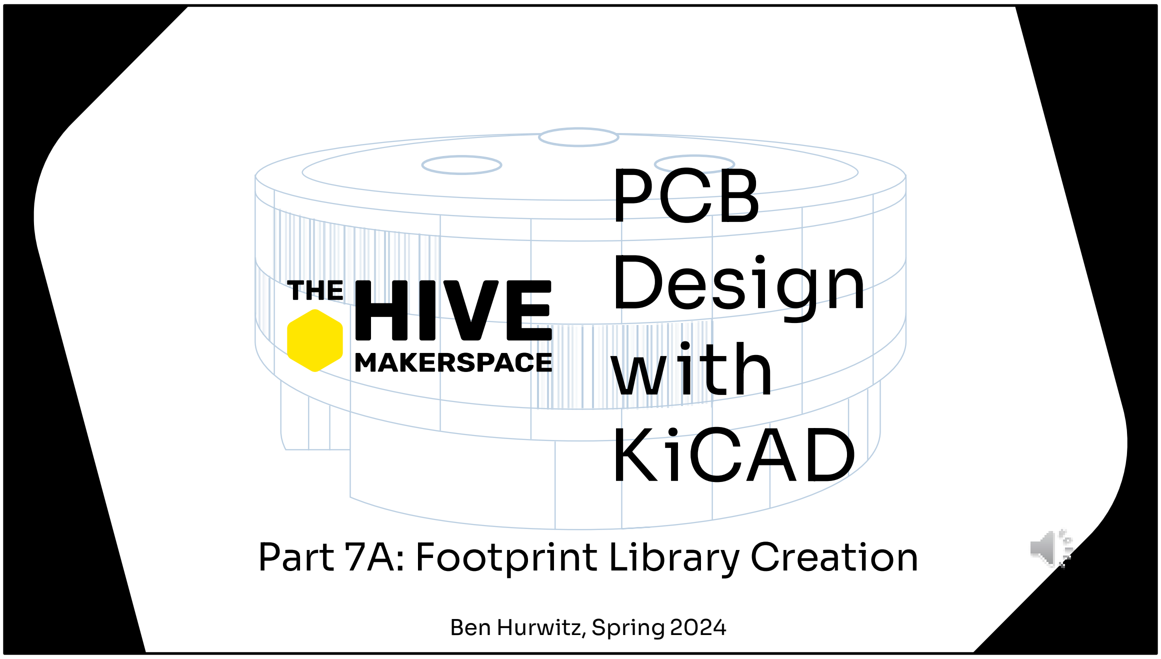


Hi all, welcome to The Hive's series on PCB Design with KiCAD. My name is Ben, and in this series, we've been walking through the PCB design process using KiCAD as our electronics design software.

The previous videos went through the design process all the way through, resulting in a complete PCB ready for fabrication. One thing that I mentioned during that process, and was featured in the original "EDA Design Flow" in part 2, was library management, and the idea of using only project-scope libraries, but when we actually did the design, I ignored this for simplicity and time-constraints.

The last video, part 6, went through a project-specific symbol library.

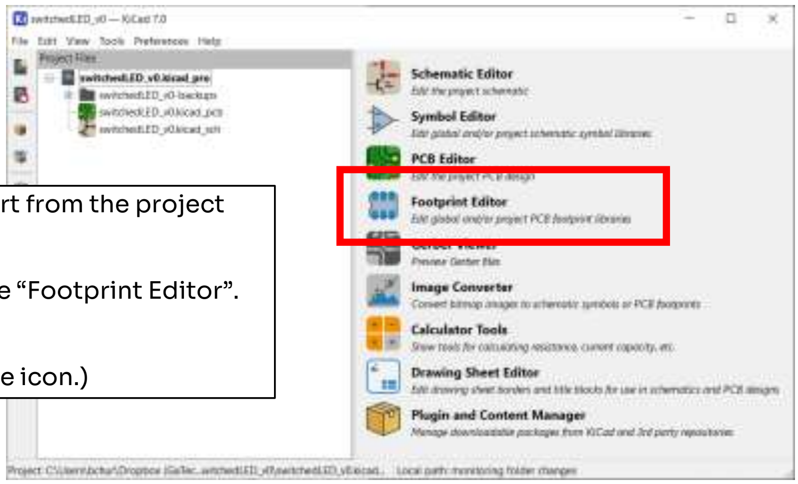
In this video, I will walk you through generating a single project-scoped footprint library to package with the rest of your project, and keep your work insulated from external changes, and then populating it with some of KiCAD's built-in models.

This material is of course not required for a functional design, but it is good design practice, for KiCAD at least, to keep all your parts in a project-level library.

Because this is not related directly to the design flow of the previous videos, I'll make no assumptions about the state of your system or knowledge. So I apologize if some of this is repetition for some of you.

Let's get started.

Footprint Library

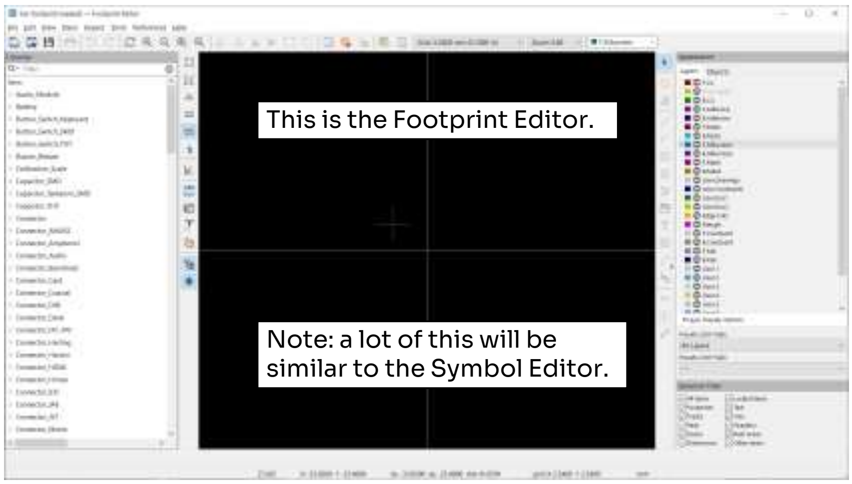


We'll start from the project window.

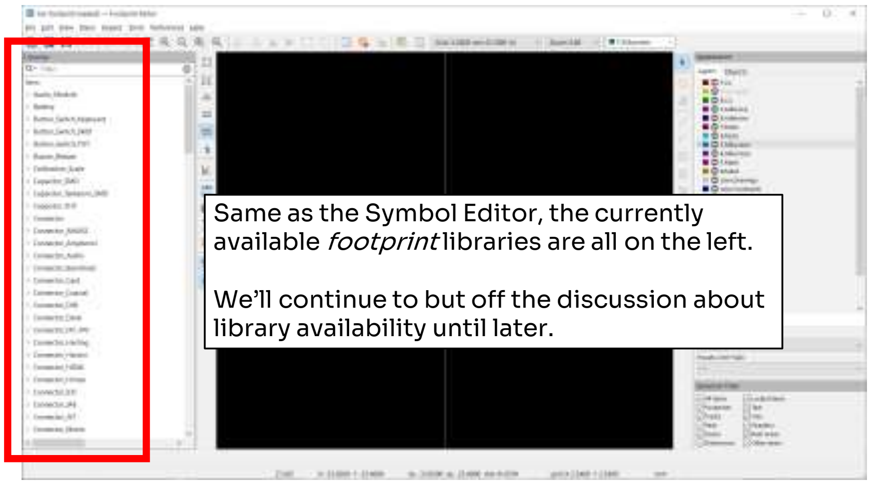
Open the "Footprint Editor".

(Click the icon.)

Footprint Library



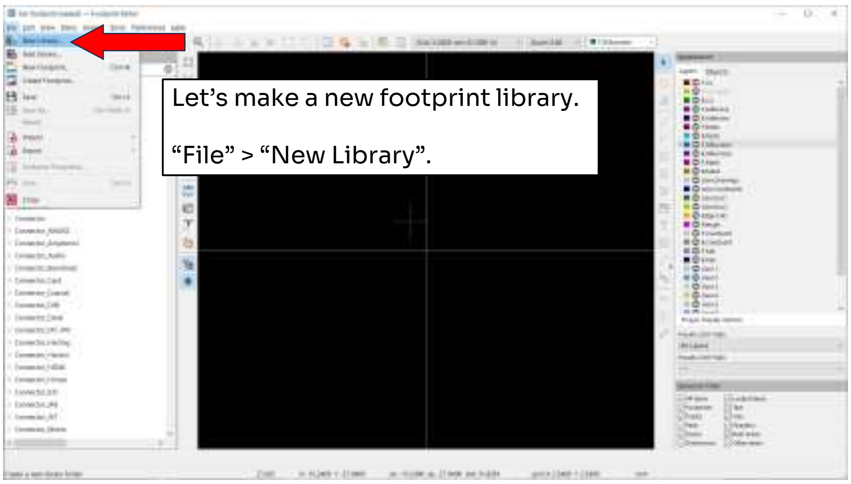
Footprint Library



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

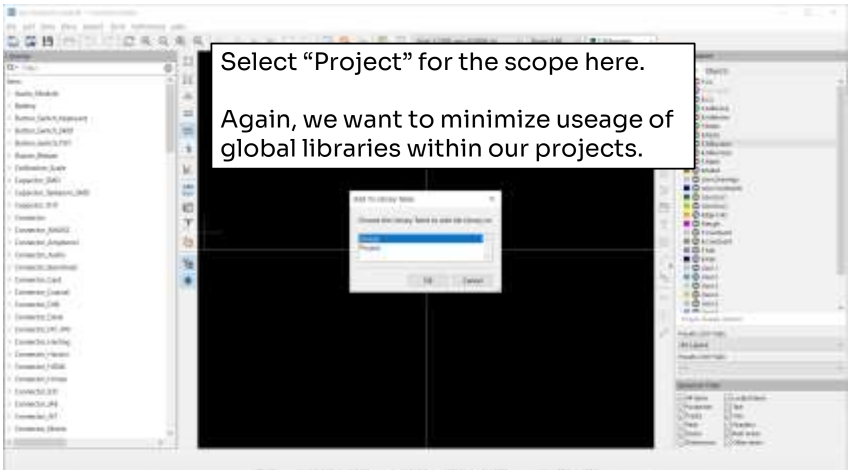
We'll continue to but off the discussion about library availability until later.

Footprint Library



We'll be using the flashlight circuit that was developed in videos part 1-5 as our parts list to add, so if you've already made a footprint library during those videos, don't bother to make another one.

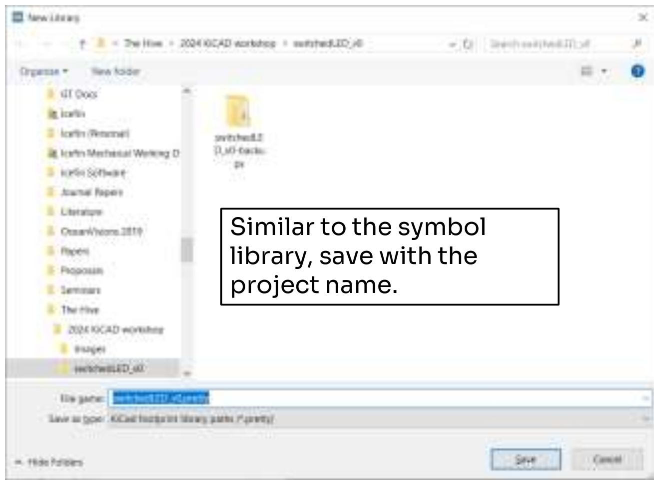
Footprint Library



Select "Project" for the scope here.

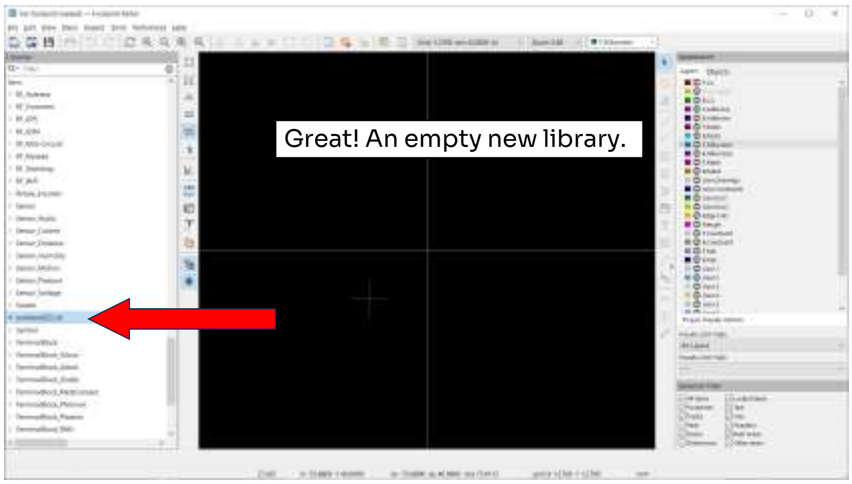
Again, we want to minimize useage of global libraries within our projects.

Footprint Library

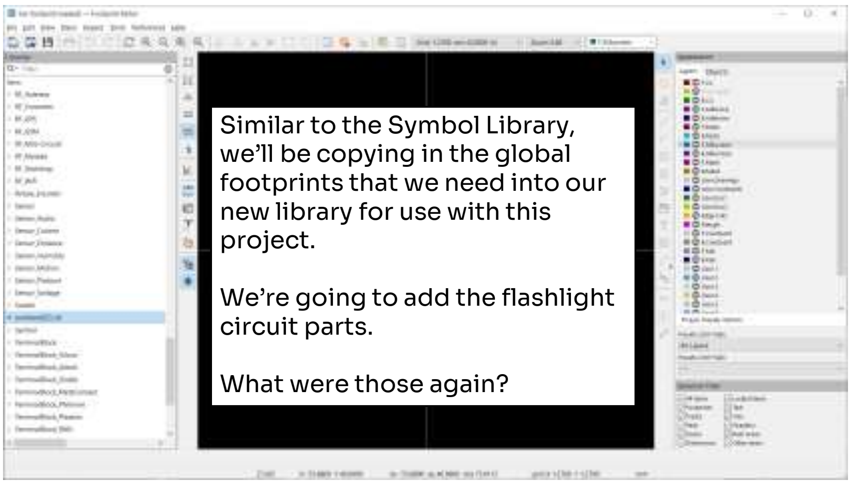


Similar to the symbol library, save with the project name.

Footprint Library



Footprint Library



Parts List

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, you don't have to memorize this.

Note! Blindly using global footprints can leave you exposed to potential issues if the parts aren't actually standard.

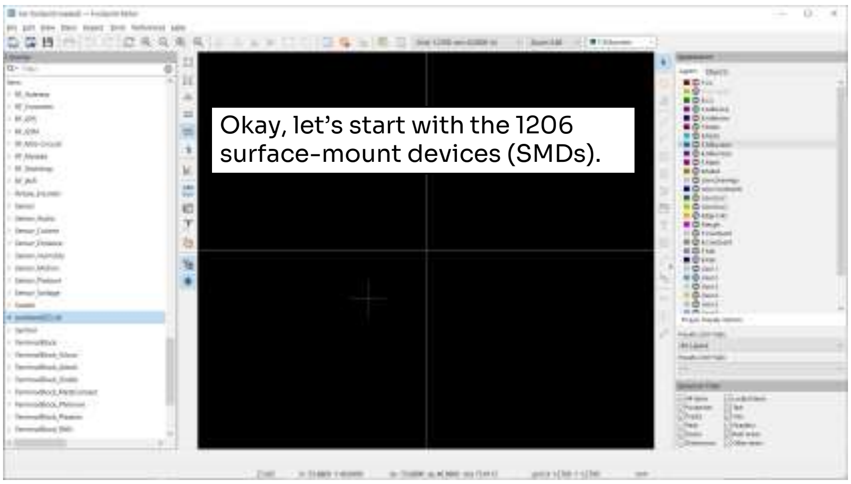
It's up to you as the designer to confirm the dimensions of your parts and footprints.

Failure to do so is at your own risk.

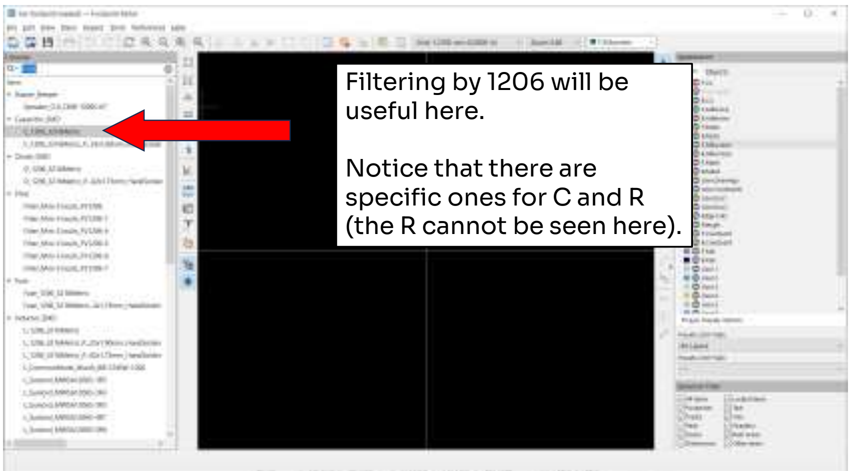
Assume, and make an ass out of you and me.

(I'll leave that for an exercise for the reader.)

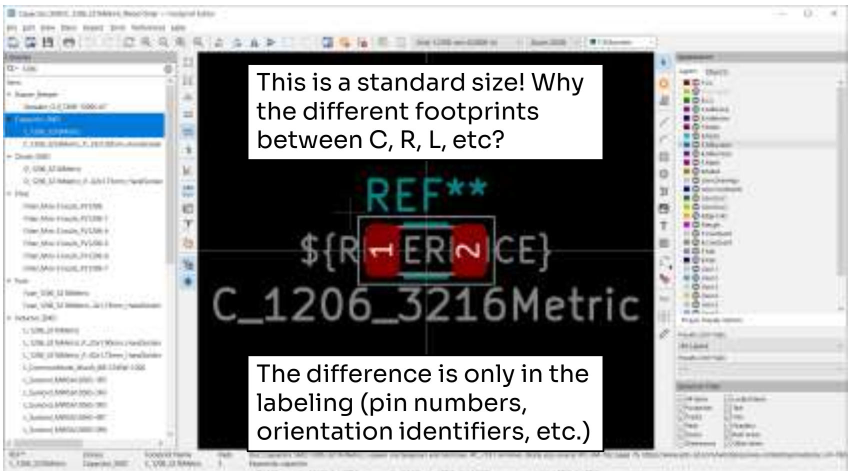
Footprint Library



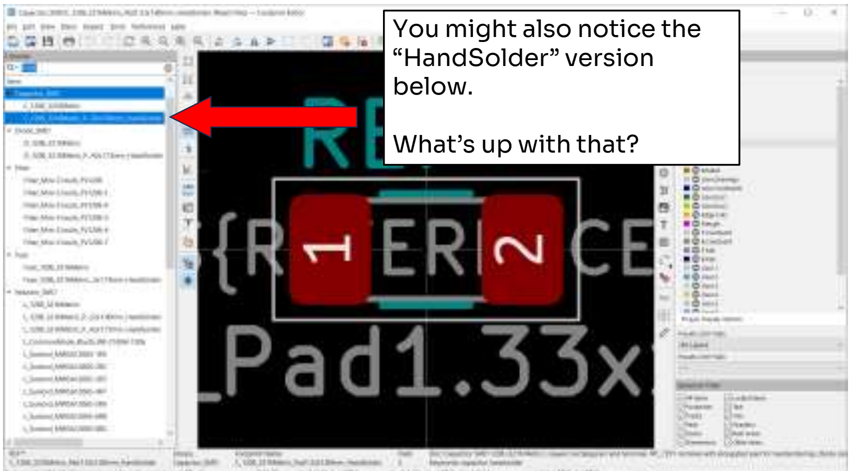
Footprint Library



Footprint Library



Footprint Library



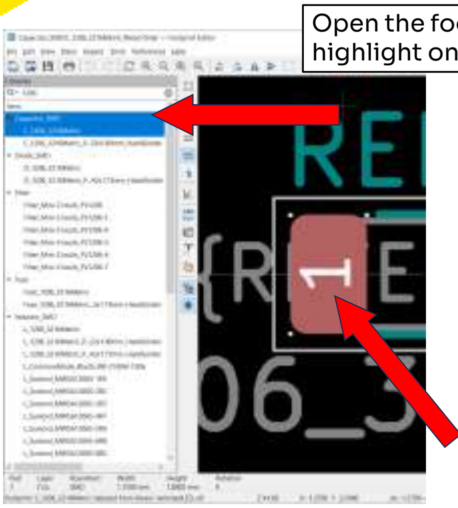
You might also notice the “HandSolder” version below.

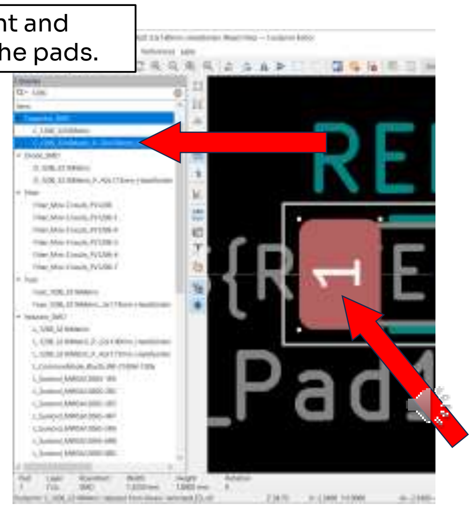
What’s up with that?

(Answer on next slides)

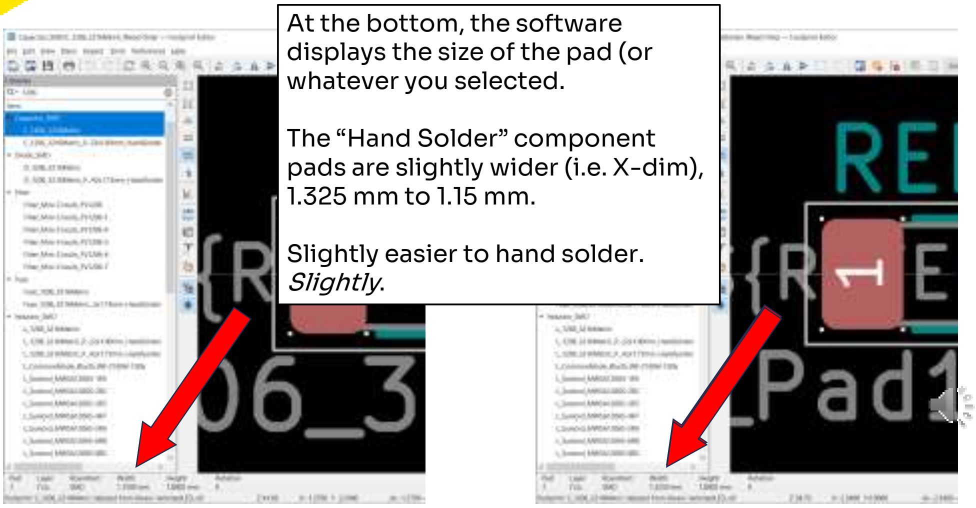
Footprint Library

Open the footprint and highlight one of the pads.





Footprint Library

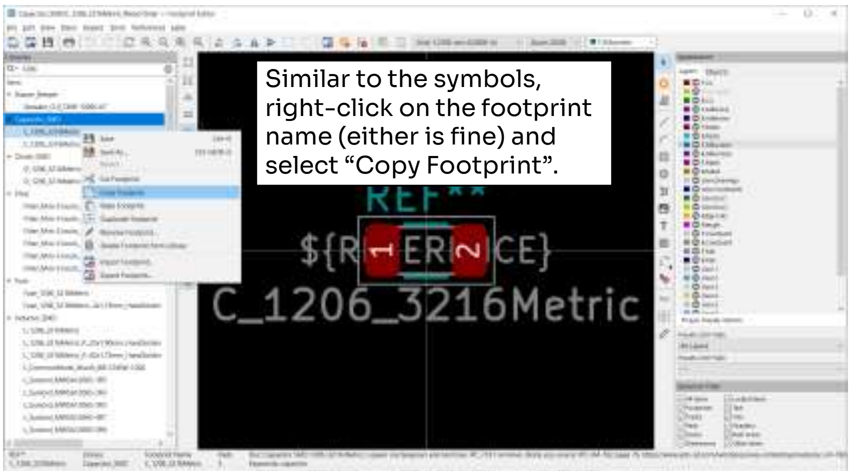


At the bottom, the software displays the size of the pad (or whatever you selected.

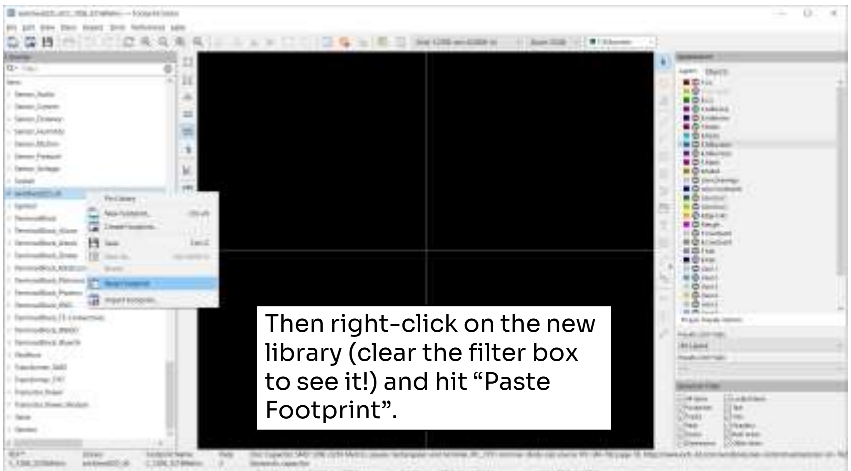
The “Hand Solder” component pads are slightly wider (i.e. X-dim), 1.325 mm to 1.15 mm.

Slightly easier to hand solder. *Slightly*.

Footprint Library

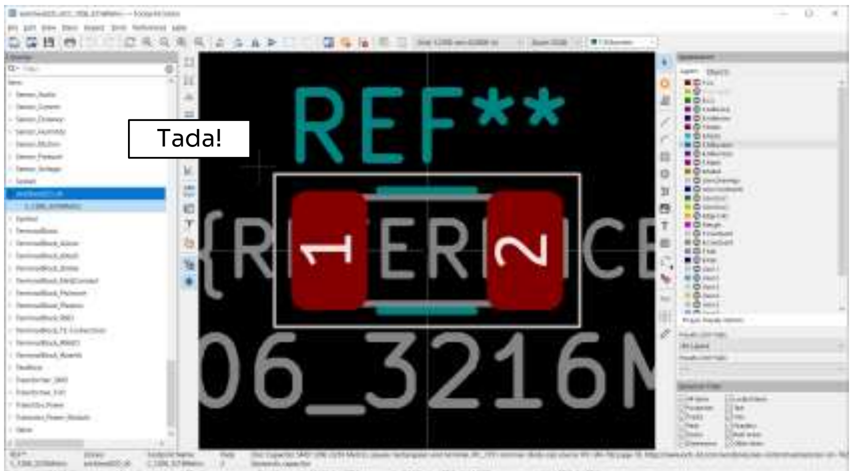


Footprint Library



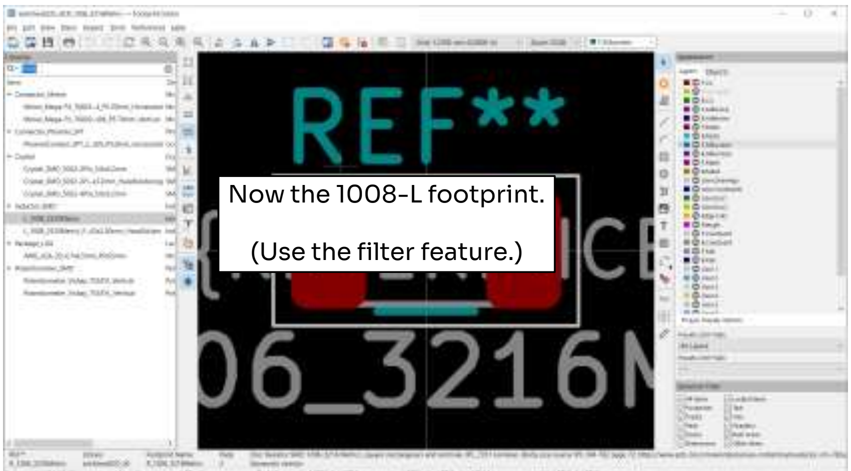
Then right-click on the new library (clear the filter box to see it!) and hit “Paste Footprint”.

Footprint Library

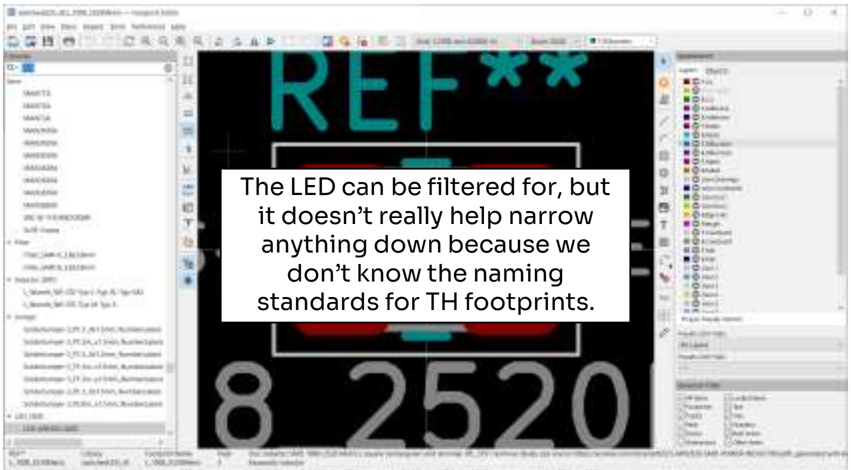


Footprint Library

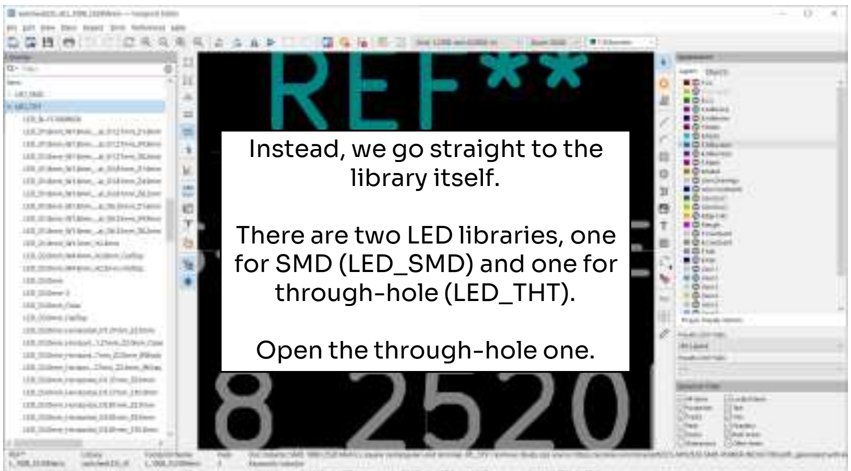
Footprint Library



Footprint Library



Footprint Library

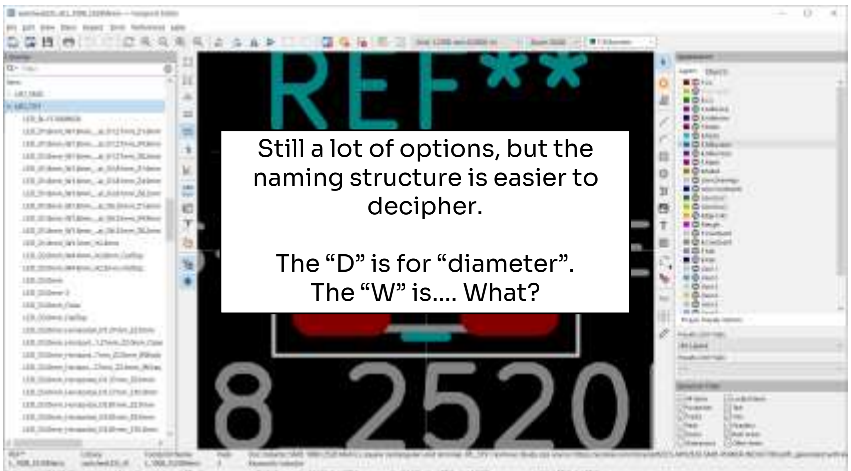


Instead, we go straight to the library itself.

There are two LED libraries, one for SMD (LED_SMD) and one for through-hole (LED_THT).

Open the through-hole one.

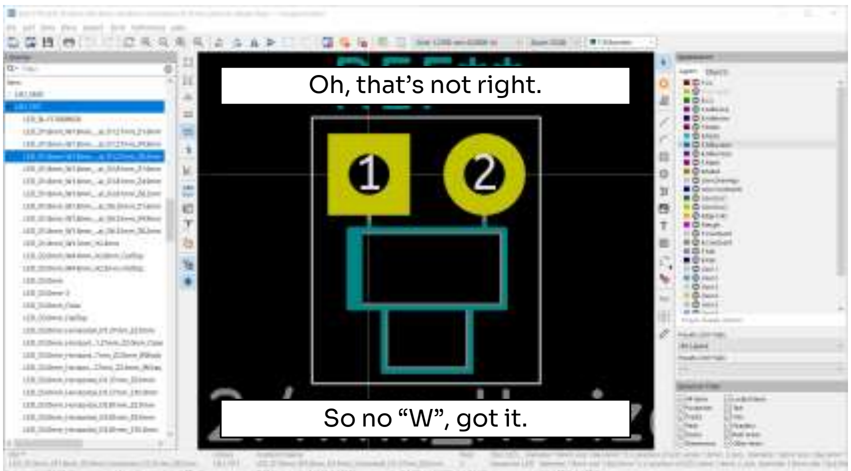
Footprint Library



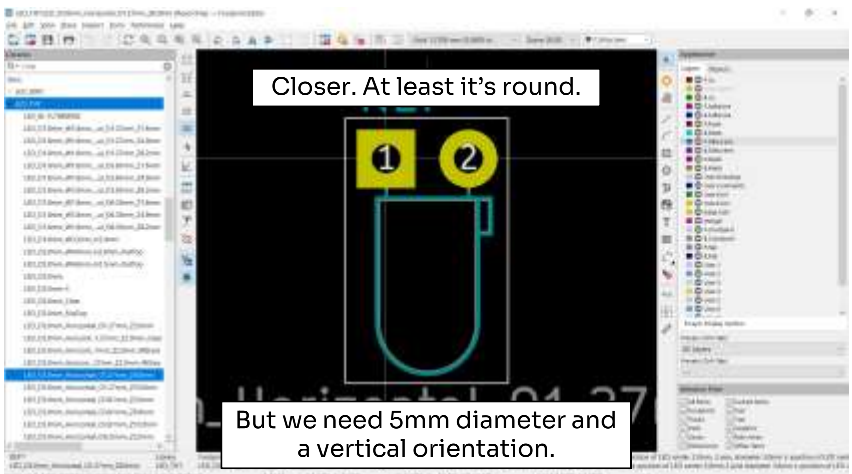
Still a lot of options, but the naming structure is easier to decipher.

The “D” is for “diameter”.
The “W” is…. What?

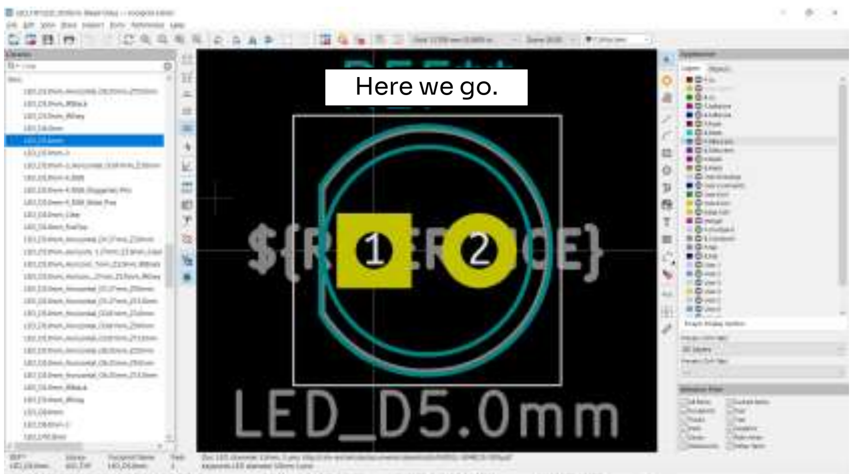
Footprint Library



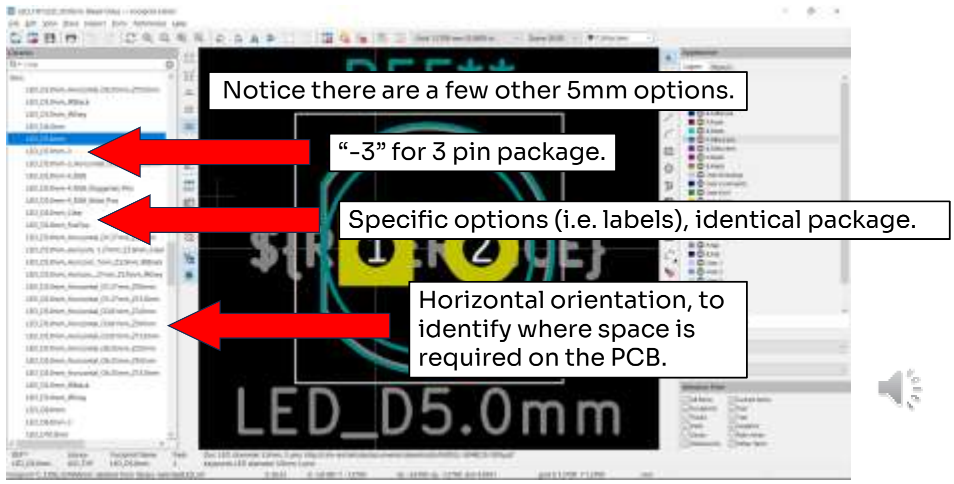
Footprint Library



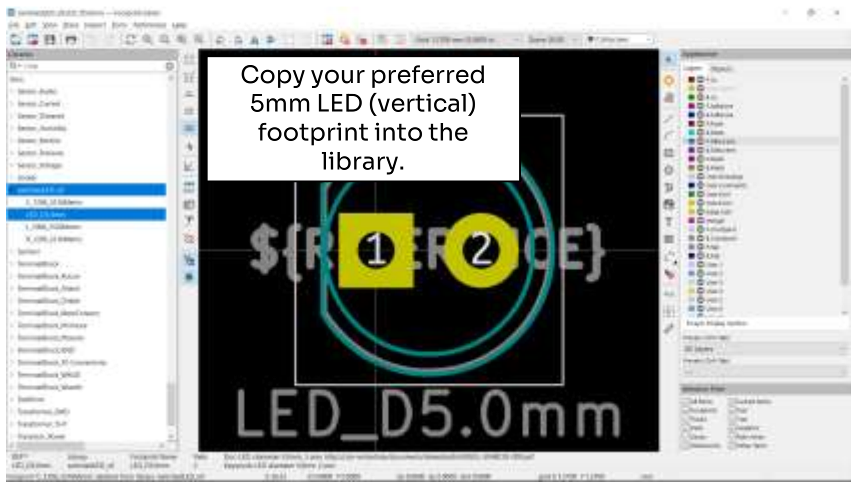
Footprint Library



Footprint Library



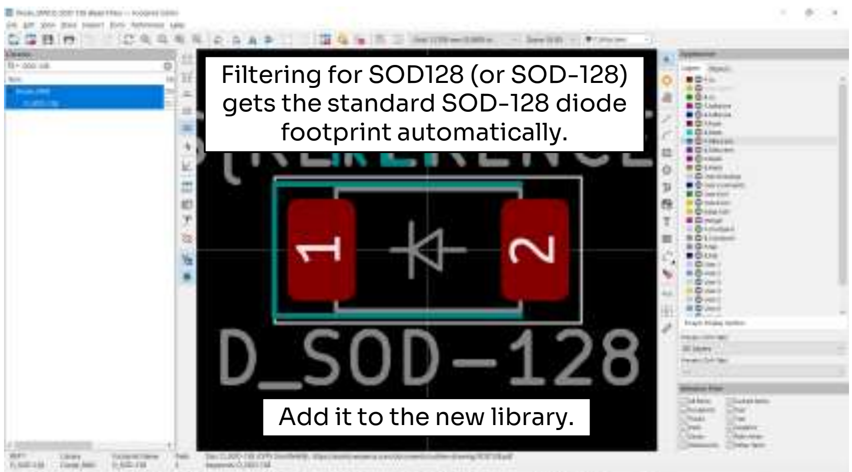
Footprint Library



What’s left?

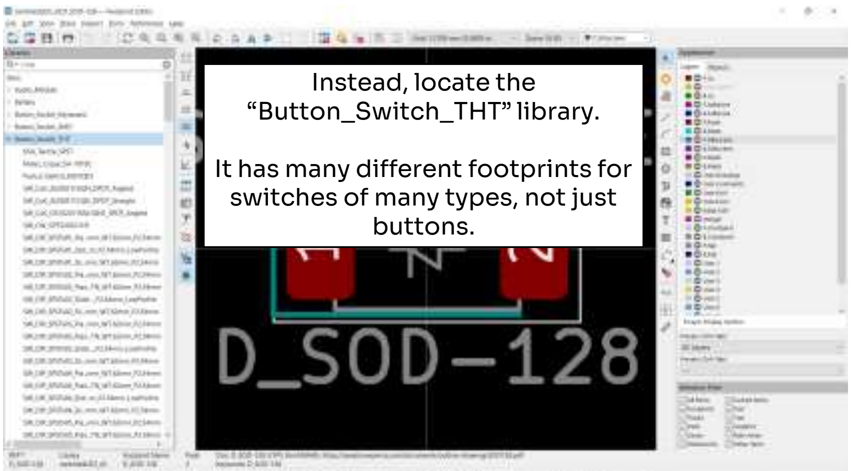
	Description	Part Num.	Mounting	Footprint
	LED drive IC	RT4526GJ6	SMD	TSOT-23-6 (≤ 3.1 x 1.8 x 1 mm)
	Battery holder	BC2032-E2	TH	Custom
	Switch	TS02-66-70-BK-160-LCR-D	TH	4-TH 6mm x 6mm
✓	~~Cin, 2.2uF~~	~~C3216X5R1C225KT~~	~~SMD~~	~~1206/3116 (3.1 x 1.6 x 0.55 mm)~~
✓	~~Cout, 1uF~~	~~C3216X7R1C105KT~~	~~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~~

Footprint Library

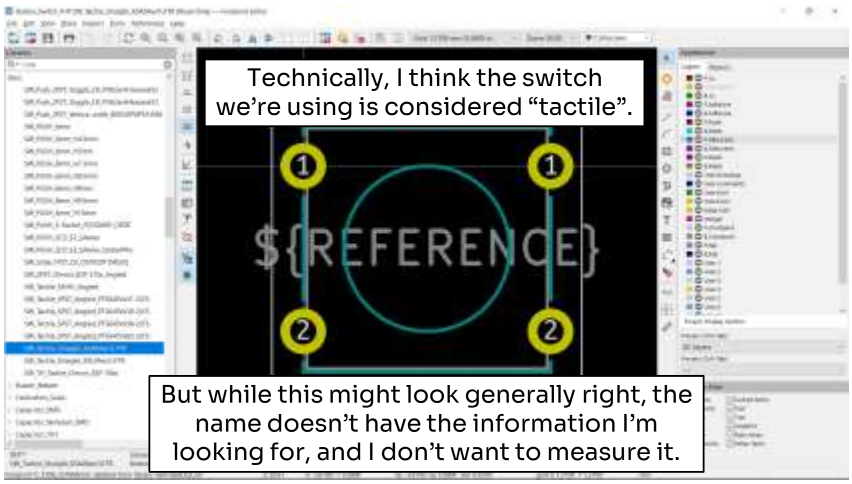


Footprint Library

Footprint Library



Footprint Library



Technically, I think the switch we're using is considered "tactile".

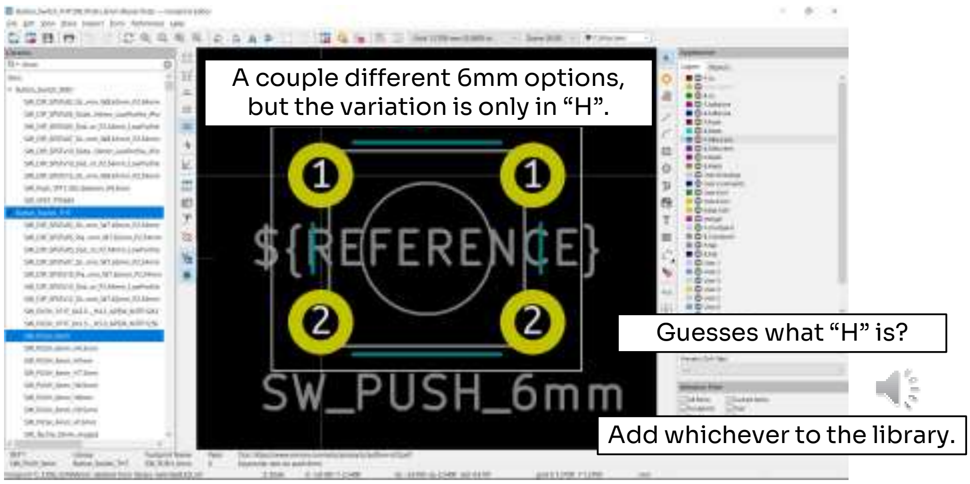
But while this might look generally right, the name doesn't have the information I'm looking for, and I don't want to measure it.

Footprint Library

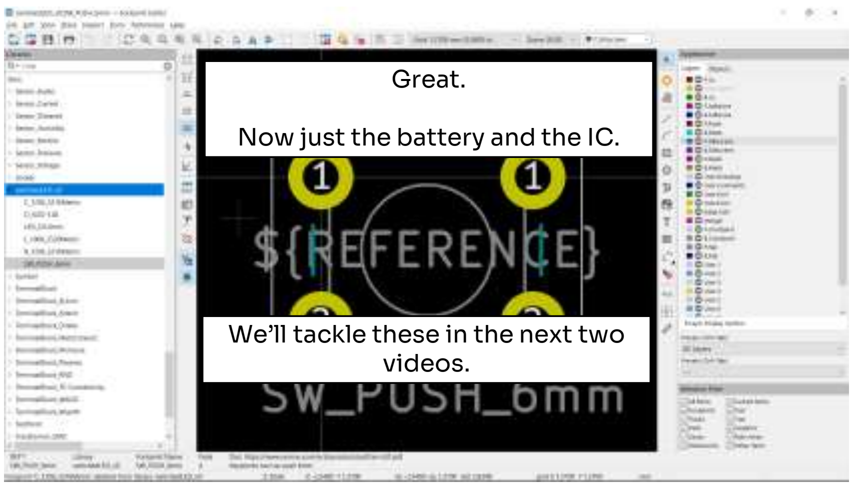
We can scroll through the library to locate the 6mm options, or filter on “6mm” and locate the library.

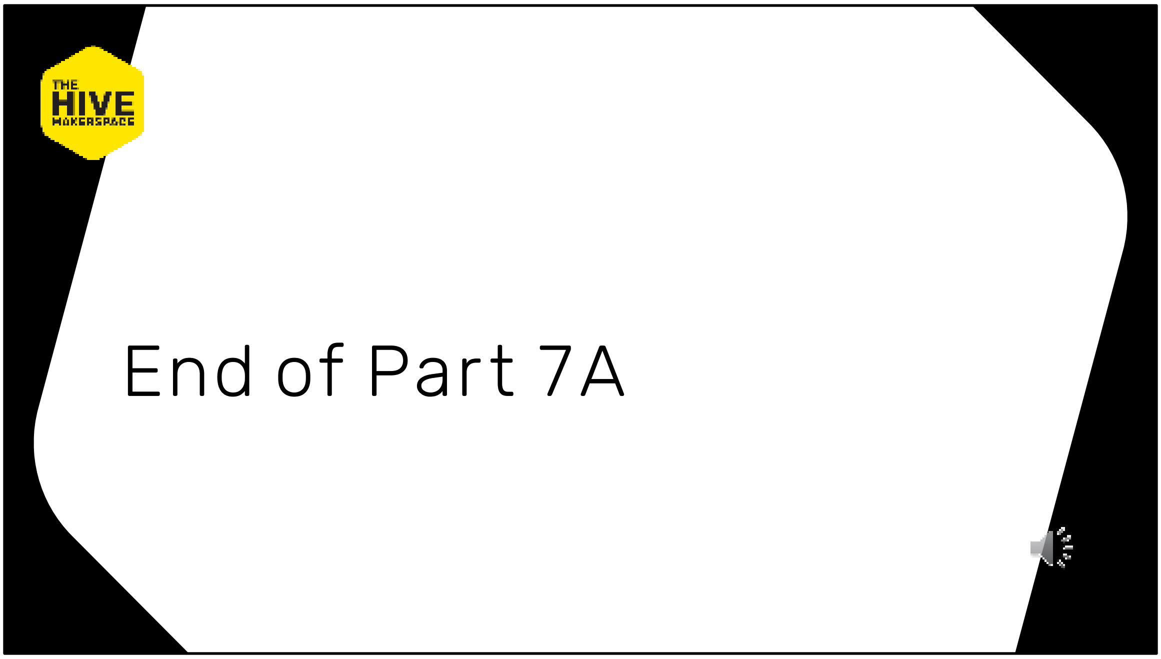
Silly that you can’t filter within a single library alone.

Footprint Library



Footprint Library





And that ends part 7A, in which we covered creating a new footprint library and copying in global components. A PDF of this video is available as well, linked in the description and hosted on The Hive's Wiki.

In the next video, I'll walk through the process of creating non-standard footprints, like the battery holder, from scratch in KiCAD, and then importing the footprint from the internet instead.

See you then!