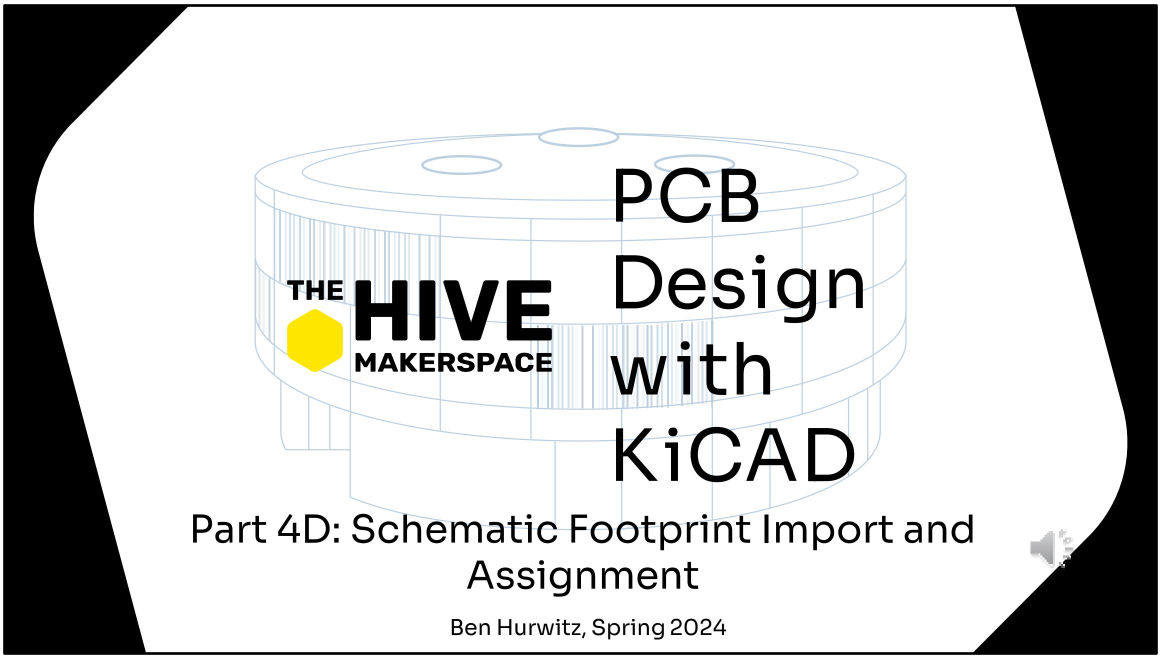
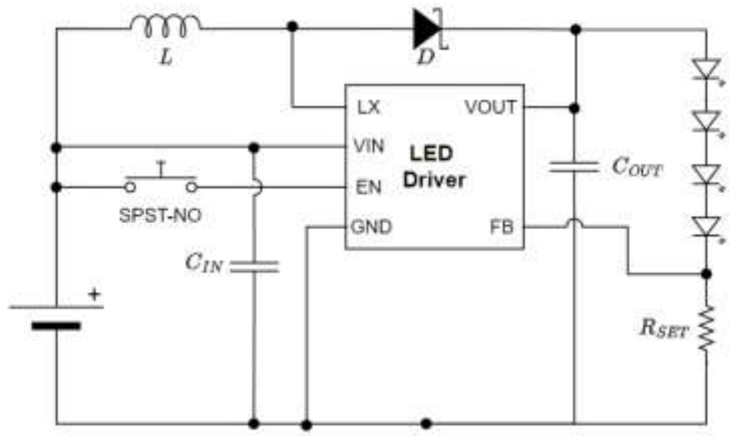


Hi, and welcome to part 4D of The Hive's PCB Design With KiCAD series. My name is Ben, and I'll be walking you through this section. Part 4 as a whole will cover the entirety of the schematic creation. In this part 4D, I'll cover assigning footprints to symbols and how to import footprints that you download from the internet. Creation of footprints won't be covered because you should almost never have to do that, and it's far more error-prone than custom symbol creation, though I will point you towards how to do it if necessary. More details about footprint creation can be found in parts 7B and 7C.

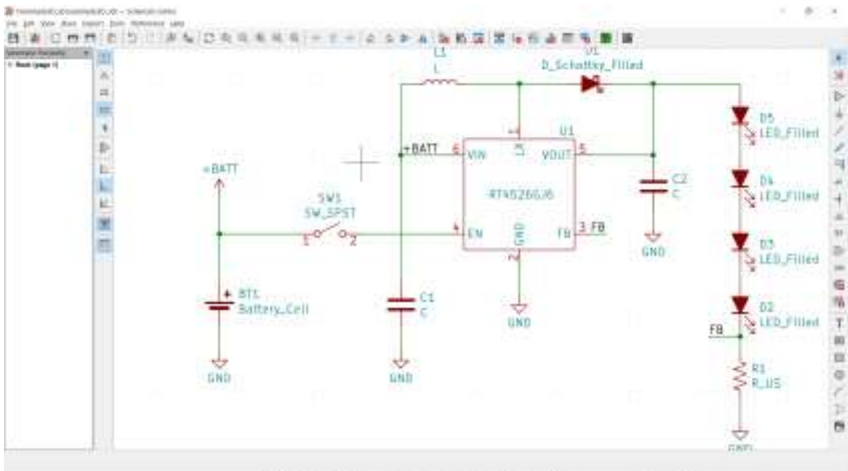
Anyway, onto footprints!

Circuit Reminder



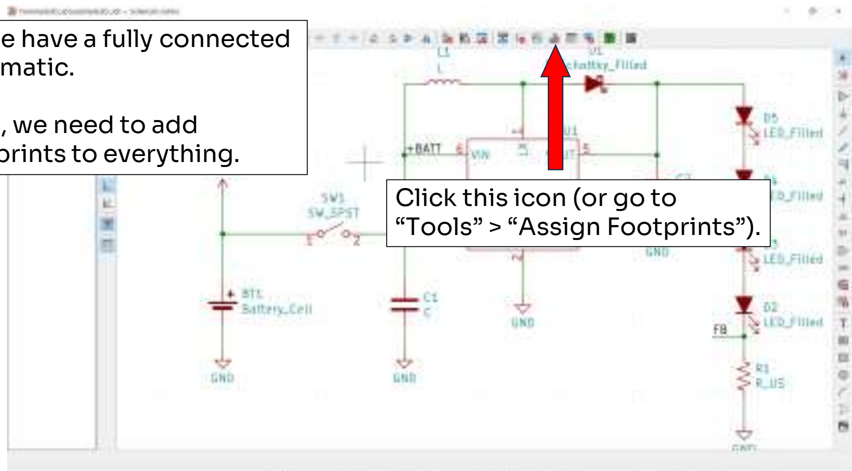
Before we get into KiCAD, just a reminder of the flashlight circuit we're developing. Note that this image was not take from KiCAD, and therefore the symbols and graphics are different from those you are about to see.

Schematic Reminder



And a reminder of the schematic that we ended part 4C with, fully populated and connected.

Assigning Footprints

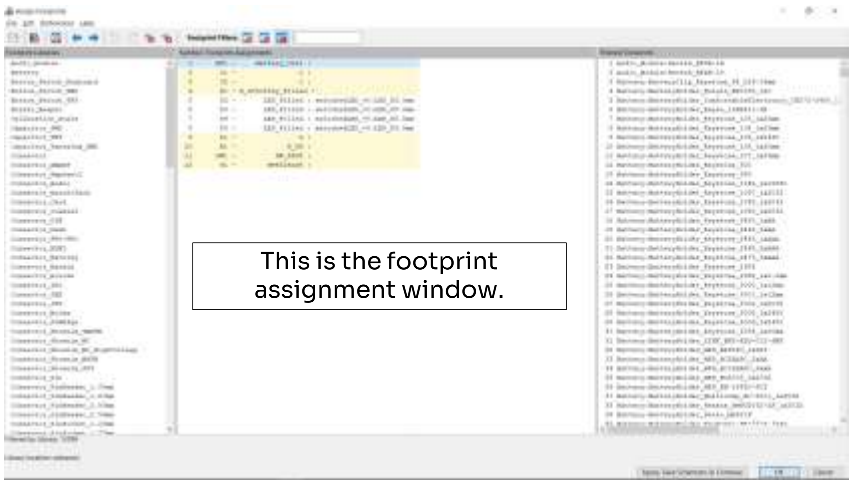


So we have a fully connected schematic.

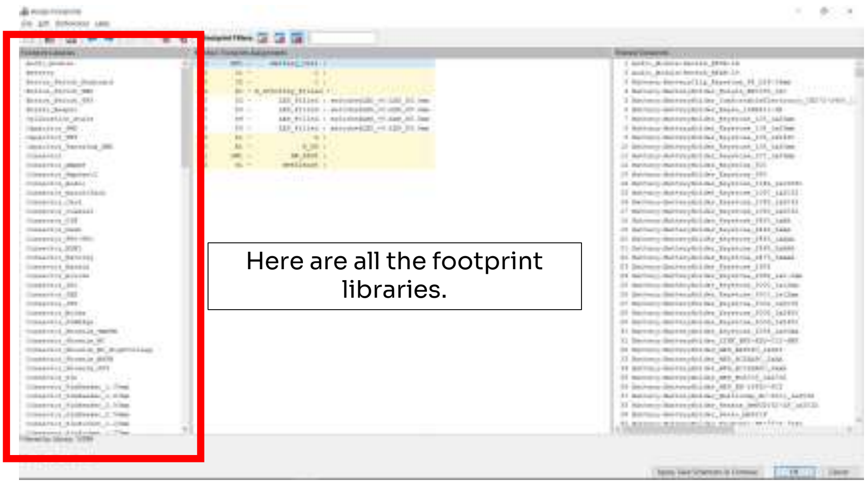
Next, we need to add footprints to everything.

Click this icon (or go to "Tools" > "Assign Footprints").

Assigning Footprints

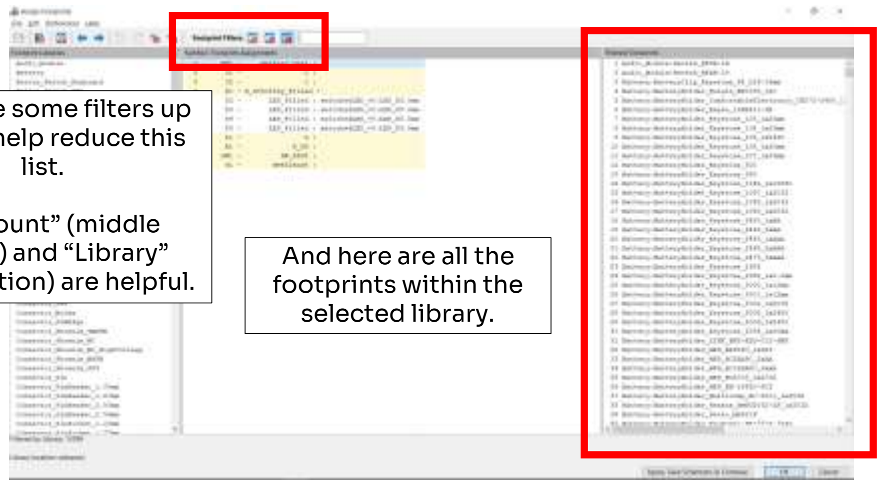


Assigning Footprints



Here are all the footprint libraries.

Assigning Footprints

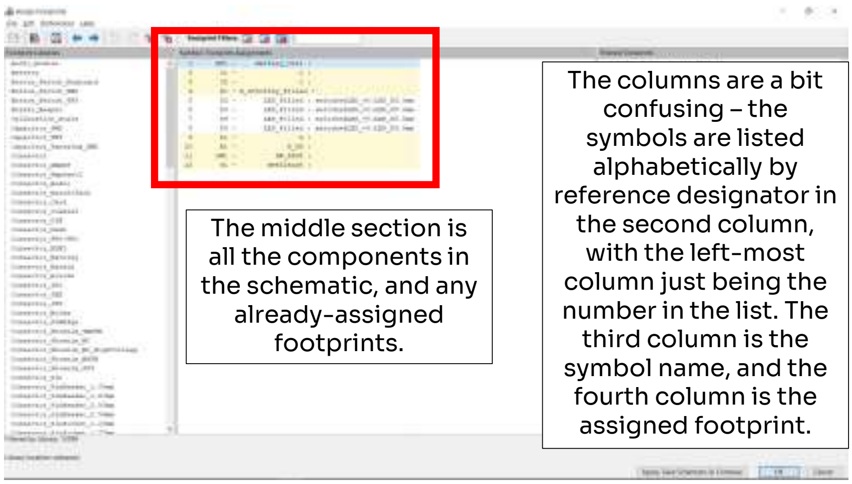


There are some filters up here to help reduce this list.

“Pin count” (middle option) and “Library” (right option) are helpful.

And here are all the footprints within the selected library.

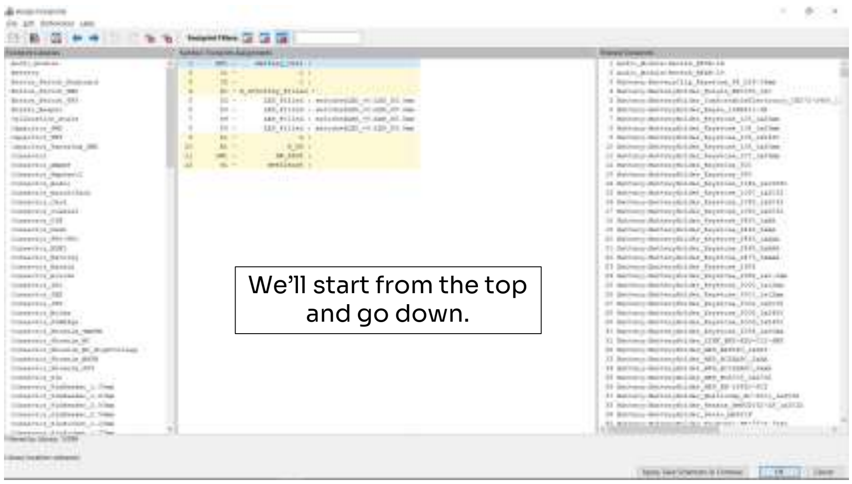
Assigning Footprints



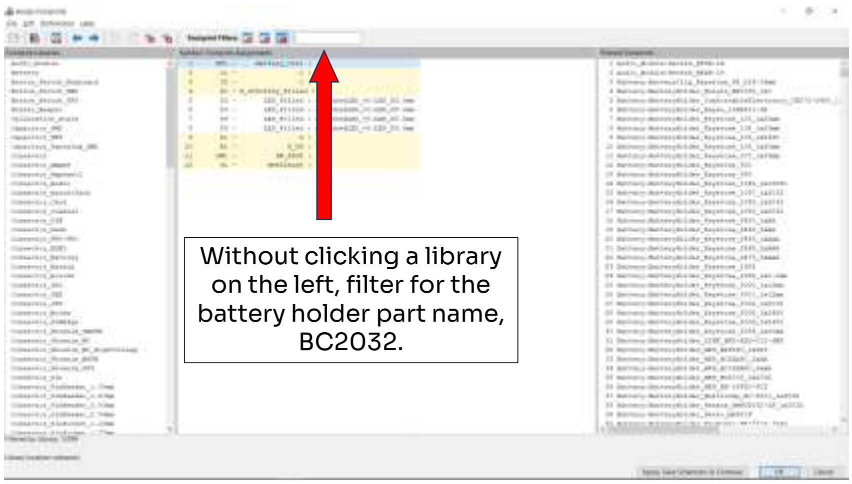
The middle section is all the components in the schematic, and any already-assigned footprints.

The columns are a bit confusing – the symbols are listed alphabetically by reference designator in the second column, with the left-most column just being the number in the list. The third column is the symbol name, and the fourth column is the assigned footprint.

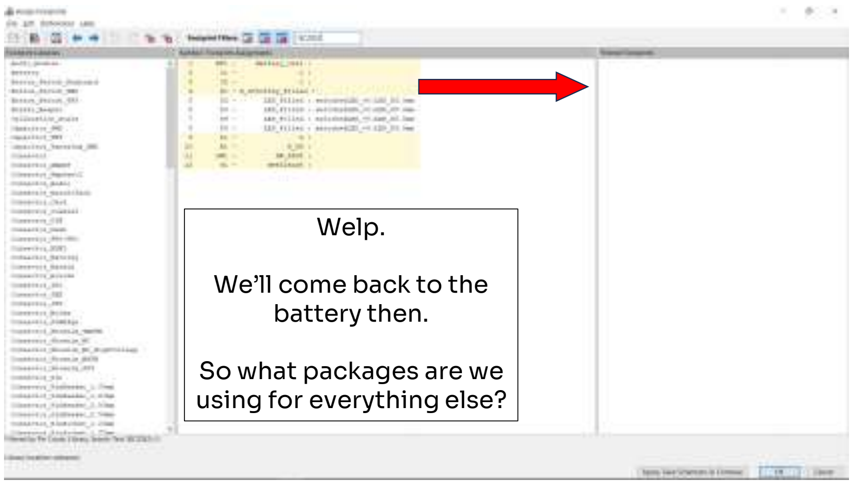
Assigning Footprints



Assigning Footprints



Assigning Footprints



Not necessary to

I didn't remember, either.

Description	Part Num.	Mounting	Footprint
LED drive IC	RT4526GJ6	SMD	TSOT-23-6 (3.1 x 1.8 x 1 mm)
Battery holder	BC2032-E2	TH	Custom
Switch	TS02-66-70-BK-160-LCR-D	TH	4-TH 6mm x 6mm
Cin, 2.2uF	C3216X5R1C225KT	SMD	1206/3116 (3.1 x 1.6 x 0.55 mm)
Cout, 1uF	C3216X7R1C105KT	SMD	1206/3116 (3.1 x 1.6 x 0.55 mm)
L, 22uH	LBR2518T220M (22uH)	SMD	1008/2518 (2.5 x 1.8 x 1.8 mm)
D	PMEG6030ELPX	SMD	SOD-128 (4 x 2.7 x 1.1 mm)
Rset, 30 Ω	Unknown (from kit)	SMD	1206/3116 (3.1 x 1.6 x 0.55 mm)
LED	C512A-WNN-CZ0B0151	TH	5mm diam, 0.6mm lead holes

Don't worry, no need to remember these. I'll remind you of them as we need them.

Note! Blindly using global footprints can leave you exposed to potential issues!

Parts or suppliers may say it’s a standard footprint, but only the datasheet is truth.

It’s up to you as the designer to confirm the dimensions of your parts and footprints.

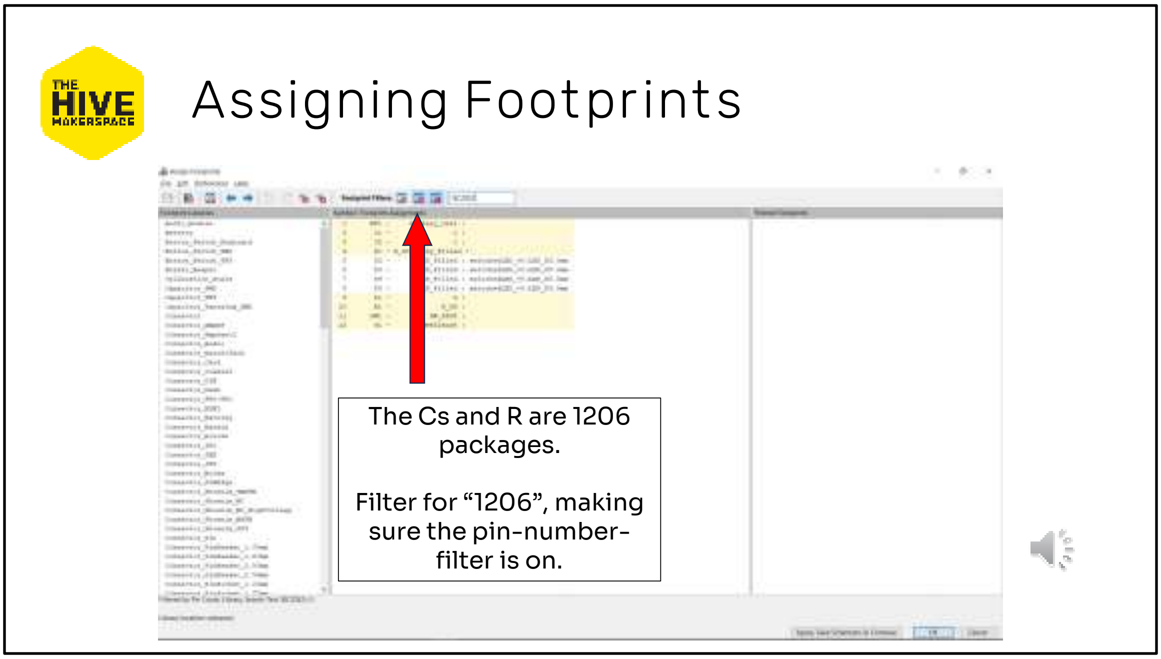
Failure to do so is at your own risk.

Assume, and make an ass out of you and me.

Aside: Package nomenclature

- Package nomenclature is *the worst*, and it's something you'll get familiar with as you design boards.
- There are so many standards, and the variation between some are so tiny. It can be really painful.
- The datasheet's mechanical drawings are the *one and only truth* for package sizes.
- Use caution!

This is a very (very) small subset of package families.



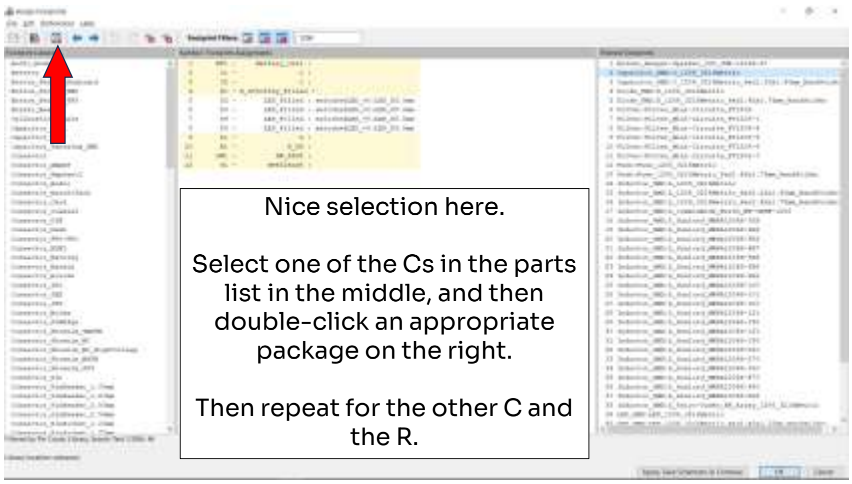
Back to assigning footprints.

If you’re unsure about which library to find footprints, or you know exactly the footprint you need, the filter box is the best way to limit your options.

The leftmost filter option uses some keywords in the symbol to narrow down the options. The middle option filters by number of pins in the symbol. The right one shows you only footprints in the libraries you’ve selected on the left, if any. I find the middle one most useful when I don’t know which library to look in, and the right one only when I know the library. I’ve selected those two in this screen here.

Then we can filter by the 1206 package size that the capacitors and the resistor are to find those.

Assigning Footprints

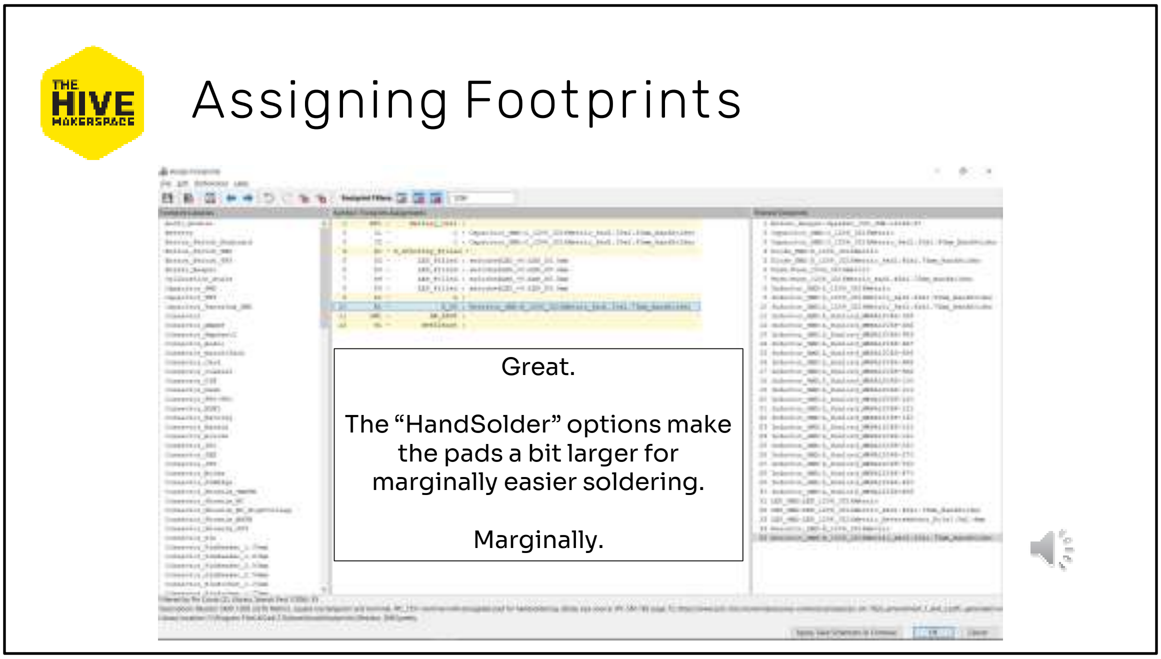


There are a number of options on the right here. Each footprint is labeled with it's library on the left of the colon, and the footprint name on the right of the colon.

For the capacitors, we want to find something that says "capacitor" or "C" and "1206". Similarly, for the resistor, something that says "resistor" or "R" and "1206".

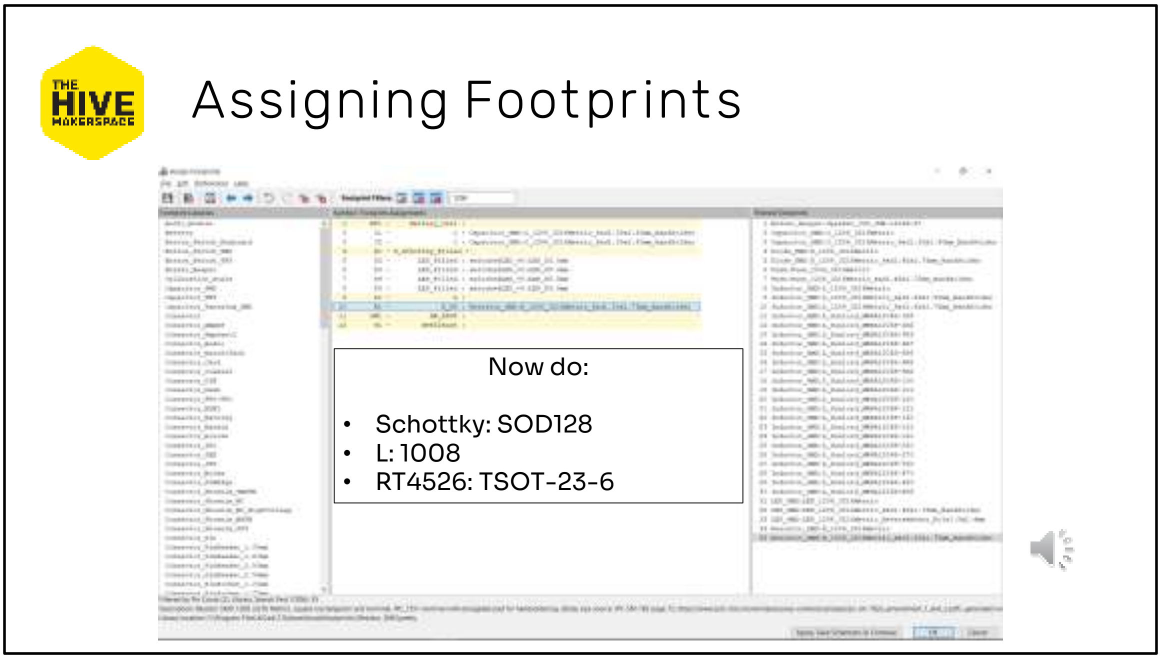
Pick the component in the middle pane to assign a footprint to, then double-click the footprint on the right to assign it.

*You can view the footprints themselves through this icon here to access the footprint viewer, from which you can also view its 3D model.



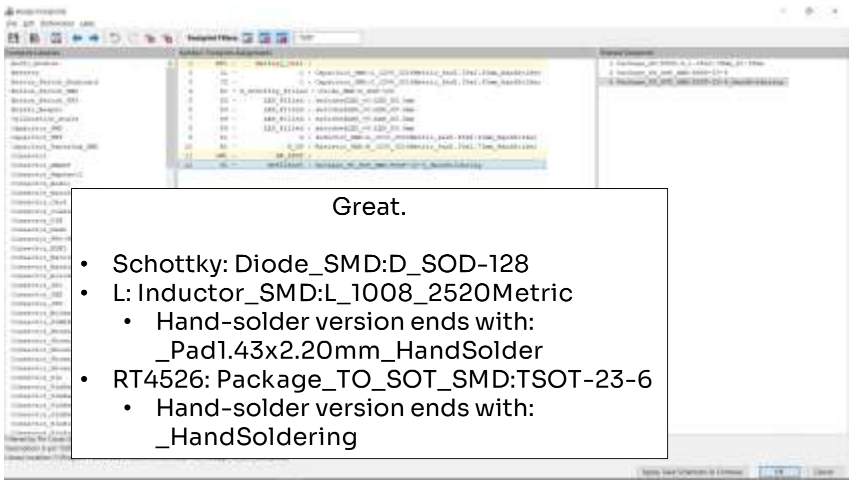
Hopefully you found the C_1206 and R_1206 footprints in the Capacitor SMD and Resistor SMD libraries, respectively, and assigned them like so.

Note that the “HandSolder” version of the footprints uses a slightly larger footprint that supposedly makes them easier to hand solder. It’s a pretty marginal change though.



Go ahead and add these three footprints to their schematic components. You can filter directly by the package name, given here. Make an educated guess as to the correct option if you're unsure. Pause the video for a minute to do this on your own before continuing.

Assigning Footprints

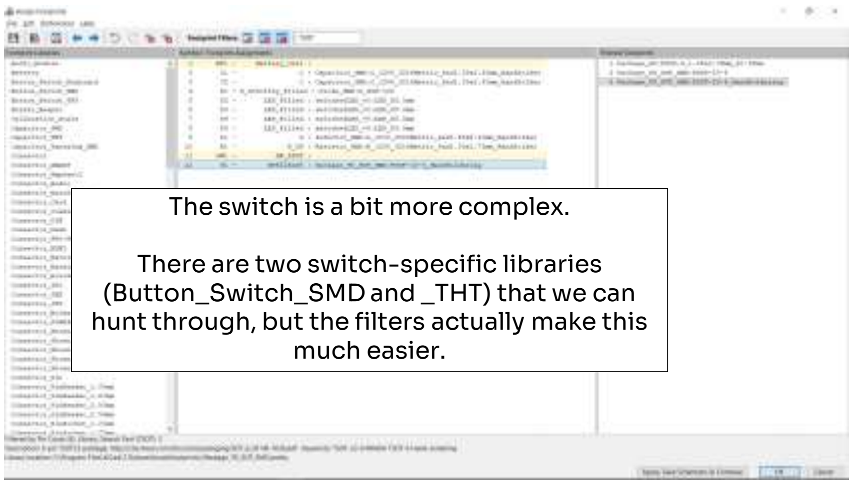


Great.

- Schottky: Diode_SMD:D_SOD-128
- L: Inductor_SMD:L_1008_2520Metric
  - Hand-solder version ends with: _Pad1.43x2.20mm_HandSolder
- RT4526: Package_TO_SOT_SMD:TSOT-23-6
  - Hand-solder version ends with: _HandSoldering

Great, hopefully you found those models okay. They’re listed here if not.

Assigning Footprints

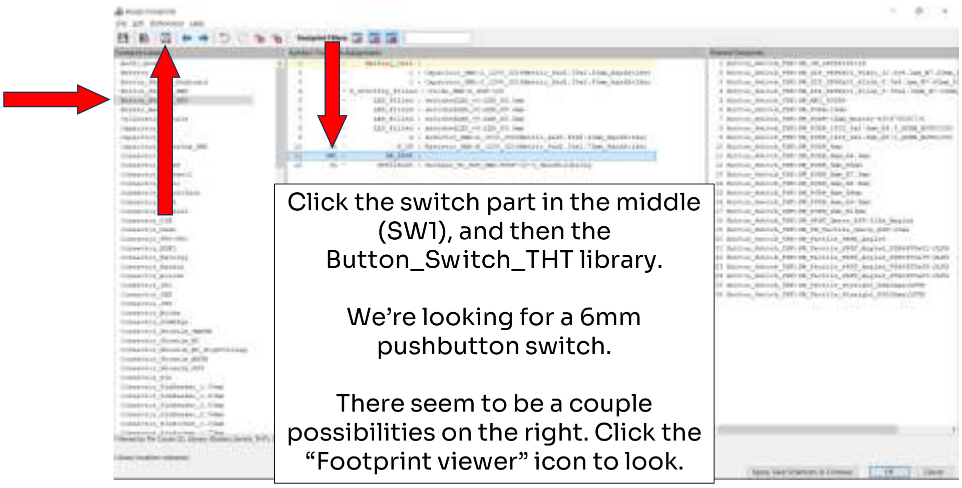


The switch is a bit more complex.

There are two switch-specific libraries (Button_Switch_SMD and _THT) that we can hunt through, but the filters actually make this much easier.

The switch is slightly more complex because there’s no standardize packages for switches. Or, at least, not in the same way as there are for passives and ICs.

Assigning Footprints

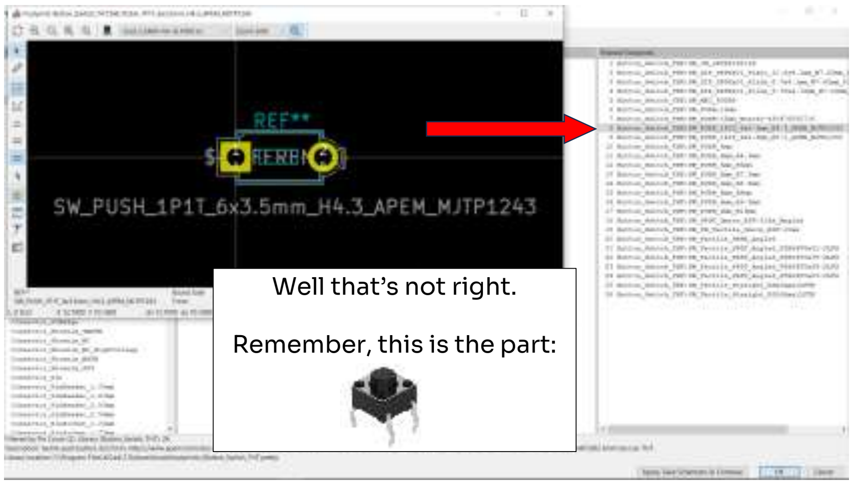


Click the switch part in the middle (SW1), and then the Button_Switch_THT library.

We're looking for a 6mm pushbutton switch.

There seem to be a couple possibilities on the right. Click the "Footprint viewer" icon to look.

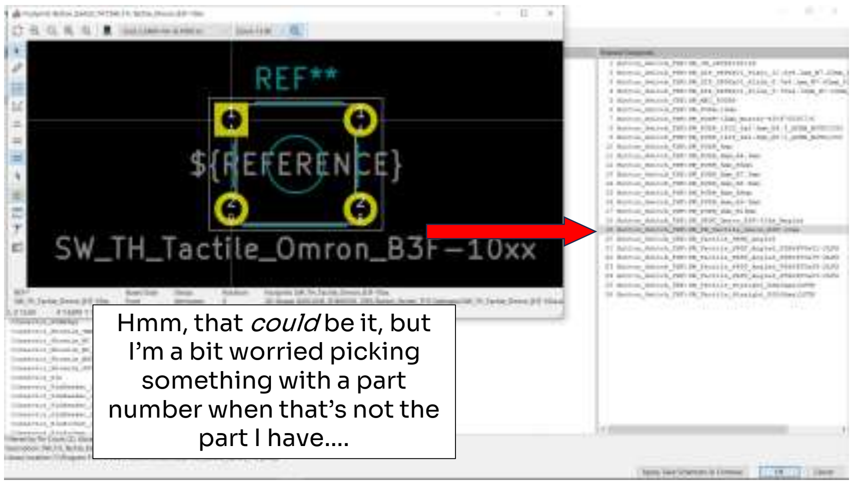
Assigning Footprints



Well that’s not right.

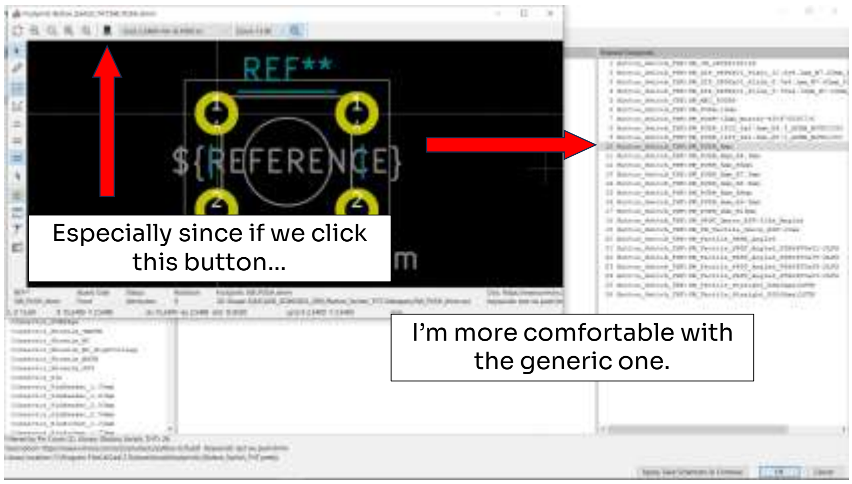
Remember, this is the part:

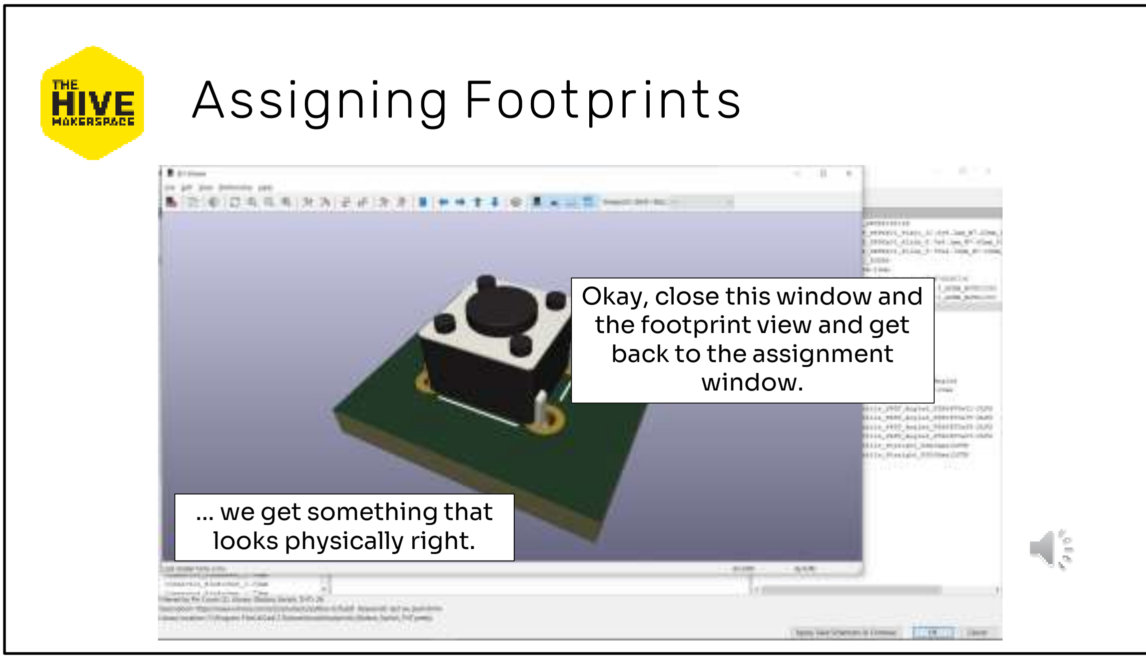
Assigning Footprints



Notice that the part number is given at the end, the BF3-10xx bit. Omron is the manufacturer, but you probably wouldn't know that beforehand.

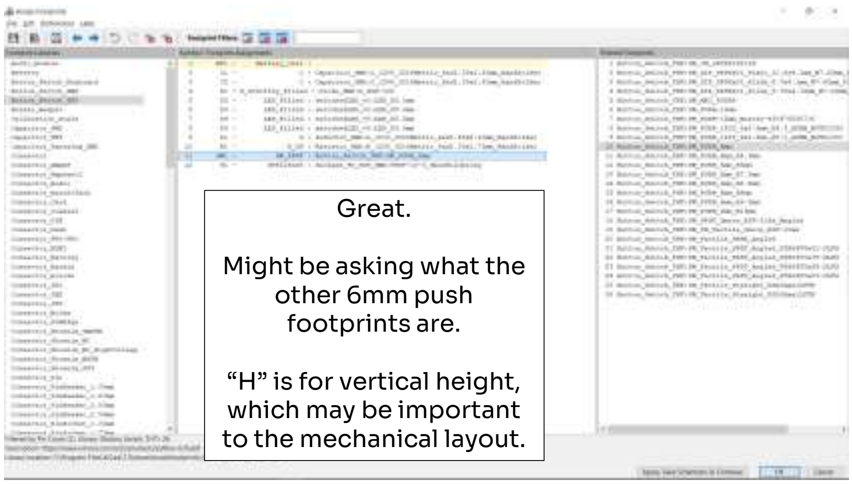
Assigning Footprints





Later, we'll go over a method to physically verify the footprint, and this selection can be changed later, as well. *

Assigning Footprints

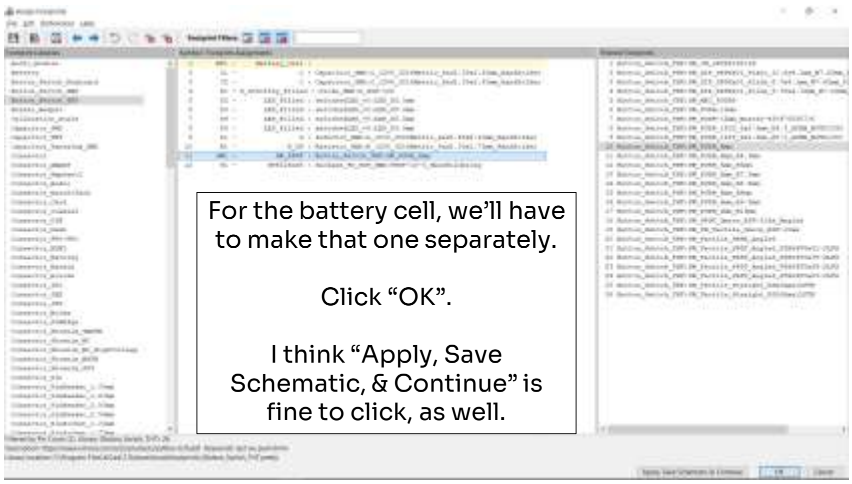


Great.

Might be asking what the other 6mm push footprints are.

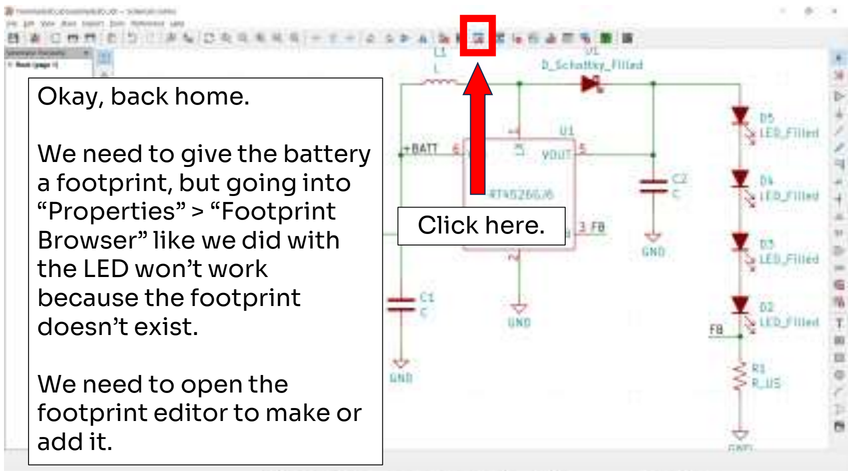
“H” is for vertical height, which may be important to the mechanical layout.

Assigning Footprints



Not necessary to

Creating a Footprint



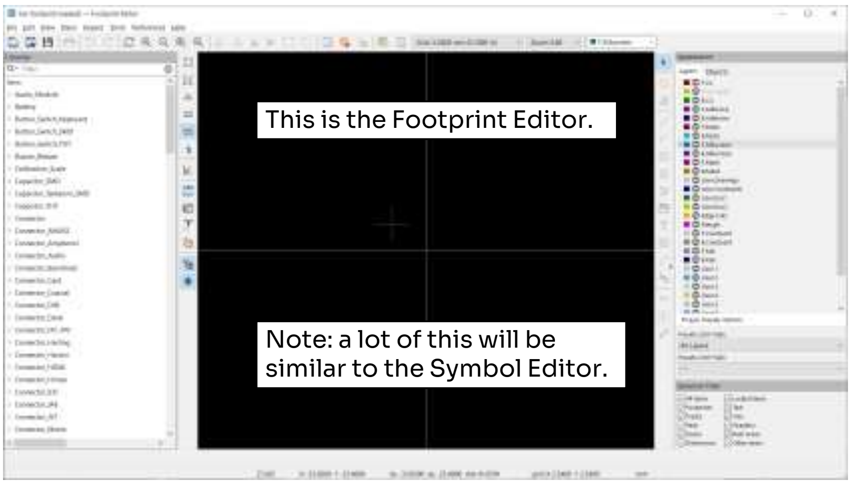
Okay, back home.

We need to give the battery a footprint, but going into “Properties” > “Footprint Browser” like we did with the LED won’t work because the footprint doesn’t exist.

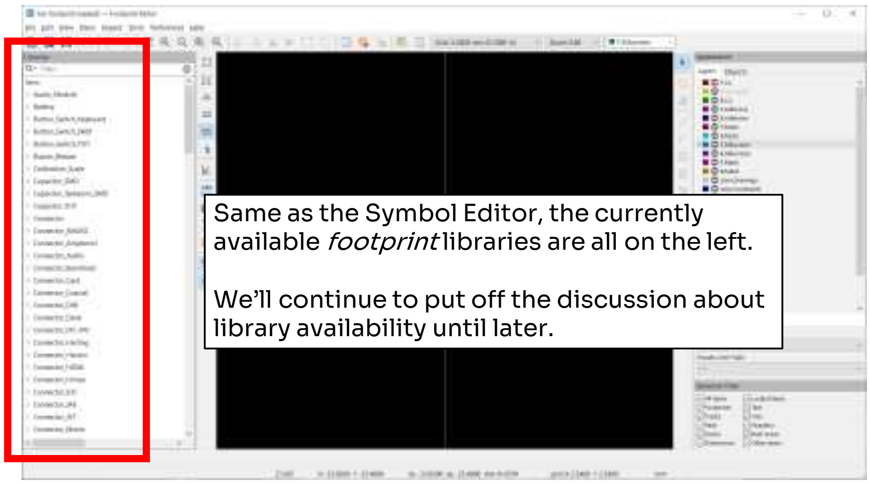
We need to open the footprint editor to make or add it.

Click here.

Footprint Library



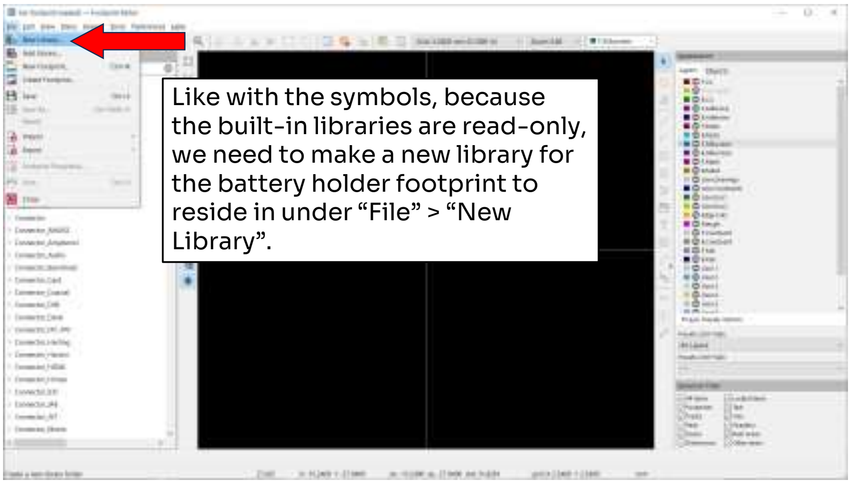
Footprint Library



Same as the Symbol Editor, the currently available *footprint* libraries are all on the left.

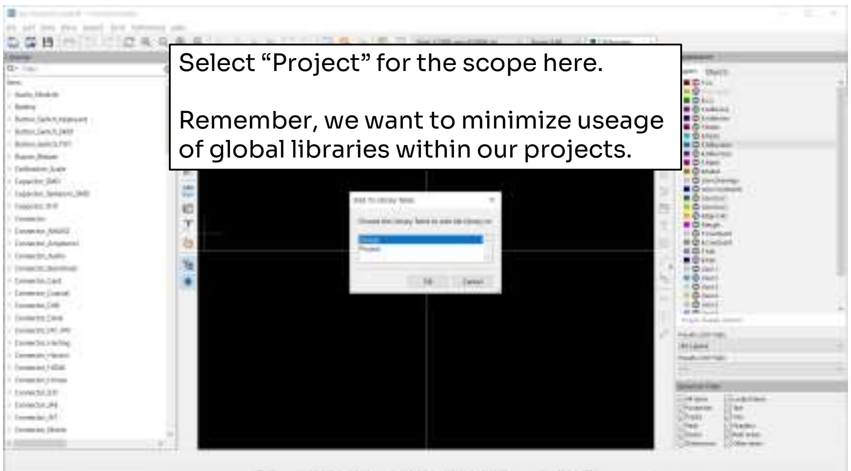
We'll continue to put off the discussion about library availability until later.

Footprint Library

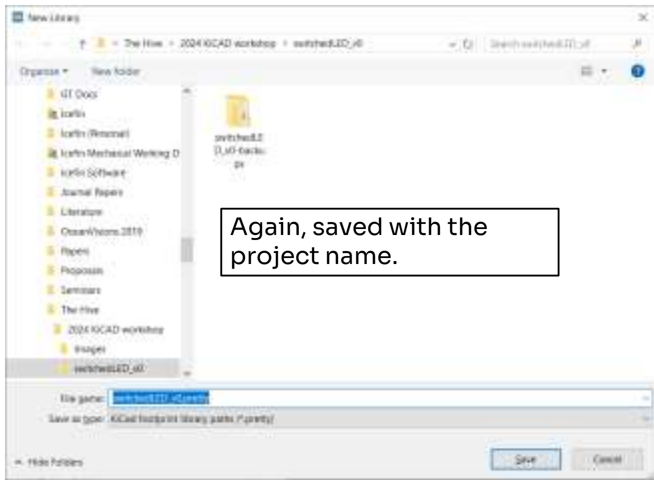


Like with the symbols, because the built-in libraries are read-only, we need to make a new library for the battery holder footprint to reside in under “File” > “New Library”.

Footprint Library

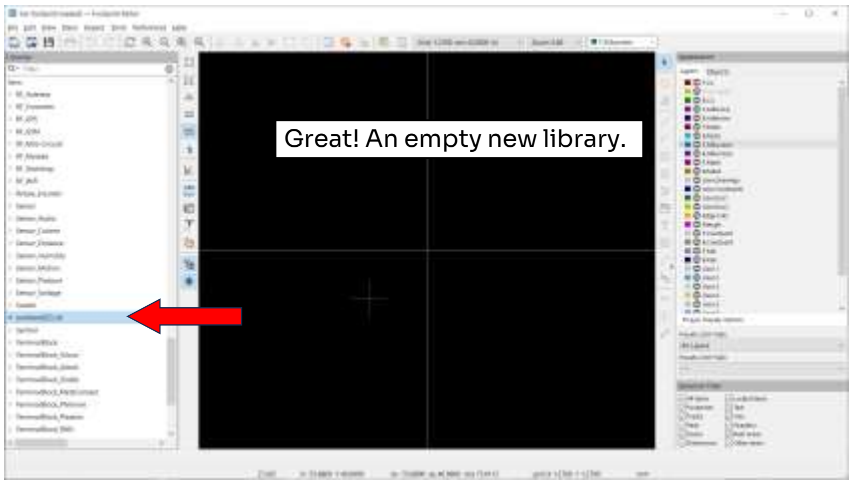


Footprint Library

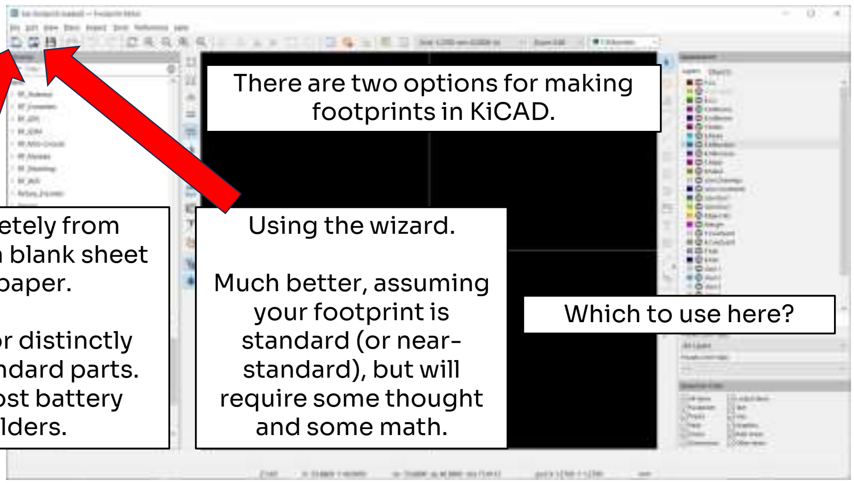


Again, saved with the project name.

Footprint Library



Footprint Library



There are two options for making footprints in KiCAD.

Completely from scratch, a blank sheet of paper.

Good for distinctly non-standard parts. Like most battery holders.

Using the wizard.

Much better, assuming your footprint is standard (or near-standard), but will require some thought and some math.

Which to use here?

Footprint Library

Normally, I would just go look for the footprint on UltraLibrarian or SnapMagic first.

But for this tutorial, we'll go look for the datasheet first to see what the footprint looks like, and then check online.

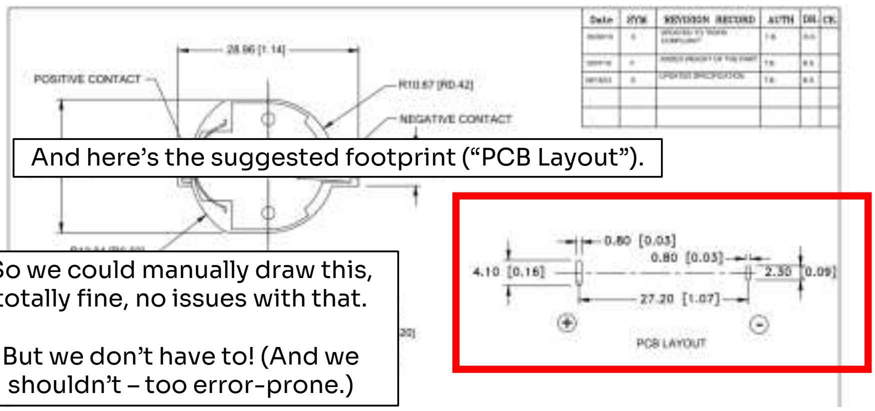
Footprint Library

I had to hunt down the technical drawing for the battery holder on the manufacturer's website because the datasheet link on Digikey gave me their catalogue.

Not helpful.

Footprint Library

Here's the technical drawing (the top half, at least).

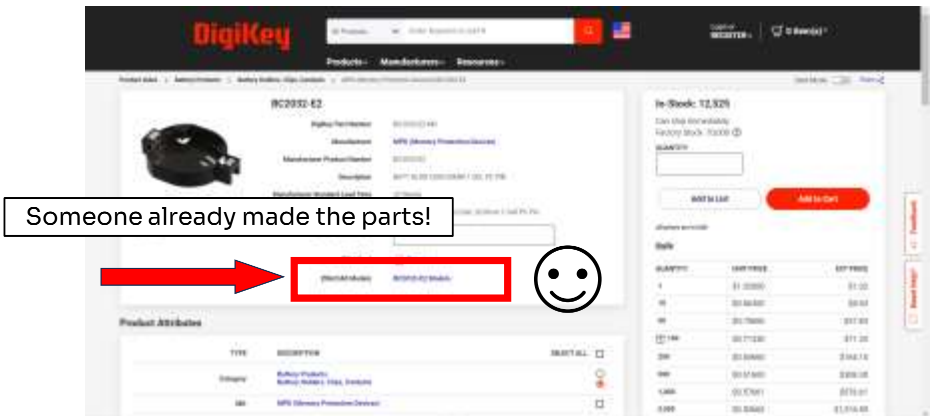


And here's the suggested footprint ("PCB Layout").

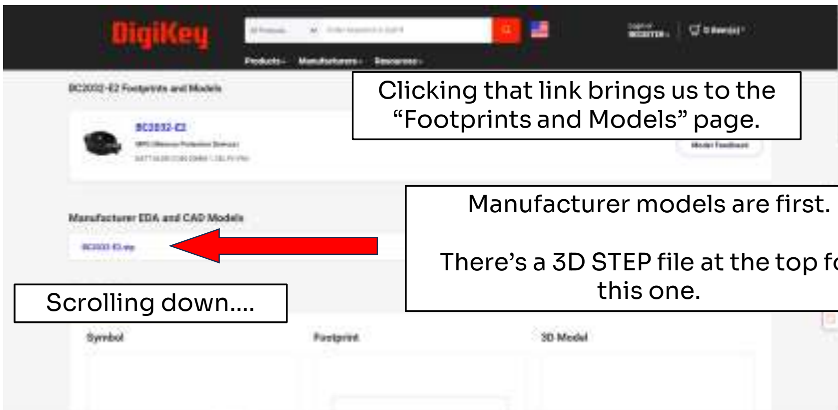
So we could manually draw this, totally fine, no issues with that.

But we don't have to! (And we shouldn't – too error-prone.)

Footprint Library



Footprint Library



Clicking that link brings us to the "Footprints and Models" page.

Manufacturer models are first.

There's a 3D STEP file at the top for this one.

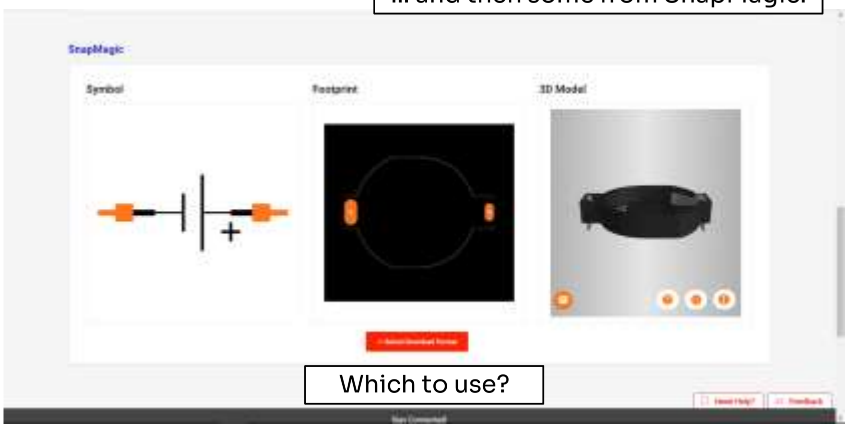
Scrolling down….

Footprint Library

... There's a set of models from UltraLibrarian...

Footprint Library

... and then some from SnapMagic.



Which to use?

Footprint Library

There is no right or wrong answer here generally. Both models are extremely likely to be sufficient.

It's on you to confirm that though!

I'm going to use SnapMagic here because I like the footprint better (and there's a 3D model).

Footprint Library

Click "Select Download Format" and select "KiCAD (V6 or later)".

(Obviously select a different format if you're using different software.)

I also selected the 3D CAD model because a 3D view of the designed PCB is useful.

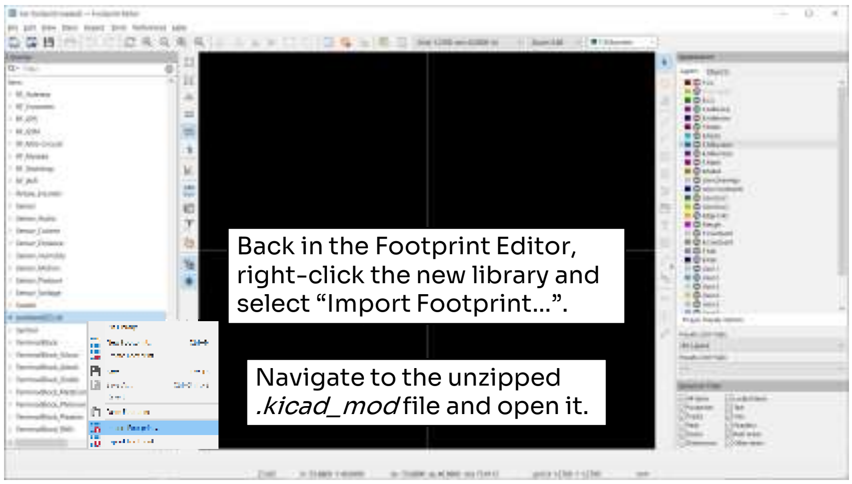
Footprint Library

Unzip the download somewhere accessible.

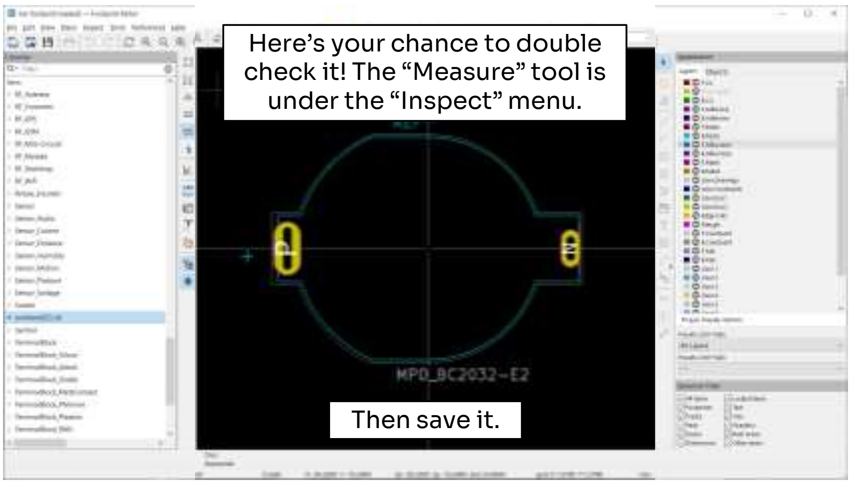
There will be five five files:

- a *.kicad_sym* symbol library
- a *.kicad_mod* footprint model library
- a *.step* 3D model
- a *.htm* link to a “How to import” webpage
- a license textfile

Footprint Library



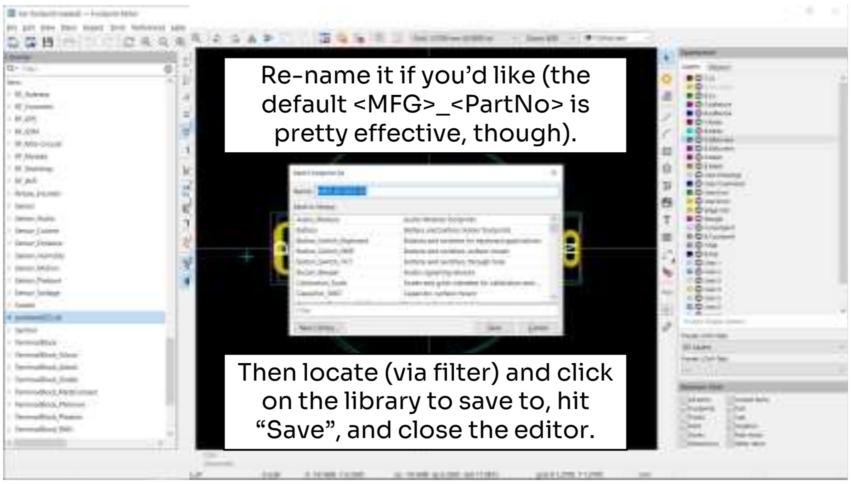
Footprint Library



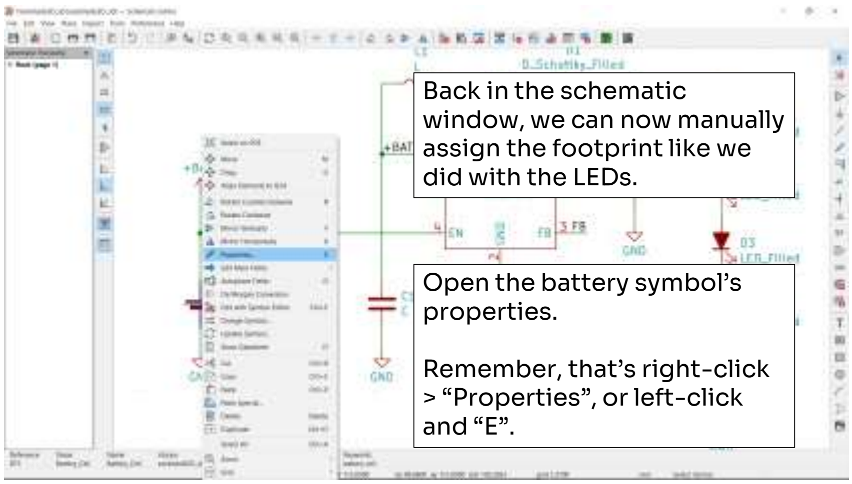
Good opportunity also to check the pin orientation with the datasheet. Is the positive pin actually the left one?

*Once you’re confirmed, save it

Footprint Library



Manual Footprint Assignment

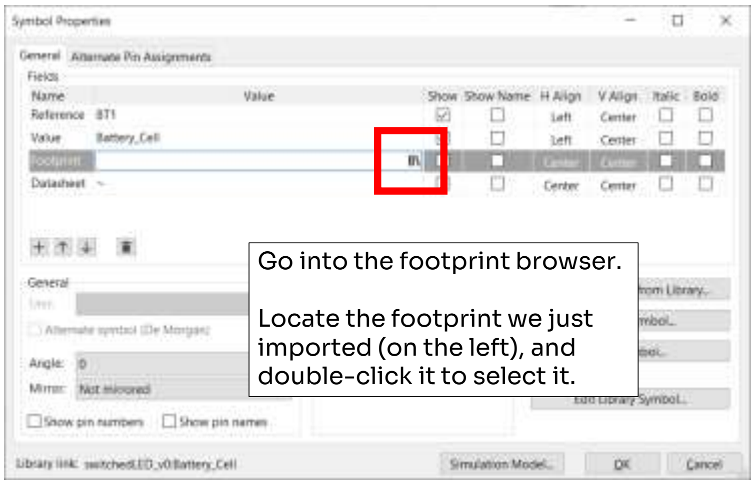


Back in the schematic window, we can now manually assign the footprint like we did with the LEDs.

Open the battery symbol's properties.

Remember, that's right-click > "Properties", or left-click and "E".

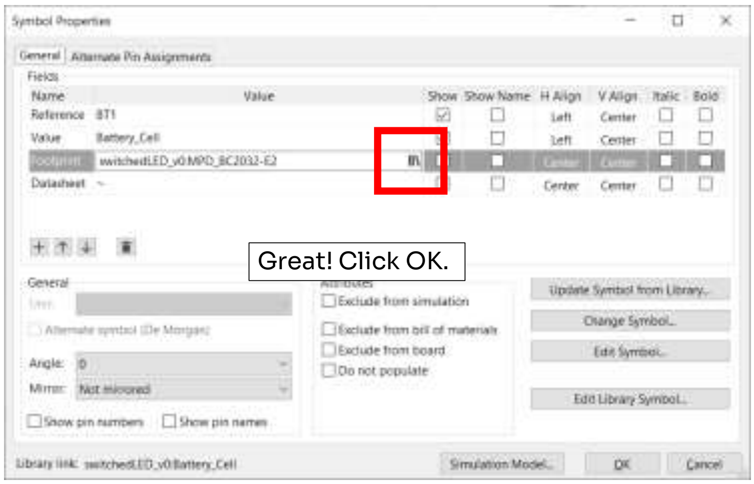
Manual Footprint Assignment



Go into the footprint browser.

Locate the footprint we just imported (on the left), and double-click it to select it.

Manual Footprint Assignment

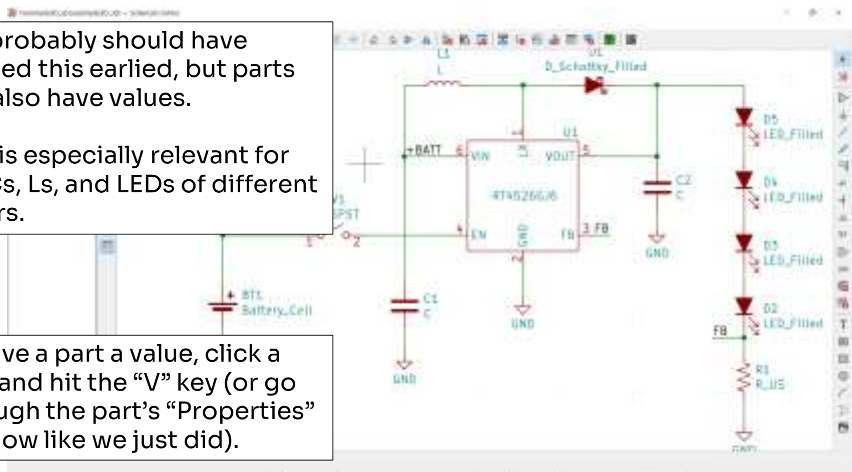


Great! Click OK.

Values

We probably should have tackled this earlied, but parts can also have values.

This is especially relevant for Rs, Cs, Ls, and LEDs of different colors.

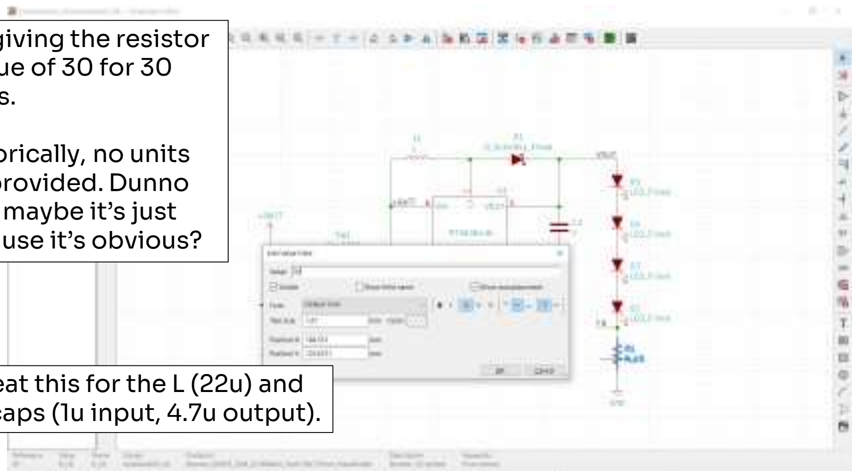


To give a part a value, click a part and hit the “V” key (or go through the part’s “Properties” window like we just did).

Values

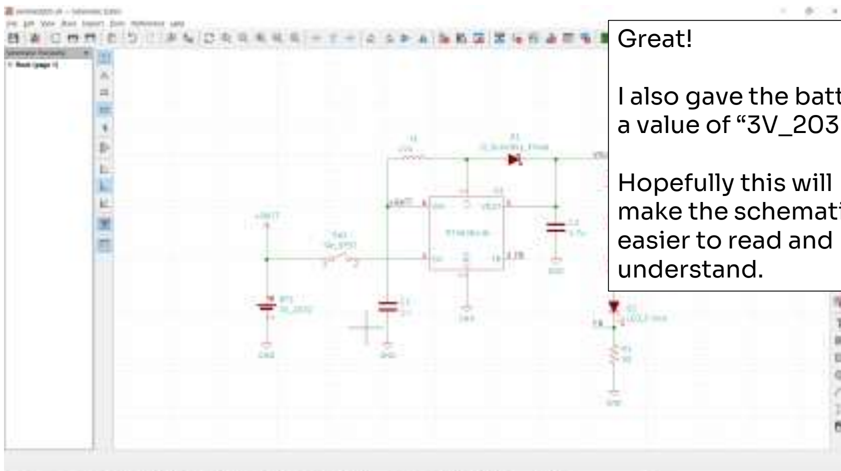
I've giving the resistor a value of 30 for 30 ohms.

Historically, no units are provided. Dunno why, maybe it's just because it's obvious?



Repeat this for the L (22u) and the caps (1u input, 4.7u output).

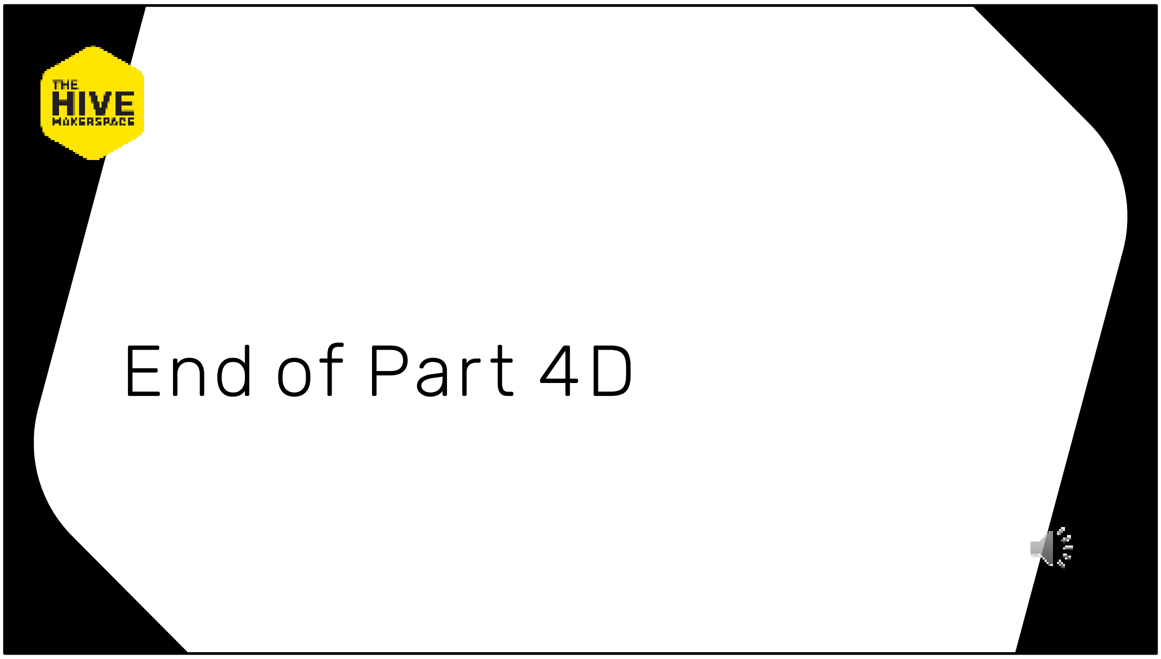
Values



Great!

I also gave the battery a value of “3V_2032”.

Hopefully this will make the schematic easier to read and understand.



And that ends part 4D of our video series on KiCAD, in which I covered assigning footprints, footprint libraries, importing pre-designed modles, and adding part values.

A PDF of this video is available as well, linked in the description and hosted on The Hive's Wiki.

In the next segment, part 4E, we'll wrap up the schematic drawing portion of the series with a discussion of ERC and some miscellaneous schematic tools you might want to be vaguely aware of. See you there.