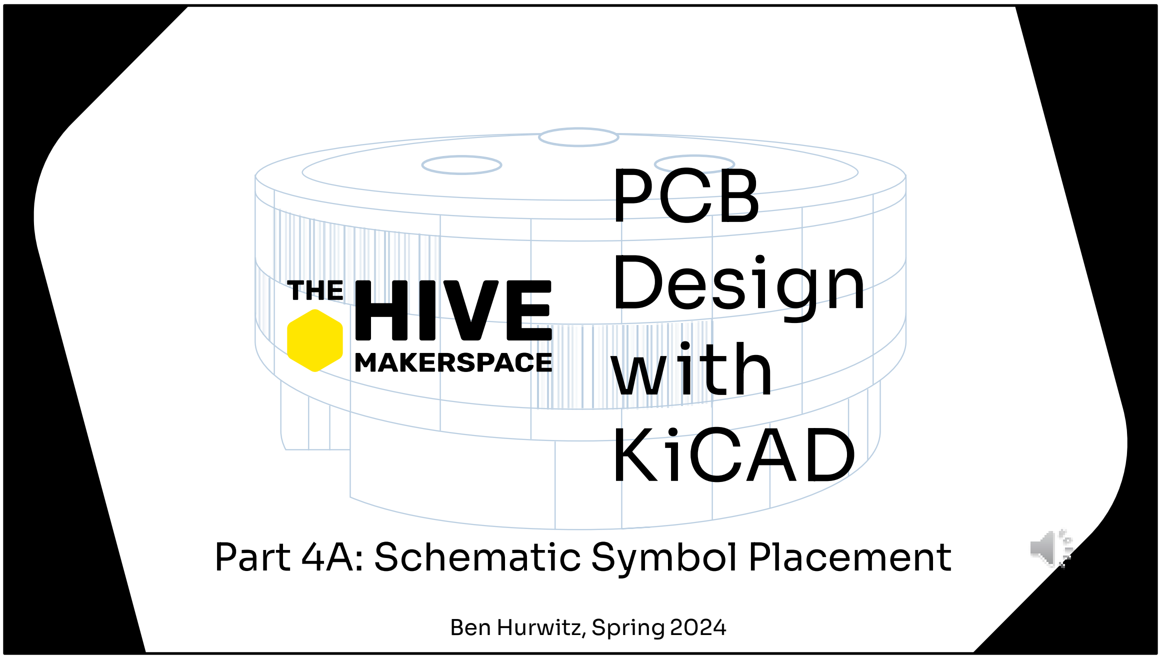
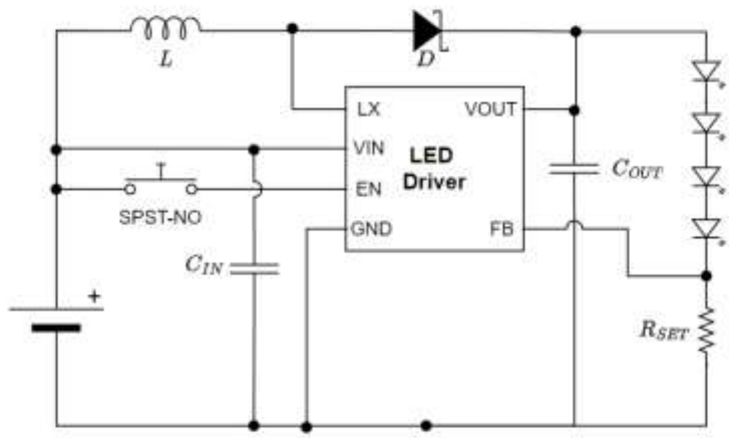


Hi, and welcome to part 4a of The Hive's PCB Design With KiCAD series. My name is Ben, and I'll be your guide here. Part 4 as a whole will cover the entirety of the schematic creation. Part 4A will look at the schematic capture window, and detail how to add built-in symbols to the design.

If you watched part 3, you might remember that in our design flow, there was a project setup and library creation step. Due to the relatively simplicity of this design, and that library management was not covered during the original workshop this tutorial is based off of, we're going to leave the library creations to parts 6 and 7. I strongly advise you to watch through those if you're considering doing design more seriously.

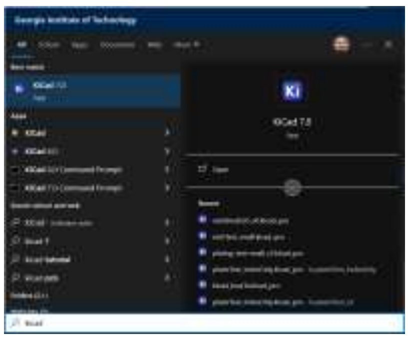
Anyway, let's get into KiCAD.

Circuit Reminder



Before we get into KiCAD, just a reminder of the flashlight circuit we're developing. Note that this image was not take from KiCAD, and therefore the symbols and graphics are different from those you are about to see.

Open KiCAD



As an aside, if you're using Mac or Linux, some of these screens will look different since I did this all on Windows 10, but the concepts are the same.

Also, this is for KiCAD version 7. I'm sure a lot of the concepts will translate to future versions, but it may look different.

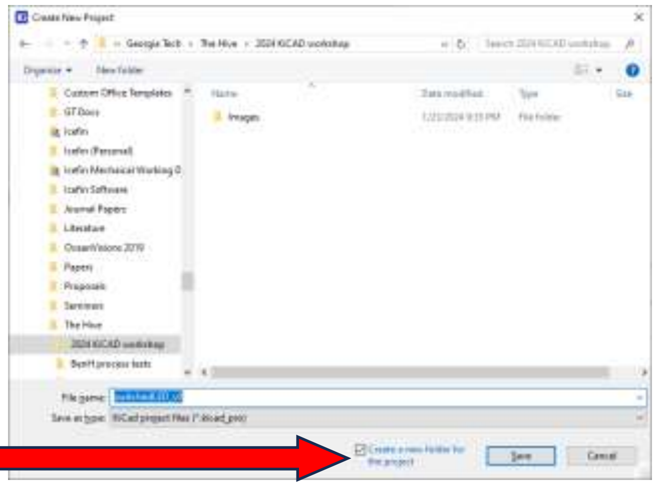
Open KiCAD however you normally open software.

As an aside, I did these slides in Windows 10 using KiCAD 7. If you're using a different operating system, or a different version of KiCAD, some of the graphics and visuals may look different or have slightly different wording from these slides, but the concepts should still hold.

Main Window

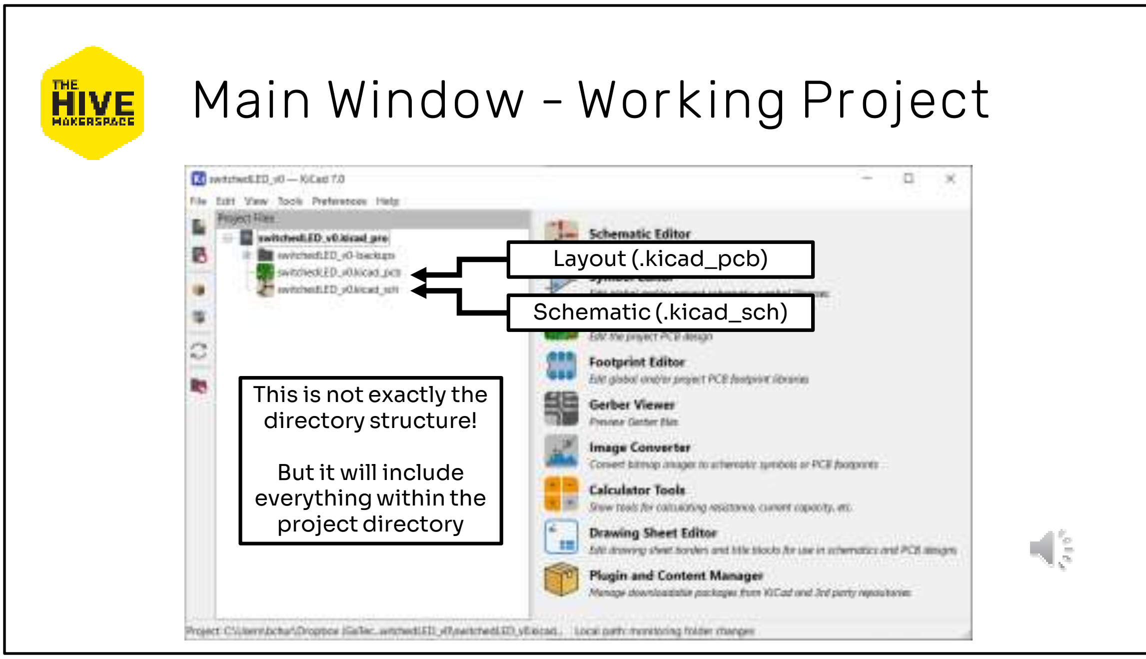
This is the first window that opens, called the Project Manager. From here, you, well, manage your project. It usually opens to some old project you've done, so we'll go ahead and create a new project under File > New Project.

File > New Project (Ctrl+N)



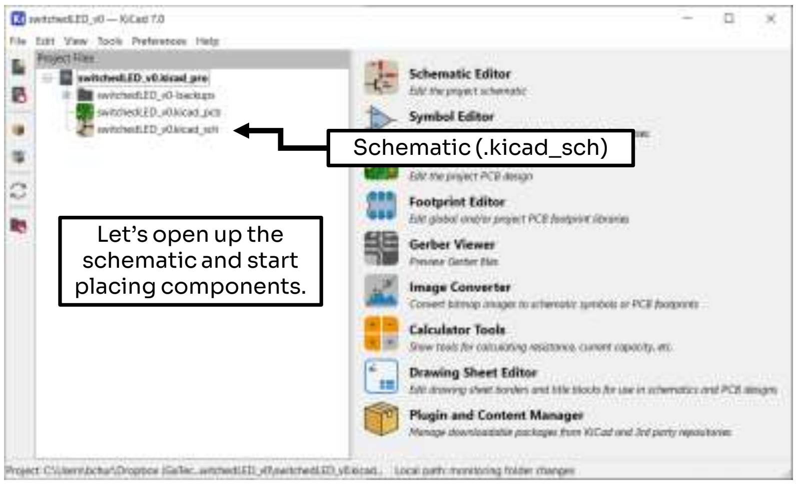
Unfortunately, this name was picked for a simpler version of this circuit.

Name is and save it wherever you'd like. I don't this name was the best choice, but whatever. I also always like to check the box at the bottom that says "create a new folder for the project". Obviously I could create the folder myself, but it's easy to do.



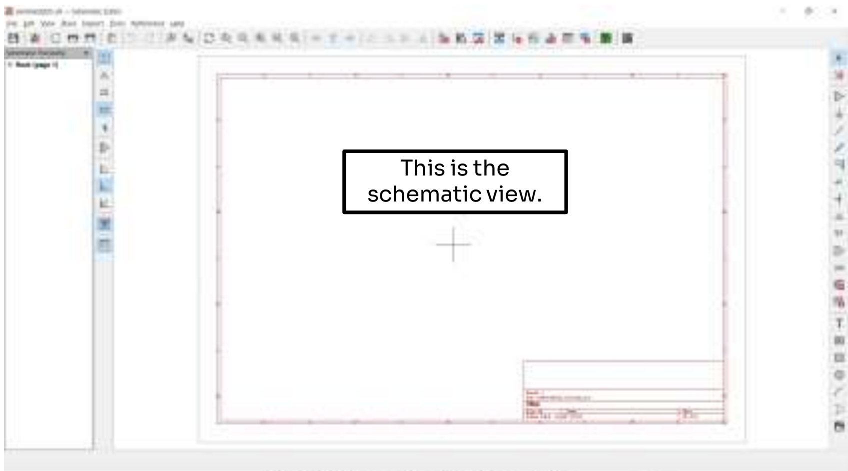
Great, so this brings our project into the Project Manager and shows us the project structure. *This isn't really a directory structure, though it looks like one. KiCAD will make backups of your project as pre-defined intervals, which can be set within the preferences, and are saved in the backups folder. On the right are shortcuts to the various tools that KiCAD offers. The two keys files are the *schematic, with the extension .kicad_sch, and the *layout, with the extension .kicad_pcb.

Main Window - Working Project



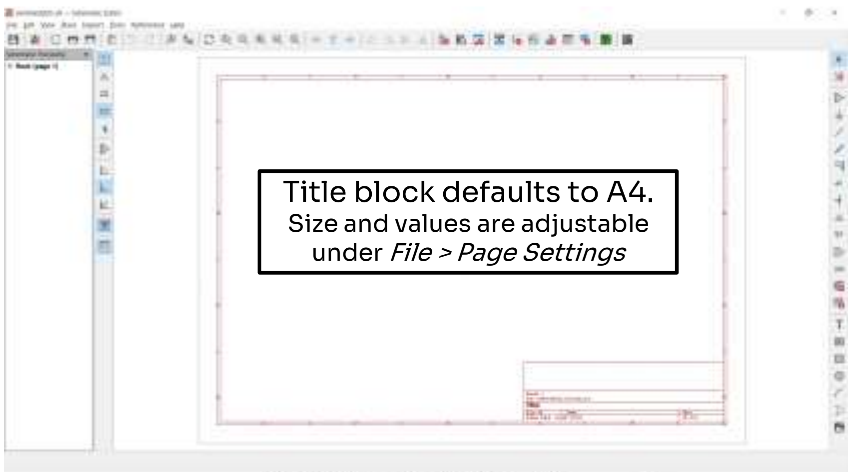
Let's open the schematic file by double-clicking to open the Schematic Editor.

Schematic View



This is the blank schematic view.

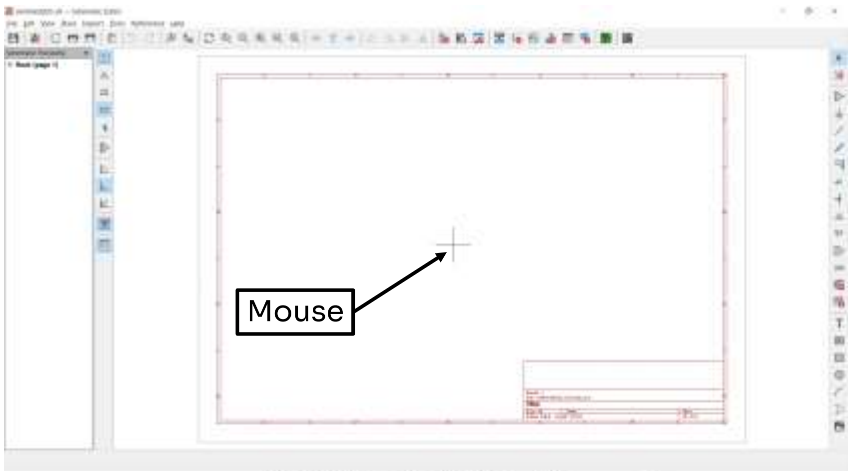
Schematic View



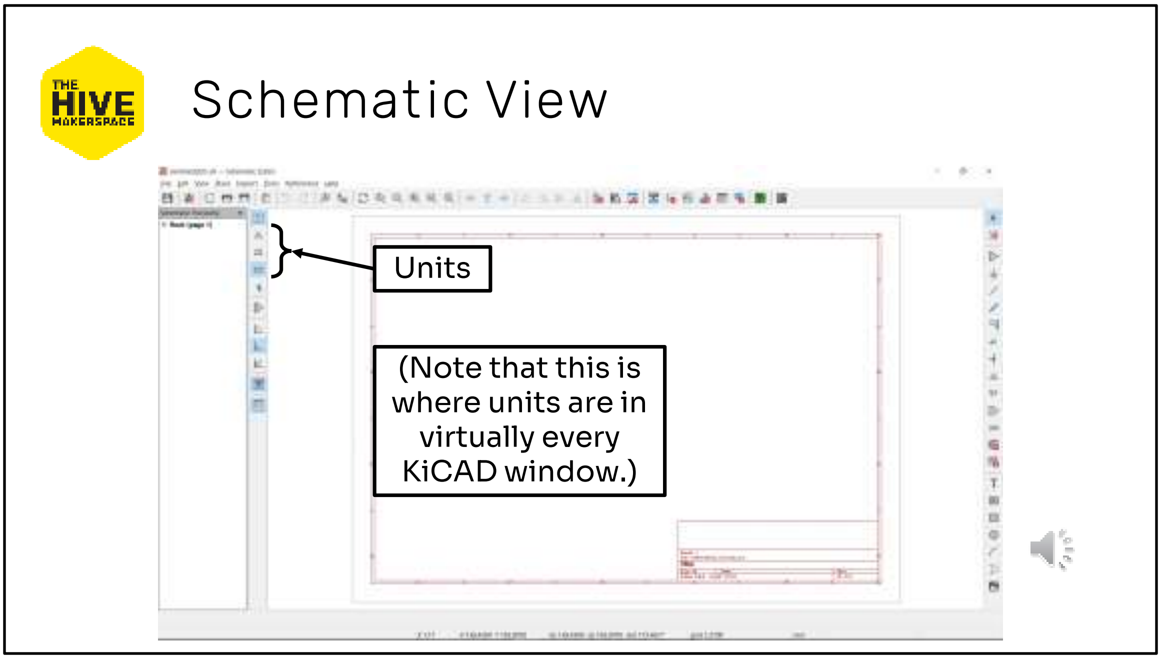
The title block defaults to A4. You can change that and adjust the text within it under the page settings. You can also safely ignore it and place whatever you'd like outside of it; it's not a limit at all. There is a way to remove it, but it requires making a custom drawing sheet. Not hard, just beyond the scope of this video. A link is found in the PDF:

https://forum.kicad.info/t/hide-title-bar-from-existing-drawings/46213

Schematic View

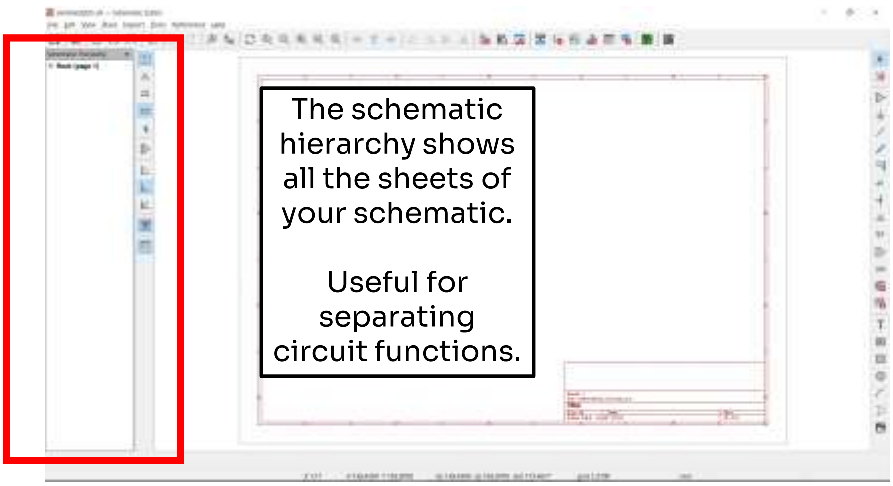


The mouse is the crosshair icon.



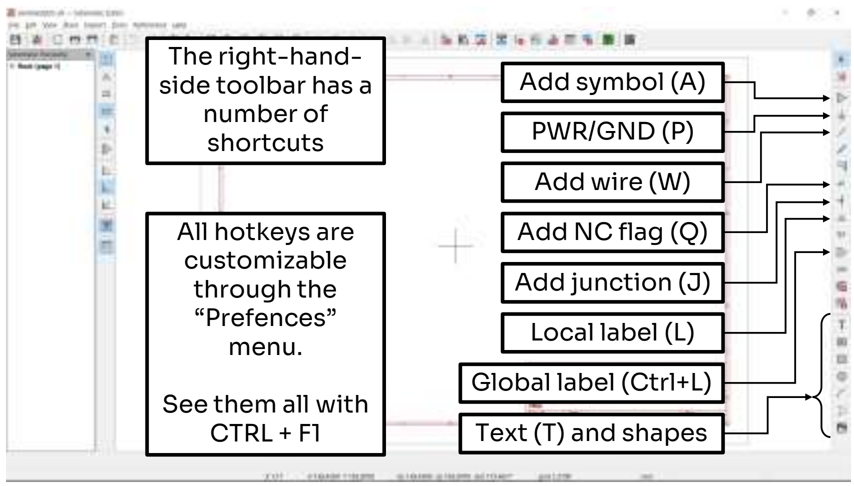
The units and grid settings are located in the upper-left. This is pretty much where these settings will be in every window in KiCAD.

Schematic View



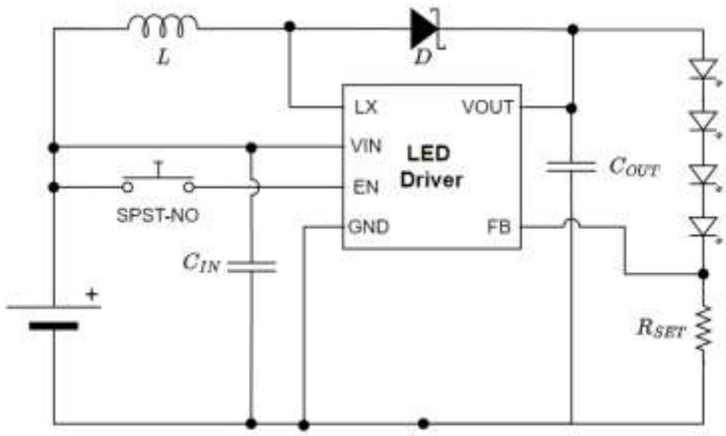
The left-hand side of the editor is the schematic hierarchy. For larger or more complex designs, it can be useful to separate the schematic into multiple sheets. Each sheet would then show up here.

Schematic View



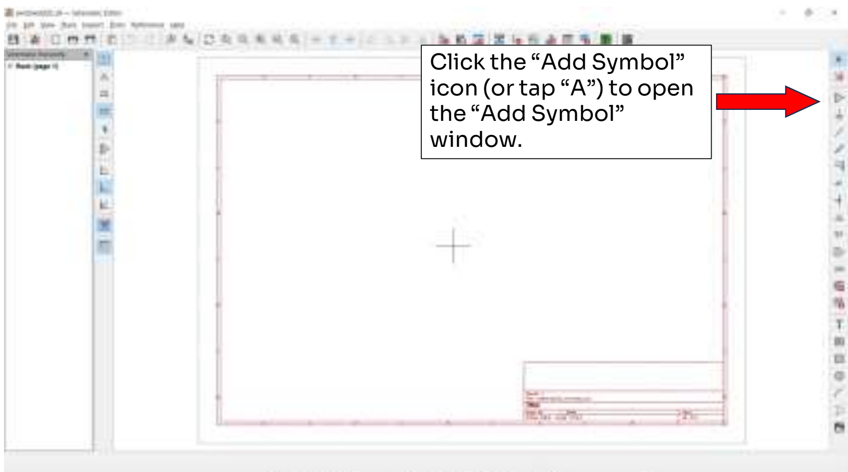
The right-hand side of the window has a toolbar that I’ll call the action toolbar, with shortcuts to many different useful actions. These include *adding symbols, *adding power symbols, *adding wires, *no connection flags, *junctions, *lables, *global labels, *text, and shapes. All these actions have hotkeys assigned to them, which can be adjusted in the preferences menu. Hit CTRL + F1 to get a list of all the currently available shortcuts.

Circuit Reminder

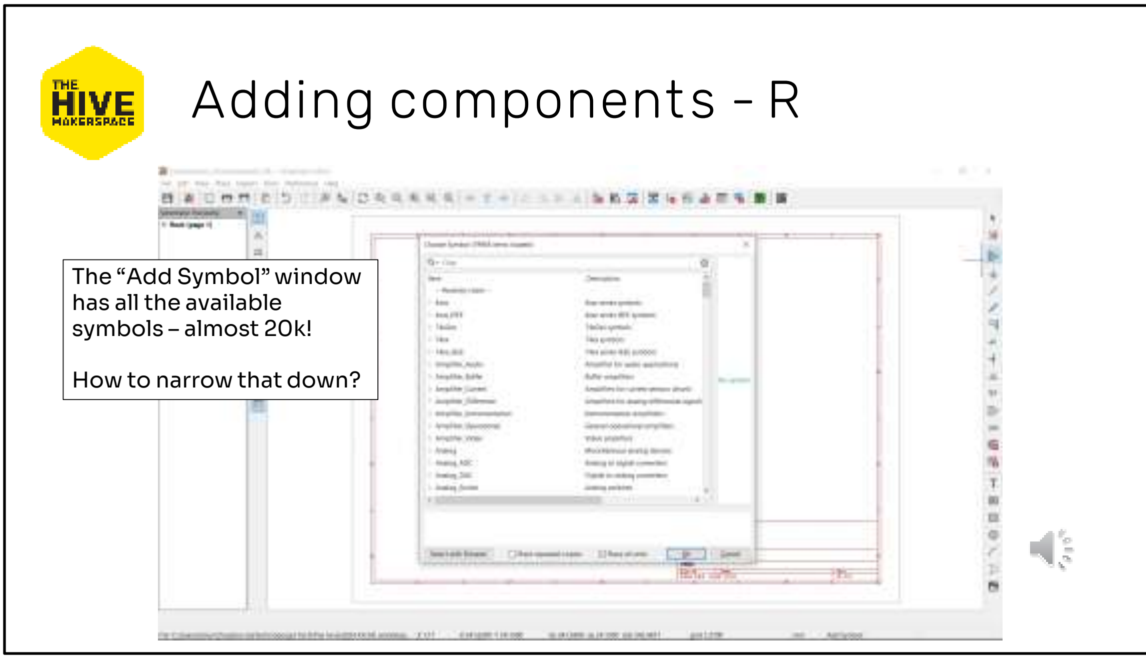


We’re going to start by adding all the symbols now. We’ll start with the generic symbols, like the resistor, LEDs, battery, and so on, and then do the switch. The IC will come last.

Adding components - R



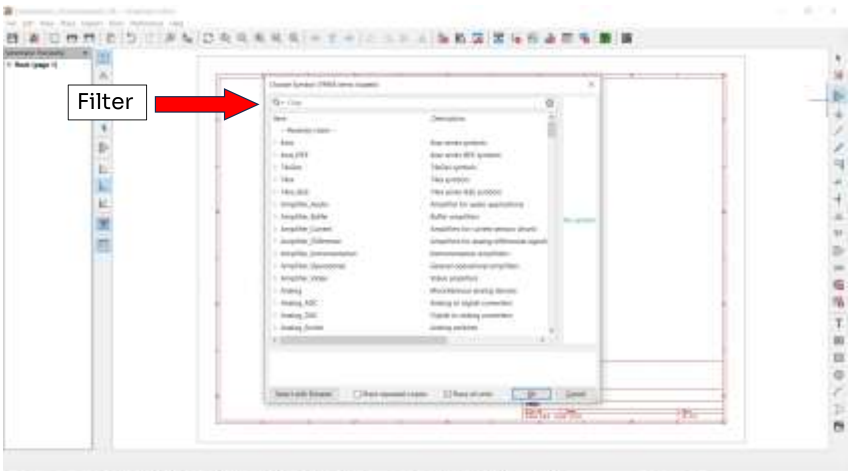
To add a symbol to the schematic, tap the "A" key, or click the icon identified in the actions toolbar.



This will open up the “Add Symbols” window.

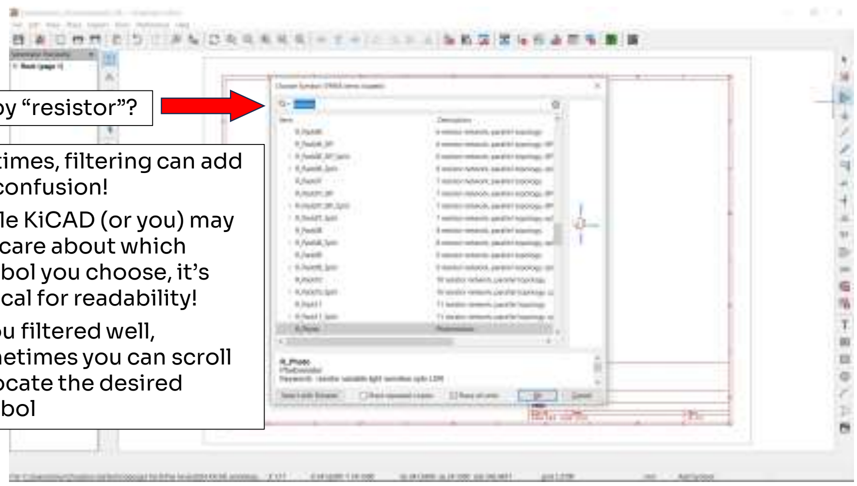
There are nearly 20 thousand symbols built-in to KiCAD!

Adding components - R



We can use the filter at the top of the window to narrow this down…

Adding components - R

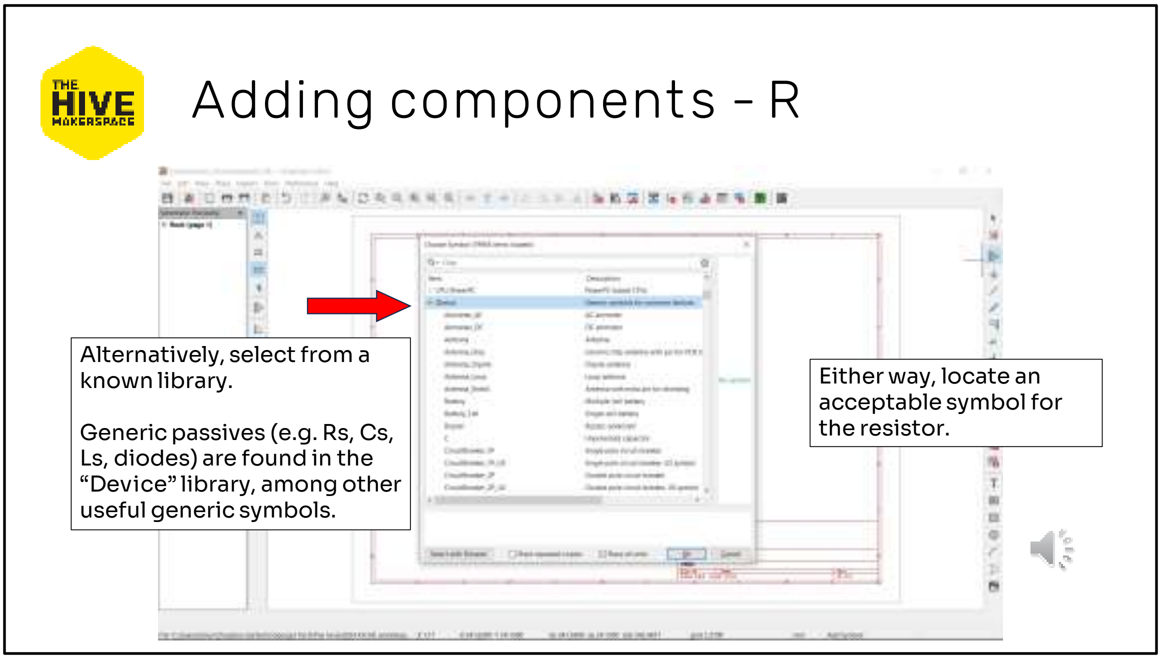


Filter by “resistor”?

Sometimes, filtering can add more confusion!

- While KiCAD (or you) may not care about which symbol you choose, it’s critical for readability!
- If you filtered well, sometimes you can scroll to locate the desired symbol

… although that may not be as helpful as we’d like, and can potentially be even more confusing. Scrolling through the filtered options can help, but only if there aren’t a thousand of them.

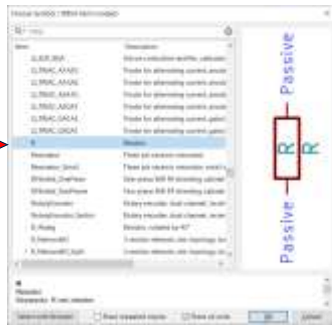


Alternatively, if you know the library, you can just open that directly.

Many, even most (though not all, as we'll see), generic components can be found in the Device library. You'll still have to scroll, but by filtering and locating that library, it can help to narrow it down when you don't know exactly what you're looking for.

Go ahead and pause the video and locate the resistor symbol.

Adding components - R



If you locate the symbol “R”, you’ve found the IEC (internation) version

If you locate the symbol “R_US”, you’ve found the ANSI (US) version

Either is fine!

Click “OK” to select it.

There are two resistor symbols you may have found.

If you just found the one named “R”, you might have noticed that it didn’t look right. That’s because the box-type symbol is the international version of the resistor symbol. In the US, this would be seen as an unknown impedance.

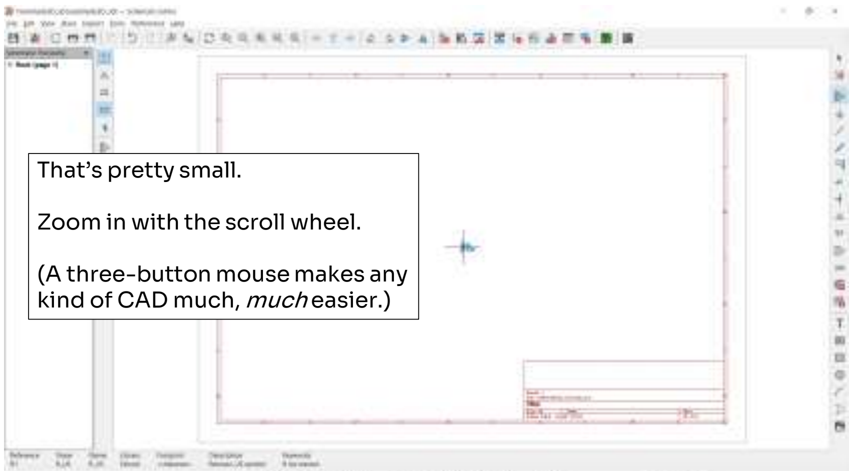
If you looked a bit harder, you might have located the US version of the symbol named “R_US”.

Either is totally fine, especially for this tutorial.

Remember that the schematic is for you, your colleagues, your boss, and your future selves to read, so use whichever symbols you’ll best be able to communicate your intentions to your boss with.

Click OK to select it.

Adding components - R



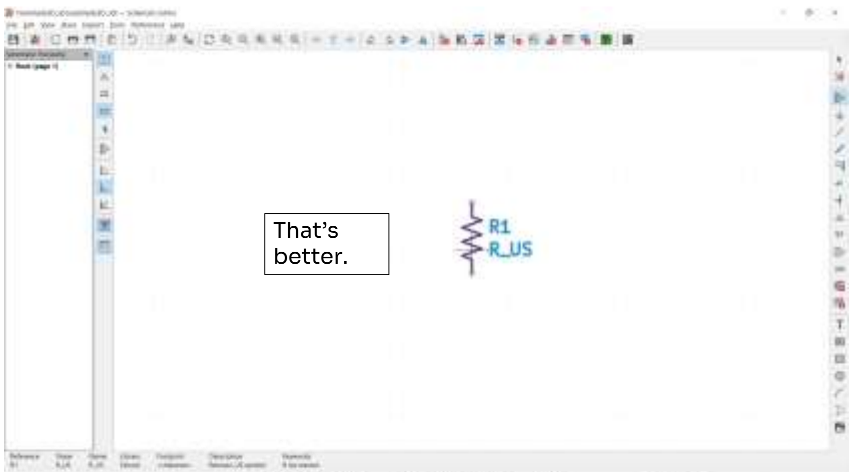
That’s pretty small.

Zoom in with the scroll wheel.

(A three-button mouse makes any kind of CAD much, *much* easier.)

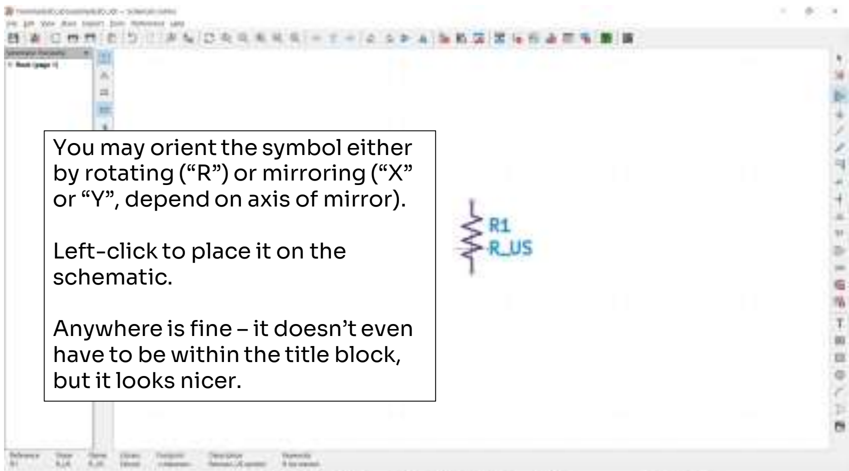
The “Add Symbol” window will return you to whatever zoom depth and viewing location you were at when you opened it. Let’s zoom in here with the scroll-wheel, or whatever trackpad combination you use for zooming. Note that a three-button mouse is really nice to have for all CAD software, KiCAD included.

Adding components - R



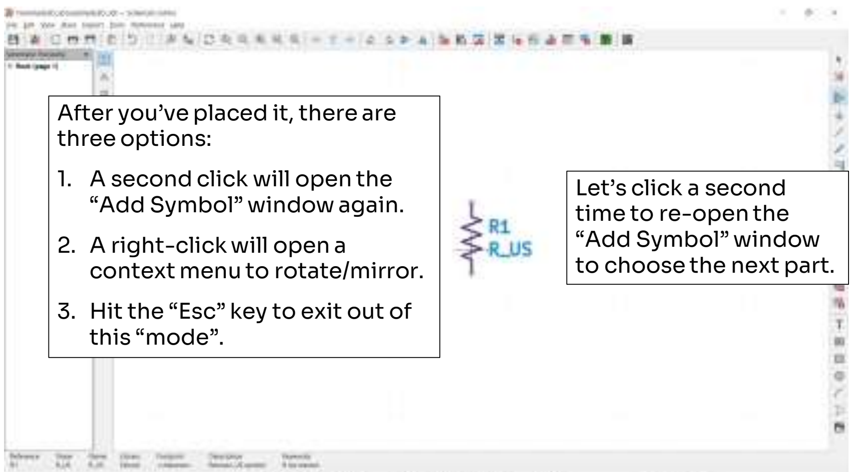
That's better.

Adding components - R



You can orient the symbol prior to placement by rotating with R or mirroring with X or Y. Place it with a left-click. Outside the title block is find if you’d like.

Adding components



There are three options after you've placed it.

*Left-click to open the add symbol window again.
*Right-click to open a context menu for things like rotating, properties, or duplicating.
*Hit "ESC" to exit out of this "add symbol mode".

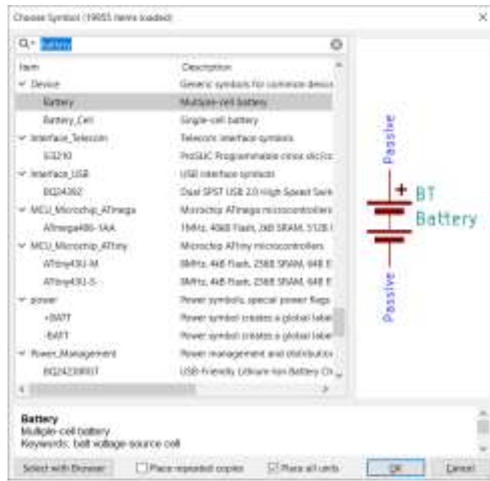
*Let's just click again to add the next symbol, the battery

Adding components - Battery

Filtering by “battery”, we can scroll to locate two generic devices in the “Device” library.

Select the single-cell option, and add it to the schematic near the resistor.

(Remember: symbols are just for people to read, so use whichever symbol will make that easiest. If it’s important that it’s multi-cell, use that.)

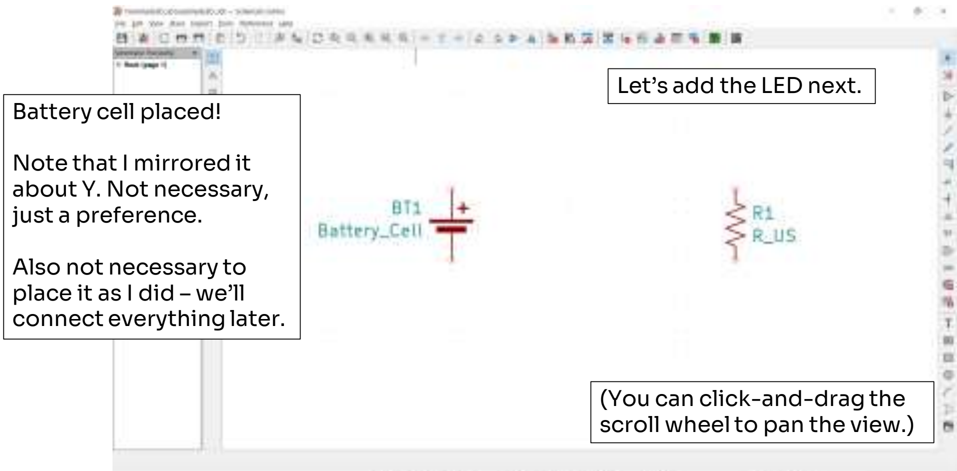


Like the resistor, the battery is a standard symbol, so there’s a reasonable expectation that we can find a symbol for it in the Device library,

If we filter by battery, we’ll see there are actually two – “battery” for a multi-cell battery, and “battery_cell” for single-cell battery.

Remember, since the schematic is for you and your boss to read (and your future selves), use whichever is most informative. If it’s important that it’s a multi-cell battery, use that one. Otherwise, use the single-cell symbol.

Adding components - Battery



Place the battery symbol on the schematic. It's not critical where right now since we'll move most of this around later. I mirrored by symbol because I preferred the text to the left.

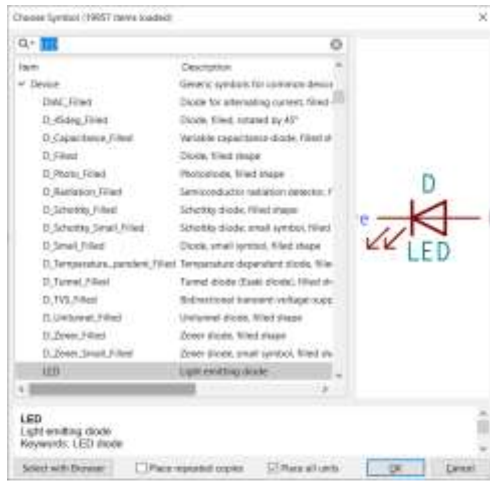
Left-click again to add an LED.

Adding components - LEDs

How convenient that there's a generic LED symbol as well!

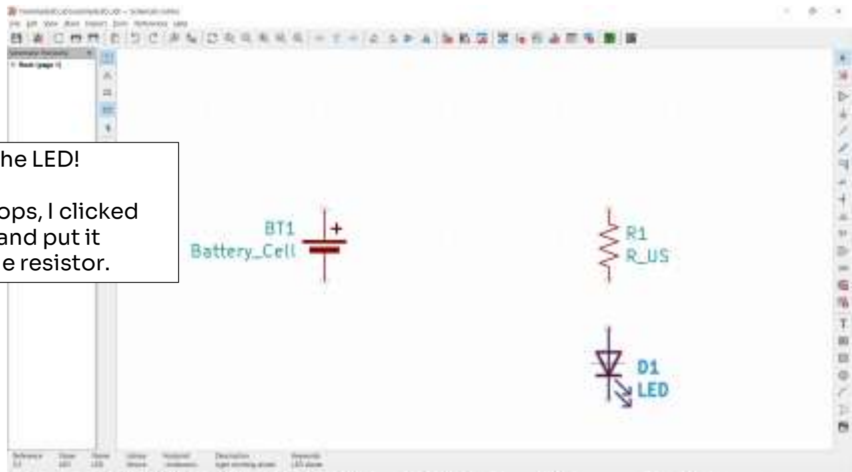
Let's add that to the schematic.

(Most non-IC components have a generic symbol that's acceptable to use, but be aware of the number of pins – that should match your actual device.)



Looking in the Device library again gives us an LED symbol. Add one to the schematic.

Adding components - LEDs

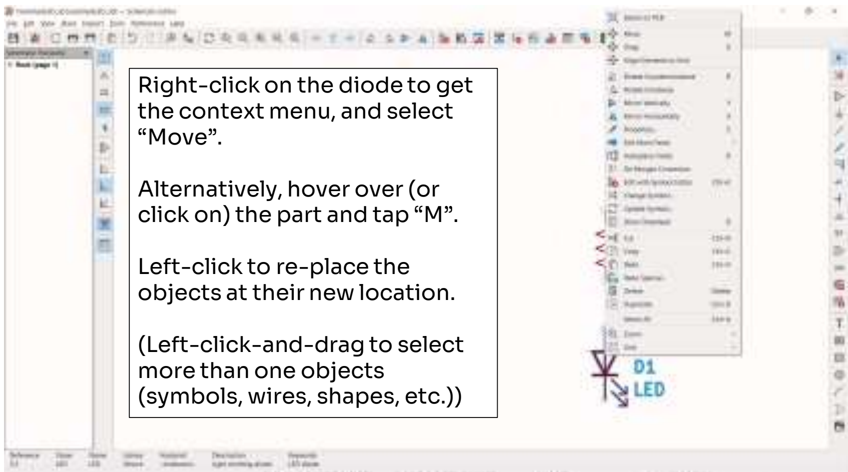


There's the LED!

But whoops, I clicked too fast and put it below the resistor.

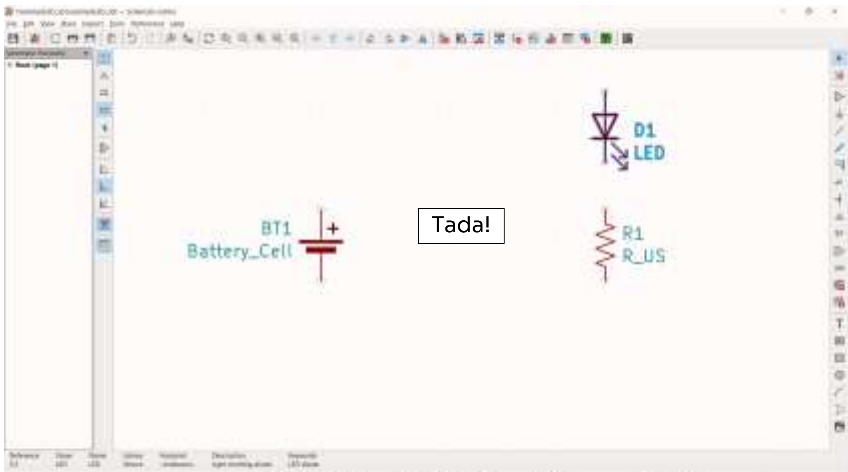
Got too excited and place it below the resistor. Whoops.

Adding components - LEDs



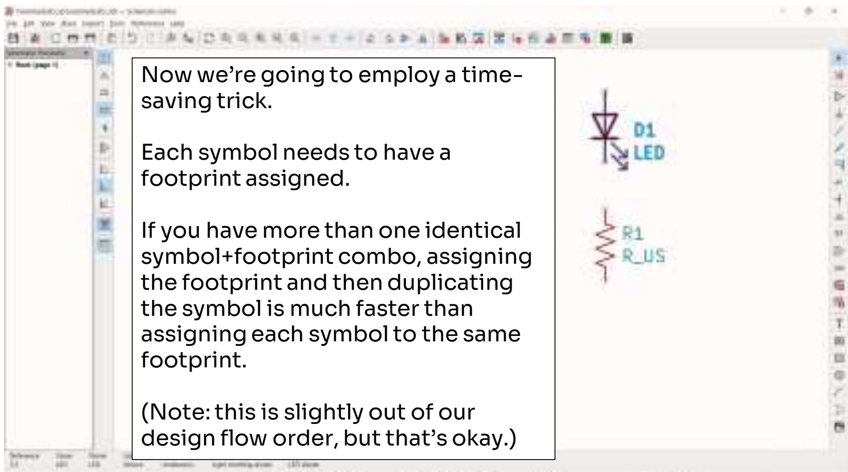
Thankfully it's easy to move. You can left-click-and-drag to move it, click or hover and hit "M", or find move in the right-click context menu. Left-click again to place it at the new location.

Adding components - LEDs



Great.

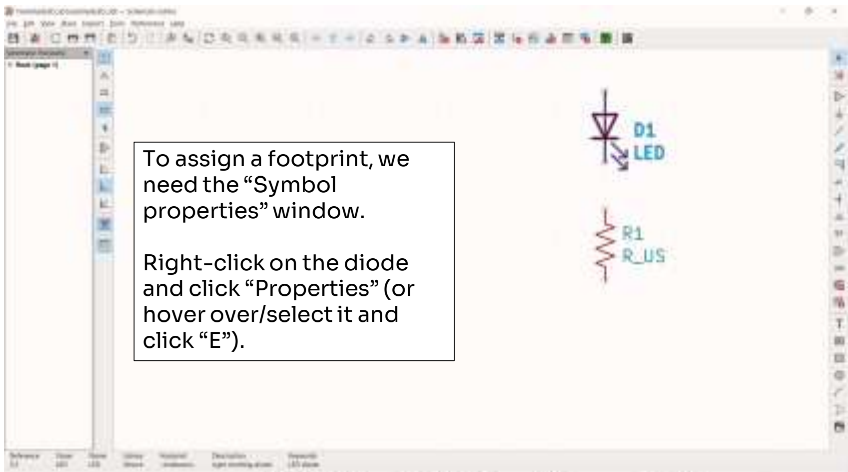
Adding components - LEDs



Now I'm going to use the duplicate feature to make the other LEDs.

But wait! If we add a footprint to the LED first (and a value if we wanted to), we wouldn't have to add footprints (or values) to the other LED symbols later. Adding a footprint can be a touch of a lengthier process (takes like five clicks and two windows), so this can be quite time-saving if you have many duplicate components.

Adding components - LEDs



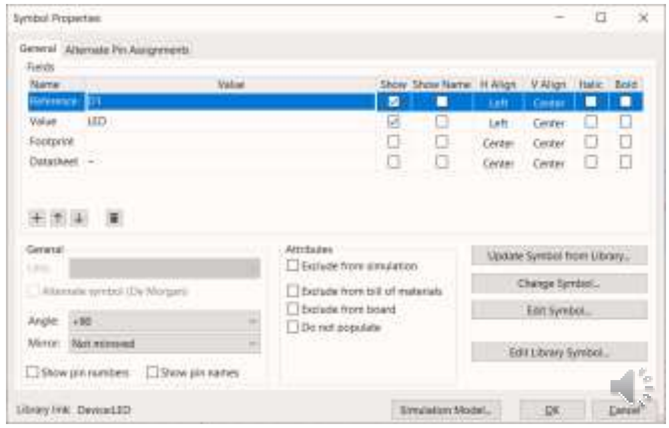
To assing a footprint (and a value), we’ll open the symbol’s properties window.

Select/hover over the symbol and press “E”, or select “properties” from the context menu (right-click).

Adding components - LEDs

There's a lot going on in this window.

Most of this we don't care about right now.



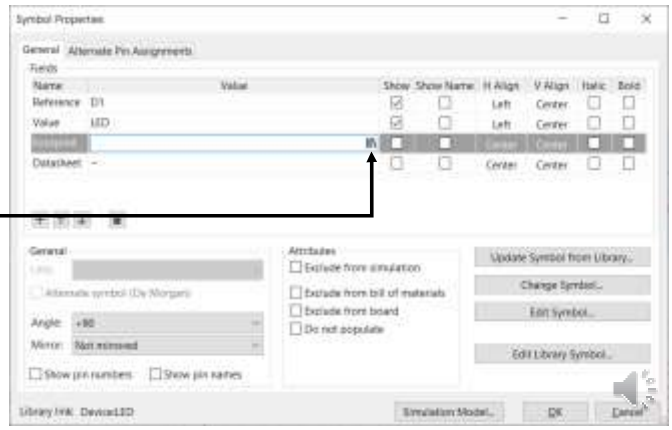
This window has a lot of options.

You can adjust the symbol's "value" here by adjusting the "value" field's value (this is confusing, but it's where it currently says "LED"). For an LED, you might put "green" or "red" in here; for a resistor, maybe 1k; for a capacitor, maybe 1u. Units are never included in the symbol value; this is a historical artifact, I think, coupled with the fact that you can't add symbols like omega into this field.

Most of this window isn't relevant right now.

Adding components - LEDs

Clicking into the “value” box for the “Footprint” field, a little icon pops up on the right (sort of looks like three books). Click it to open the footprint browser.



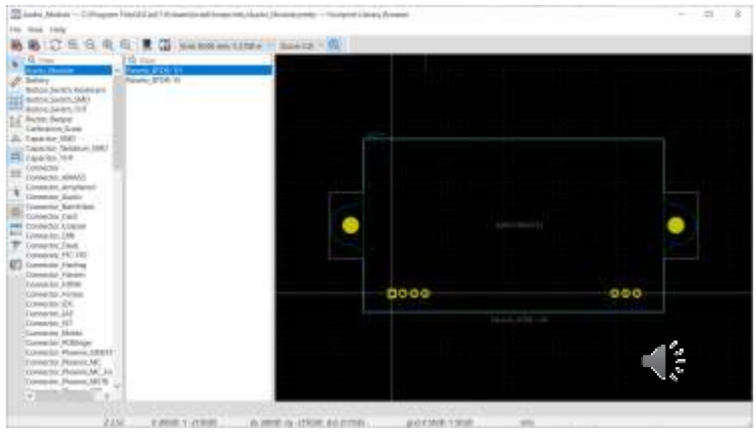
To assign or change a footprint, click the footprint box and then the little three book icon on the right.

Adding components - LEDs

This window shows all the footprints in all the libraries that are available to this project.

(Recall: footprints are the physical layout of the device.)

Filter by LED to see what there is.



This will open up the Footprint browser, which shows all the available footprints.

Filter on the top left by “LED” to see what’s available.

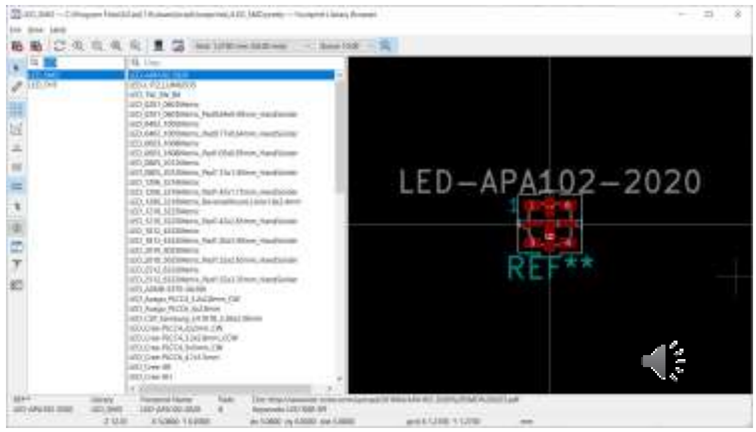
Adding components - LEDs

Wow, a lot of LEDs.

Picking the correct footprint is critical! Otherwise the part may not fit.

(Note that there may be multiple footprints that can work for a part.)

We need the 5mm through-hole LED… let's find it in the "LED_THT" library.



A lot of footprints here still.

It's critical to make sure the footprint selected will actually fit the part you're using, though there may be multiple options that will work.

Since we're using a through-hole LED, let's select the "LED_THT" library.

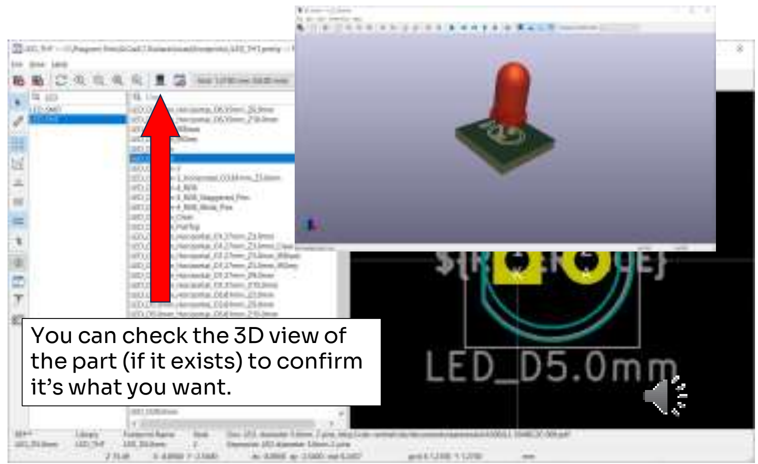
Adding components - LEDs

Lots of options here.

The “D” indicated the diameter, with additional information provided after that.

I don’t care about any special or specific additional info, so just the boring 5mm LED it is.

Double click to assign it.



You can check the 3D view of the part (if it exists) to confirm it’s what you want.

There are still a lot of options here, but here’s the trick: the “D” in the footprint name generally means “diameter”. Since we know we’re looking for a 5mm diameter 2-lead LED, that narrows down our search.

Some of these have additional information in the name, such as the -3 for three pins, or “clear” for a clear dome. We could also select a horizontally-oriented model here if we knew it needs to be horizontal.

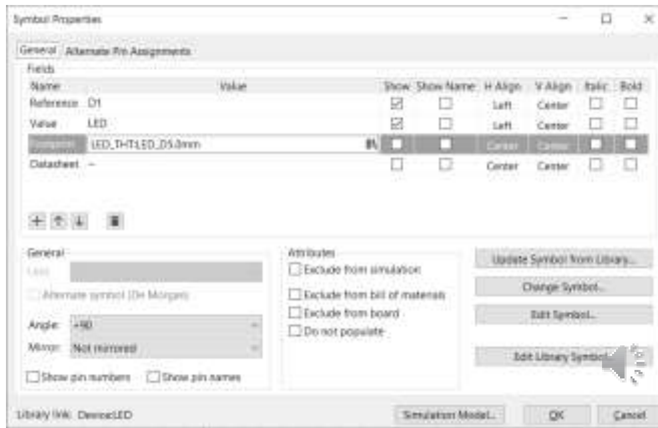
*You can also check the 3D model of the part by clicking this icon here.

Since we’re just using a standard 5mm LED, I’ll just double-click that one to assign it.

Adding components - LEDs

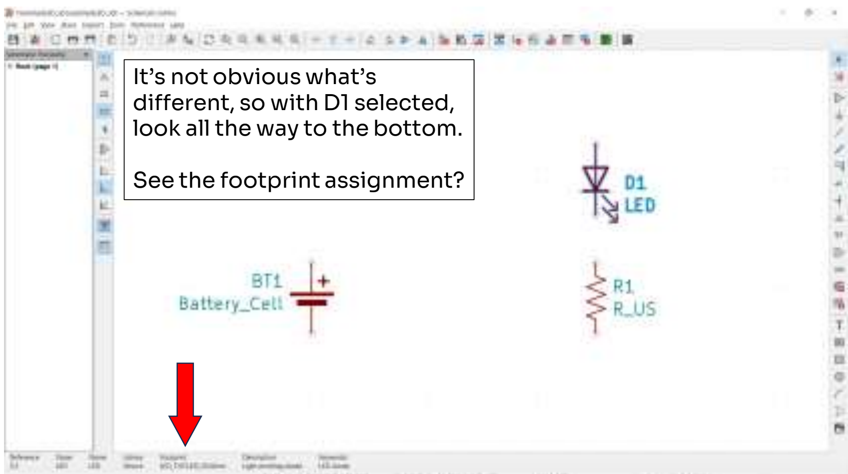
The footprint has been assigned!

Click OK.



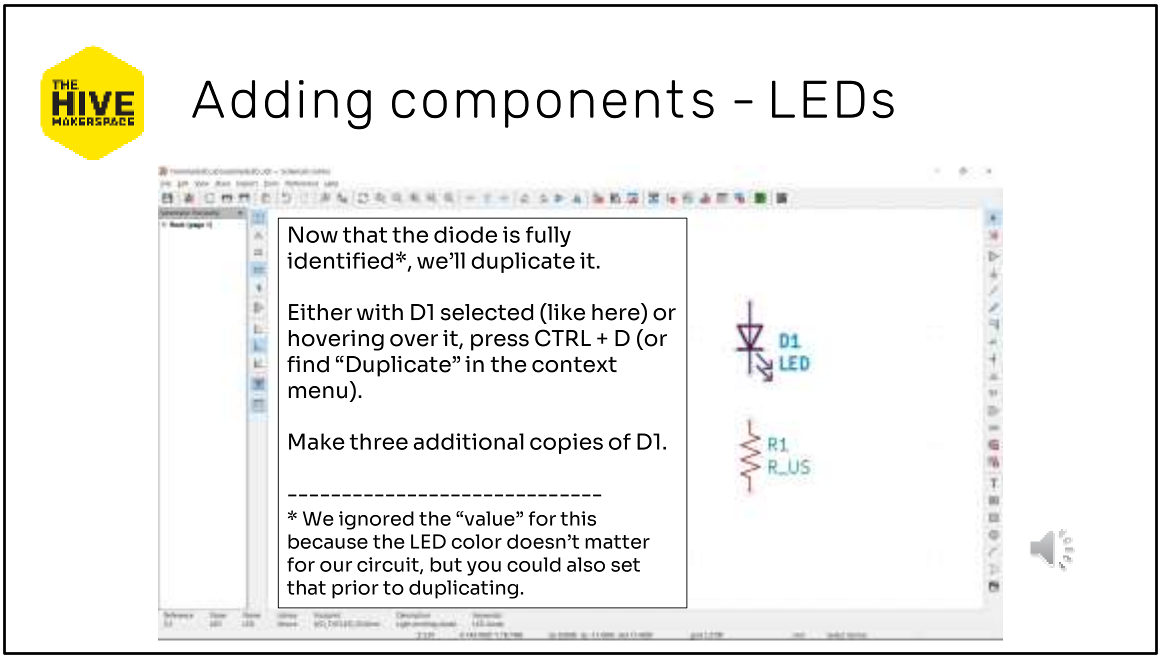
Great, the footprint is in the “footprint” value box. Click OK.

Adding components - LEDs



You can check the bottom to see that the footprint has been assigned. See it?

*It's not super obvious.

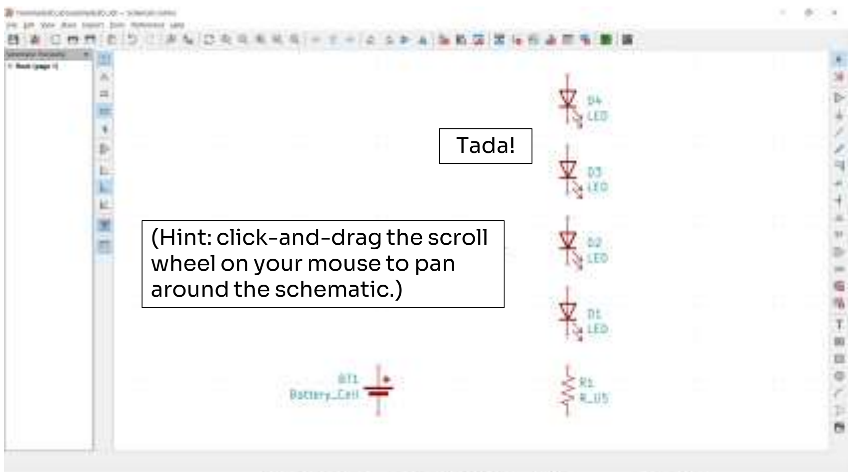


Now that we've assigned a footprint (and a value), we can duplicate it.

Select the part and press CTRL+D to duplicate, or find duplicate in the context menu. Make three copies.

Note that copy and paste will also work, and I'm not sure what the difference is between those methods.

Adding components - LEDs

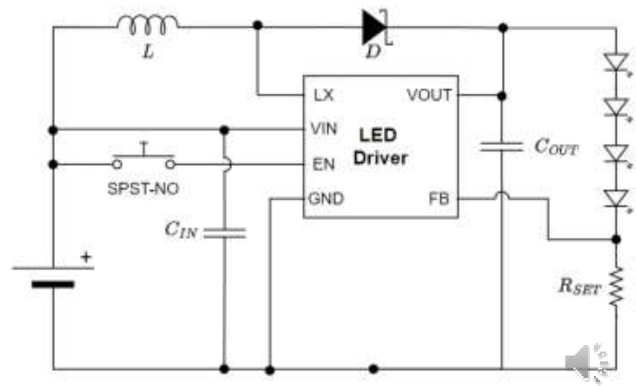


Might look like this.

Note that your schematic will almost certainly look different than mine and that's okay! There are many different ways to put a schematic together. As long as things are connected, which we'll do later, it's minimally functional.

Adding components – L, Cs, D

- Take a few minutes to add these parts.
- The L and Cs are just like the R we did before.
- The diode might require a bit more searching…

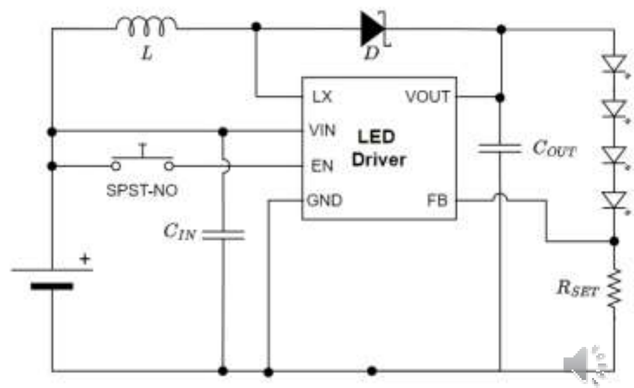


At this point, I'll suggest pausing this video and placing the inductor, capacitors, and the diode (L, C, and D, respectively).

All three can be found in the Device library, but be careful about the diode…

Adding components – L, Cs, D

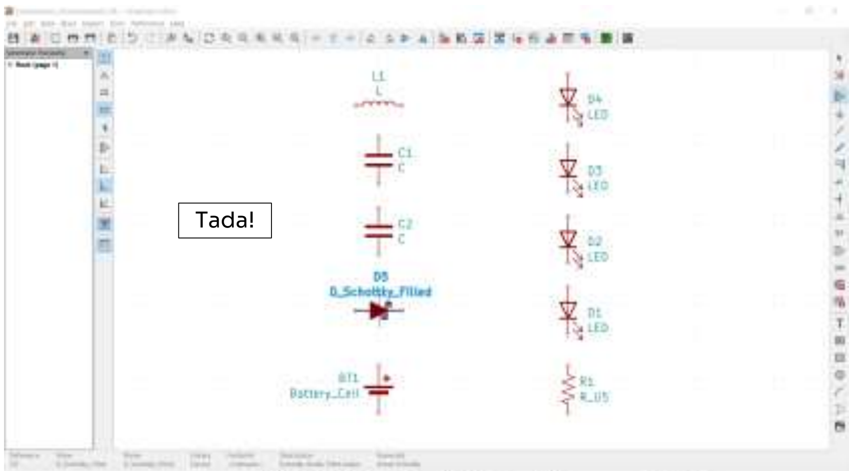
- Hint: the diode is a Schottky diode (with the weird backwards integral symbol), not a regular diode, and yes, it's a standard component.



… because the diode is a Schottky diode, meaning the symbol is slightly different than a regular diode.

Anyway, take a minute or two to find and add these four parts.

Adding components – L, Cs, D

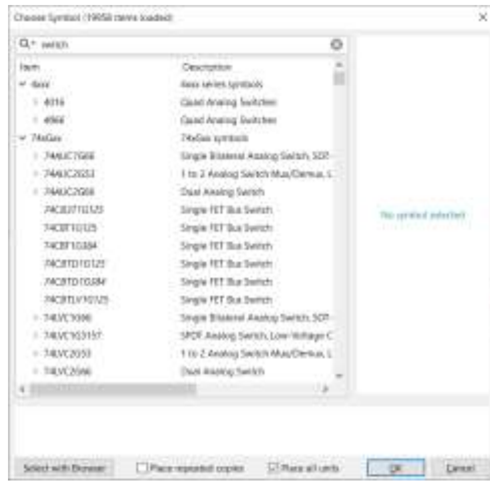


Hopefully you didn't struggle too much with that, but you should now have ten symbols.

Adding components – Switch

The last non-IC part is the switch.

Filtering by “switch” gives us a huge number of options again. Too many.

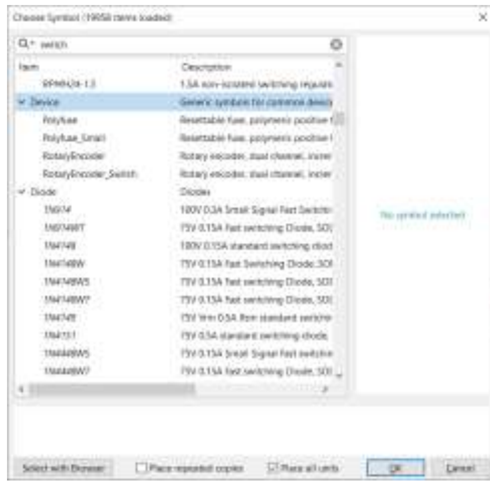


Back into the “Add Symbol” window, we’ll look for the switch next.

Lots of switches available…

Adding components – Switch

The “Device” library doesn’t have any switch symbols! Just a rotary encoder (i.e. a dial).

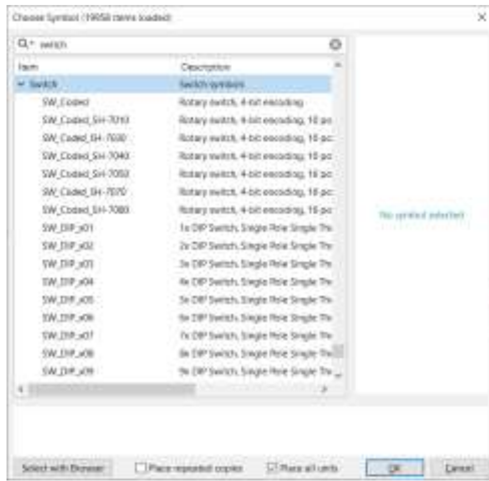


…. And none in the Device library, outside of a single rotary encoder.

Adding components – Switch

For switches, we will actually use the well-named “Switch” library instead.

But which one?



Switches are actually in their own library called, cleverly, Switches.

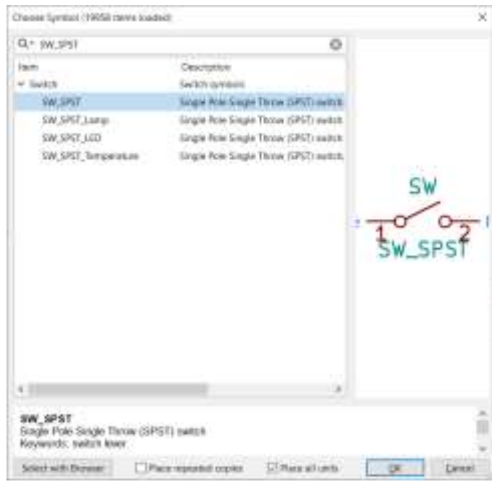
Adding components – Switch

The switch we're interested in is a SPST NO switch, the most basic switch type.

(Recall: this is a single controlled circuit with just a single output that is normally open (i.e. normally disconnected))

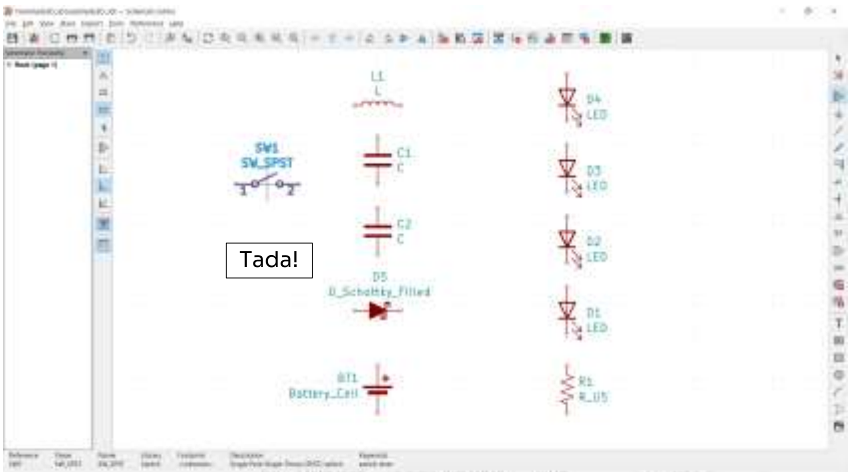
Because we know the specific type, we can actually directly filter by "SW_SPST".

Add it to the schematic.

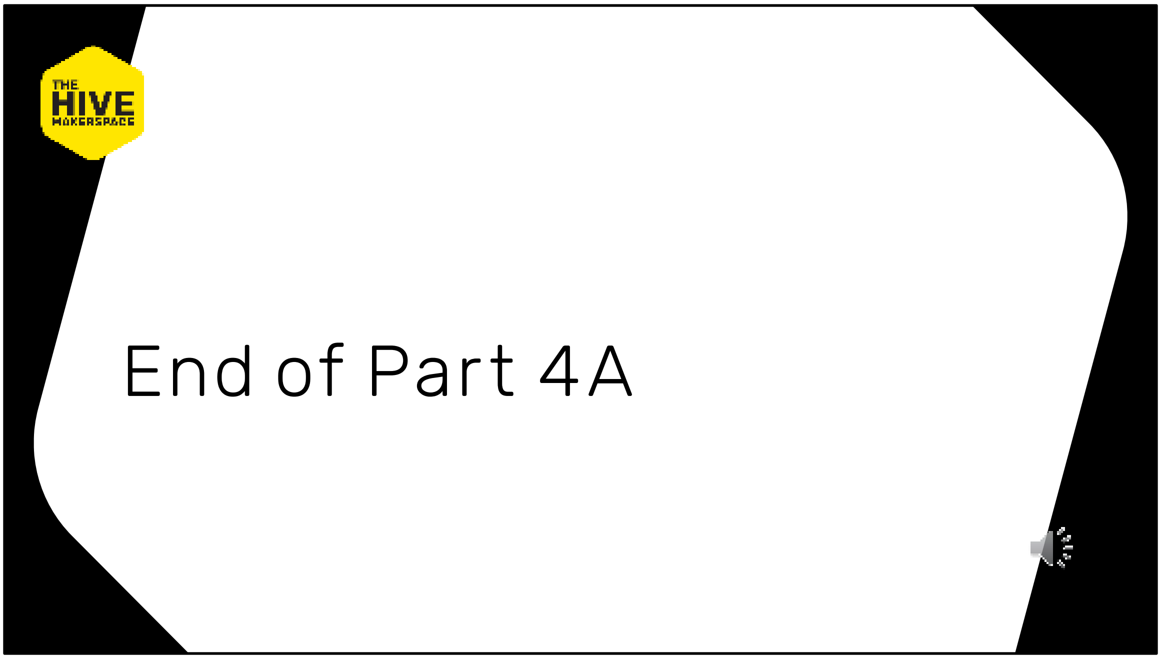


Recall the switch is a single pole single throw, normally-open, switch. Which we can filter for directly to find the right symbol. Add it to the schematic.

Adding components – Switch



Great.



And with that, we’re done with part 4A of this tutorial series. Here we took our first look at the schematic capture videw, and added the basic components to our schematic. A PDF of this video is available as well, linked in the description and hosted on The Hive’s Wiki.

Please join us for the next video, part 4B, in which we’ll cover how to handle the IC, which (spoiler alert) doesn’t have a standard symbol. See you there.