-lsta	Eagle Pcb	Female Headers etc.	
con-lstb	Eagle Pcb	Pin Headers	

Search pattern can be one or more words, separated by blanks. These words are searched case insensitively in the device names and descriptions (if *Description* is checked), and must all match.

☑ Pads ☑ Smds ☑ Description ☐ Hide Unpopular Parts ☑ Preview

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

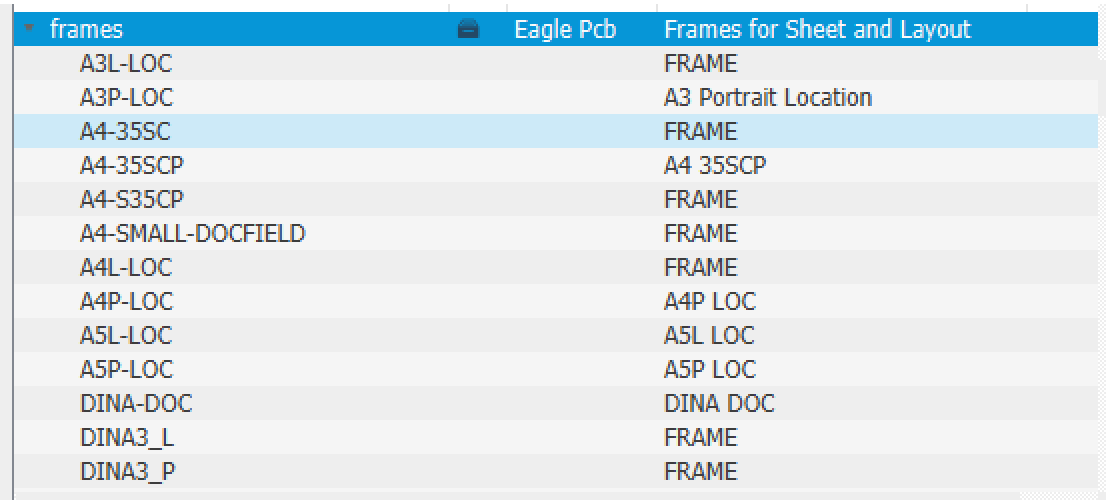
For today, we'll be building a simple battery-powered LED with a pushbutton to control whether it's on or off.

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

Frames Library

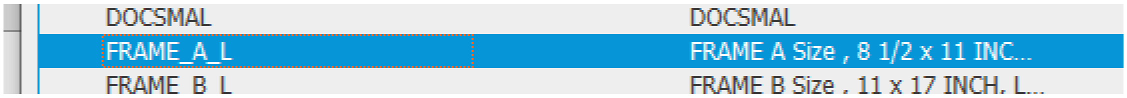
When designing a schematic, it's a good idea to use a frame to constrain the size of your design a bit.

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



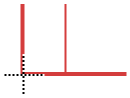
frames	Eagle Pcb	Frames for Sheet and Layout
A3L-LOC		FRAME
A3P-LOC		A3 Portrait Location
A4-35SC		FRAME
A4-35SCP		A4 35SCP
A4-S35CP		FRAME
A4-SMALL-DOCFIELD		FRAME
A4L-LOC		FRAME
A4P-LOC		A4P LOC
A5L-LOC		A5L LOC
A5P-LOC		A5P LOC
DINA-DOC		DINA DOC
DINA3_L		FRAME
DINA3_P		FRAME

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

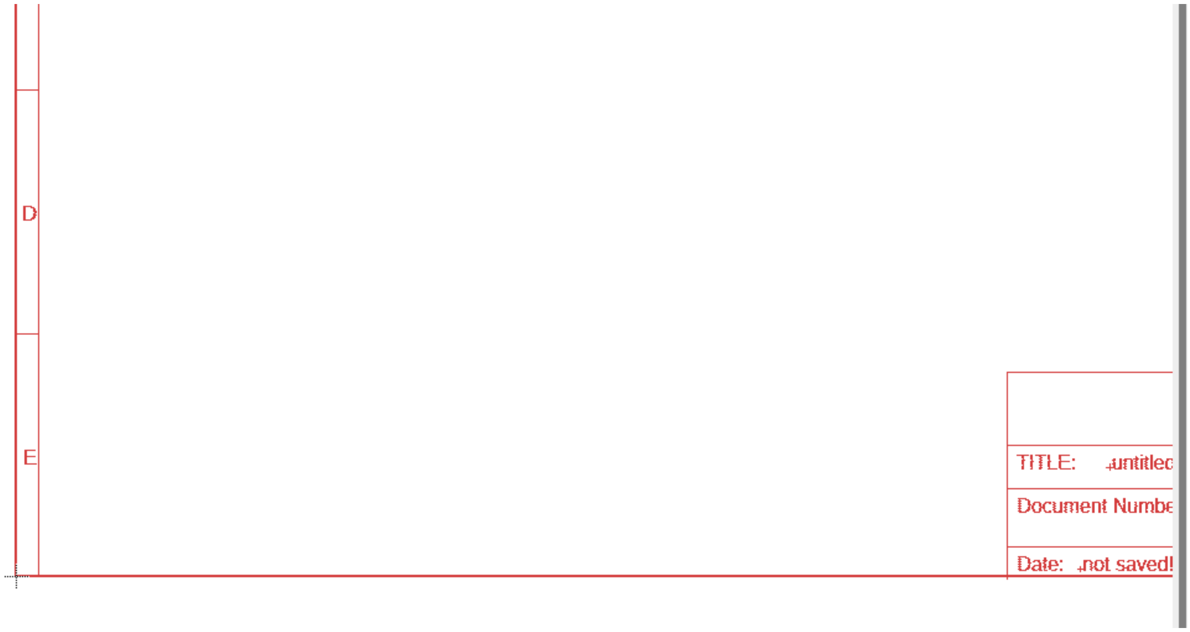


DOCSMAL	DOCSMAL
FRAME_A_L	FRAME A Size , 8 1/2 x 11 INC...
FRAME_B_L	FRAME B Size , 11 x 17 INCH, L...

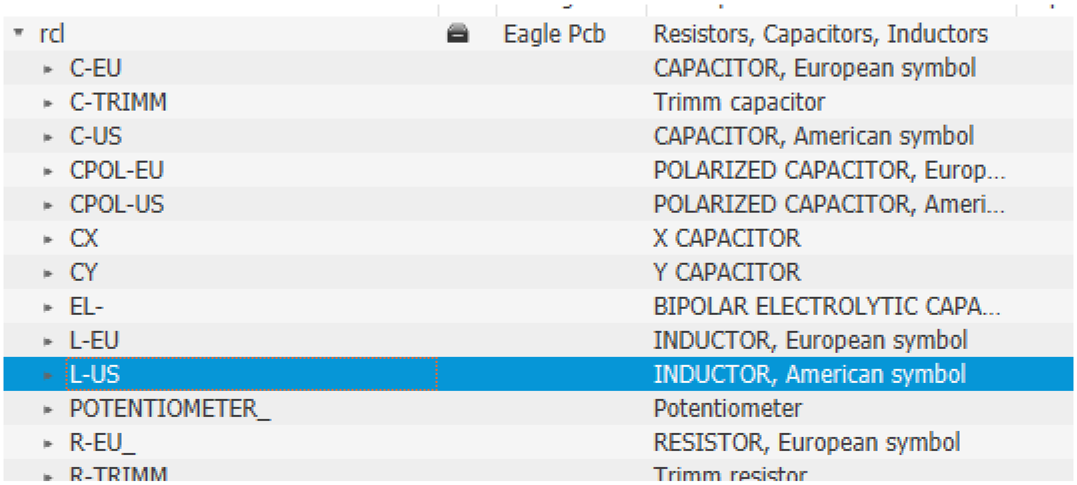
Once you've found it, double click to select and then click to place over the small origin X in the bottom left of the main screen.



Here's part of the placed frame:

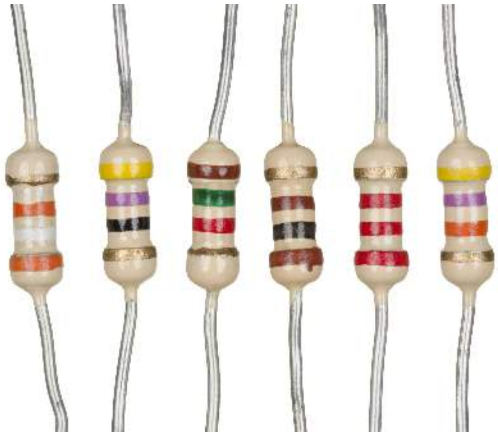


RCL Library



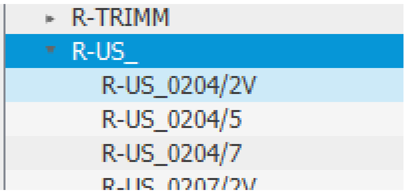
▾ rcl	Eagle Pcb	Resistors, Capacitors, Inductors
▸ C-EU		CAPACITOR, European symbol
▸ C-TRIMM		Trimm capacitor
▸ C-US		CAPACITOR, American symbol
▸ CPOL-EU		POLARIZED CAPACITOR, Europ...
▸ CPOL-US		POLARIZED CAPACITOR, Ameri...
▸ CX		X CAPACITOR
▸ CY		Y CAPACITOR
▸ EL-		BIPOLAR ELECTROLYTIC CAPA...
▸ 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)



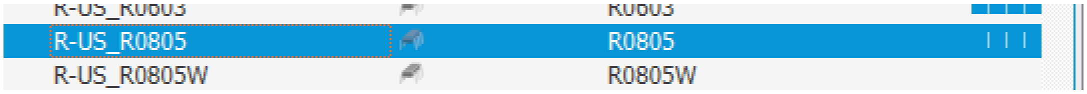
The relative size of these two images is roughly accurate to real life, as you can see the surface mount packages are far smaller

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)



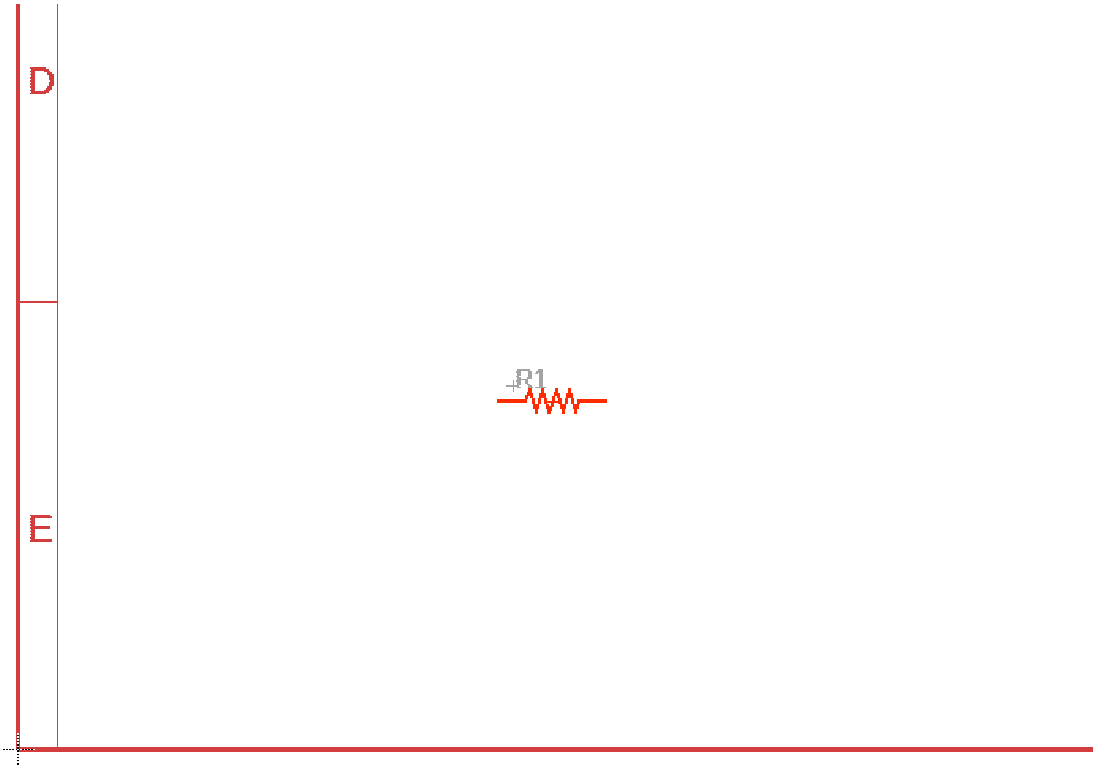
- R-TRIMM
- R-US_
  - R-US_0204/2V
  - R-US_0204/5
  - R-US_0204/7
  - R-US_0207/2V

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



R-US_R0603	R0603
R-US_R0805	R0805
R-US_R0805W	R0805W

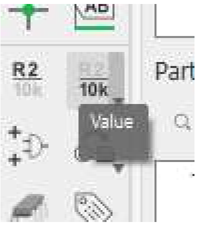
And click on it once to select and then again within the frame you placed to place it in the schematic. Then press ESC twice to escape the menu. You should have this:



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

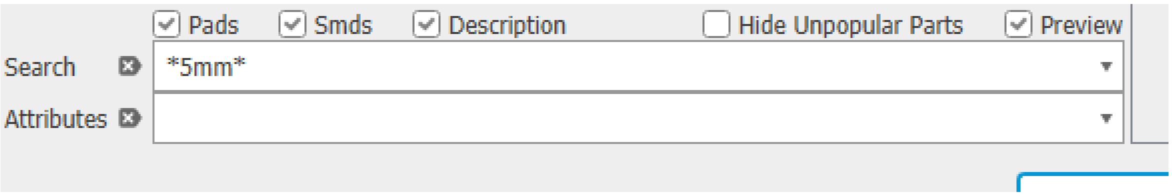


Then on the resistor, and enter 1k for the value. Then press ESC to escape the value tool

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.

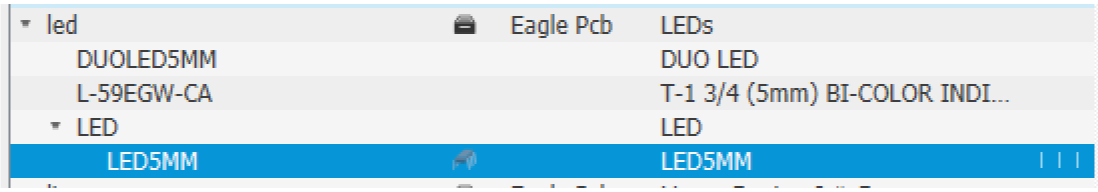


☑ Pads ☑ Smds ☑ Description □ Hide Unpopular Parts ☑ Preview

Search: *5mm*

Attributes:

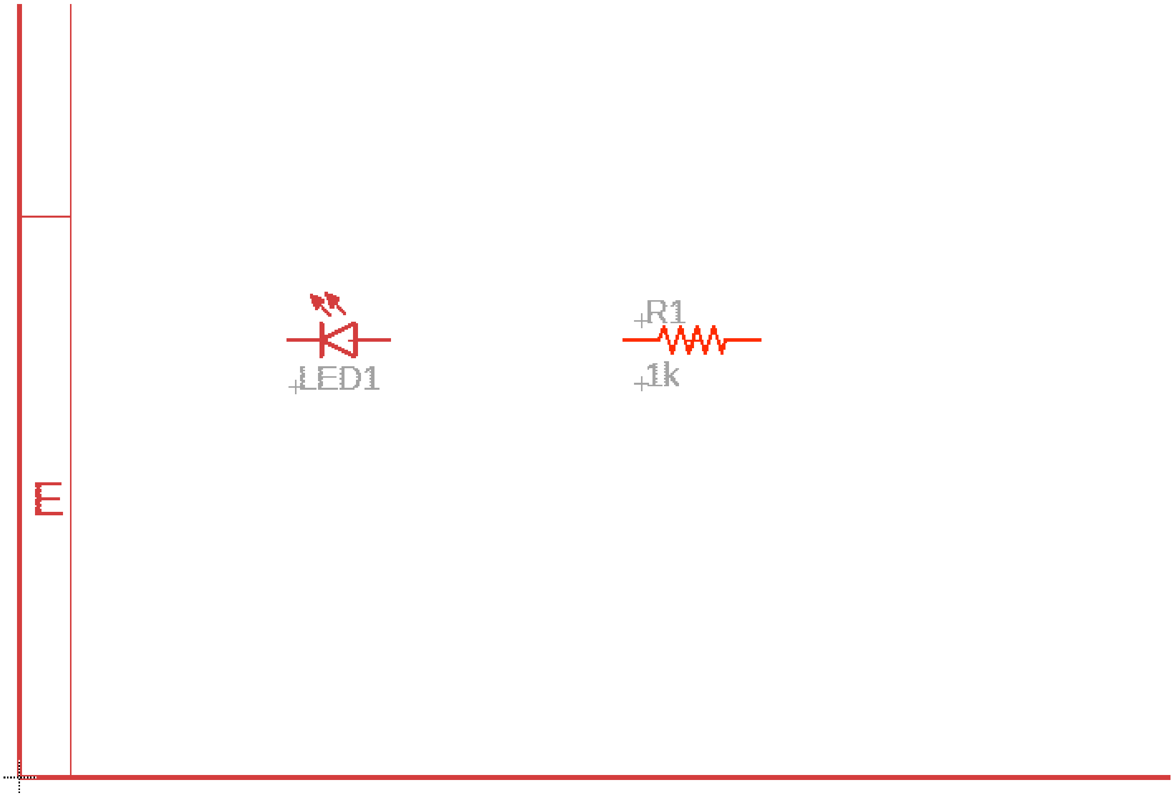
You'll see a lot of results, but from the LED library the results have been narrowed a lot - making it a lot easier to search



▾ 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!

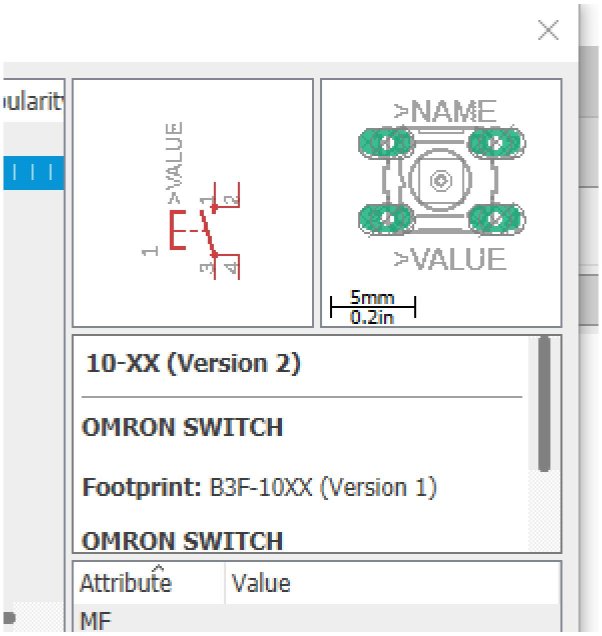
Add it to the schematic. When placing components, Left click+drag to move and right click to rotate



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:



10-XX (Version 2)

OMRON SWITCH

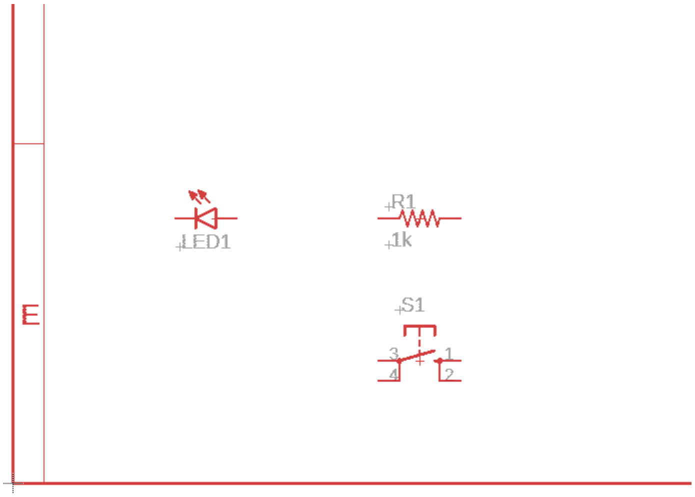
Footprint: B3F-10XX (Version 1)

OMRON SWITCH

Attribute	Value
MF	

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.

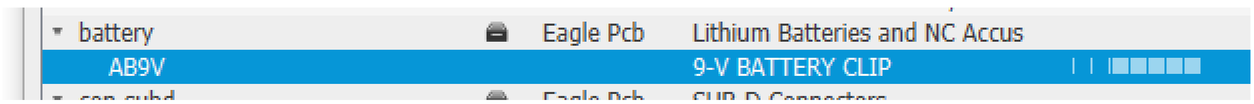
Here’s the schematic again:



Last component to add is a battery.

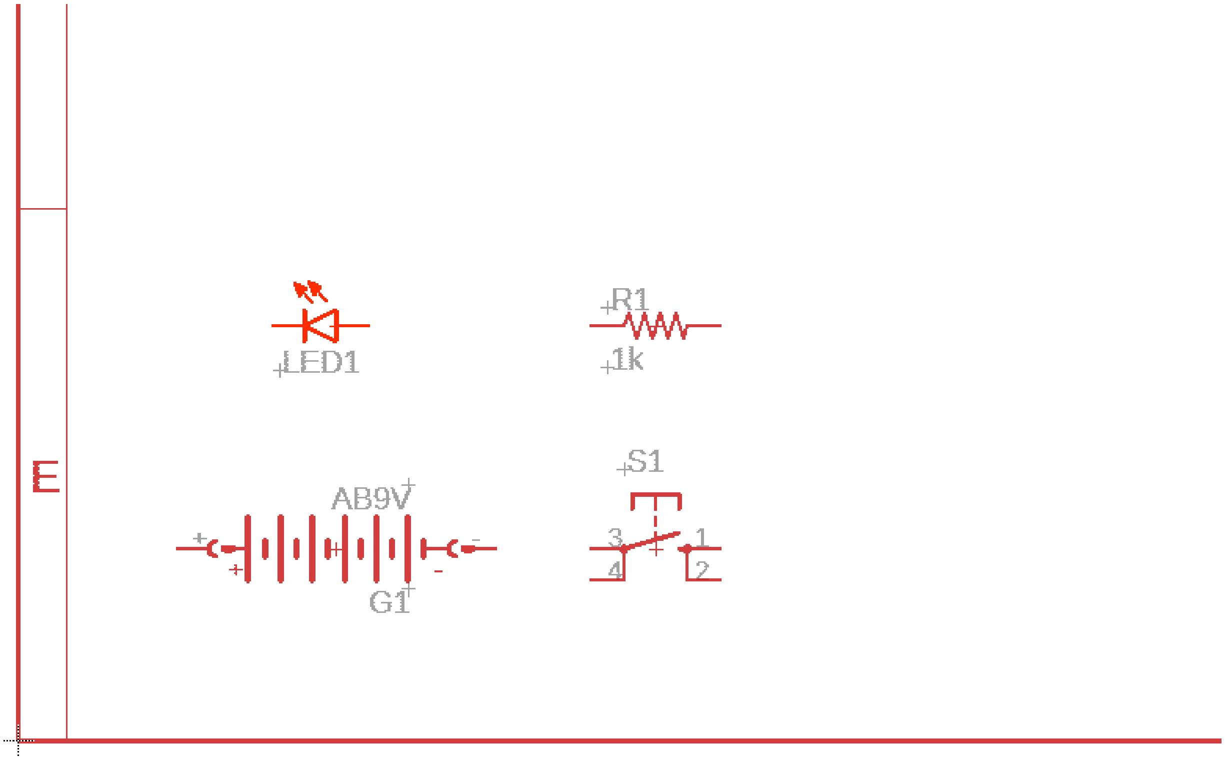
Battery Library

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



battery	Eagle Pcb	Lithium Batteries and NC Accus
AB9V		9-V BATTERY CLIP
con-subd	Eagle Pcb	SUB-D Connectors

Alright, now all the components are added!



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

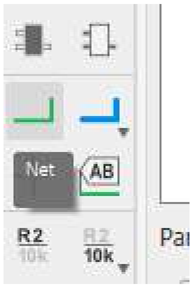
Alright, that’s it for adding stuff!

Part 5: Connecting everything together!

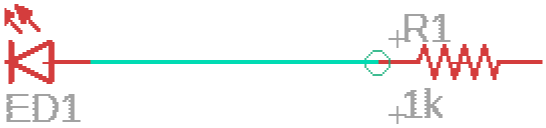
Now that all the components have been added into the schematic, it’s time to connect them together using virtual wires. Again, these don’t need to be realistic; they can overlap (although it’s a bad design practice, so try to avoid it if you can)

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

Here’s the Net tool

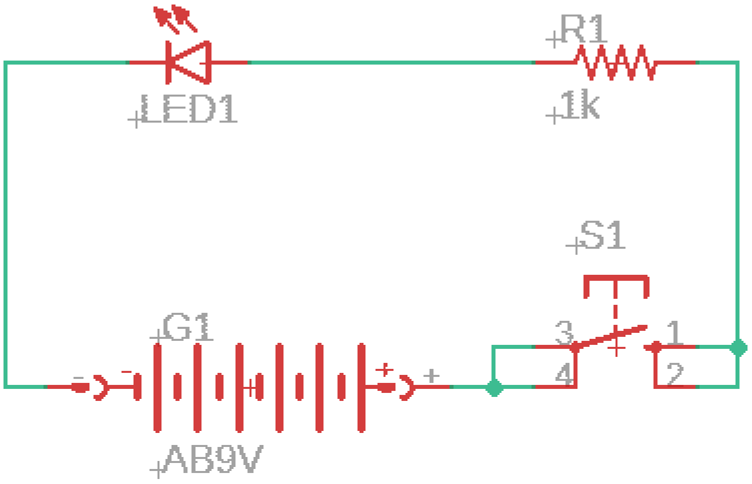


Click on it, then you can click on one of the “outputs” from the components to connect everything together



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.