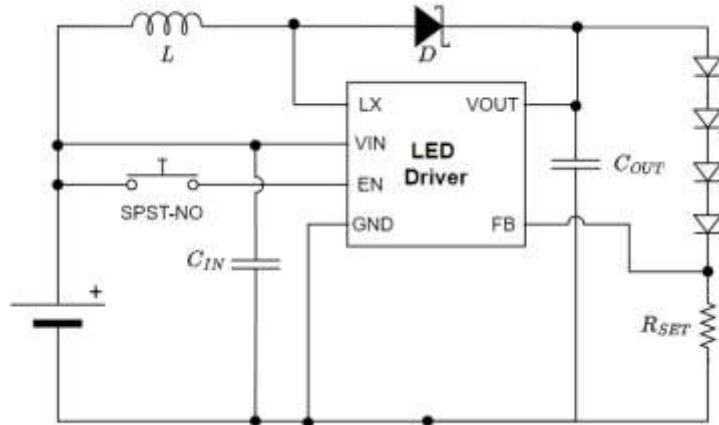


Hi, and welcome to part 4a of The Hive's PCB Design With KiCAD series. My name is Ben, and I'll be your guide today. Part 4 as a whole will cover the entirety of the schematic creation. The segment, part 4C, will cover wiring the symbols together.

As with previous videos, it's recommended that you follow along and pause the playback frequently.



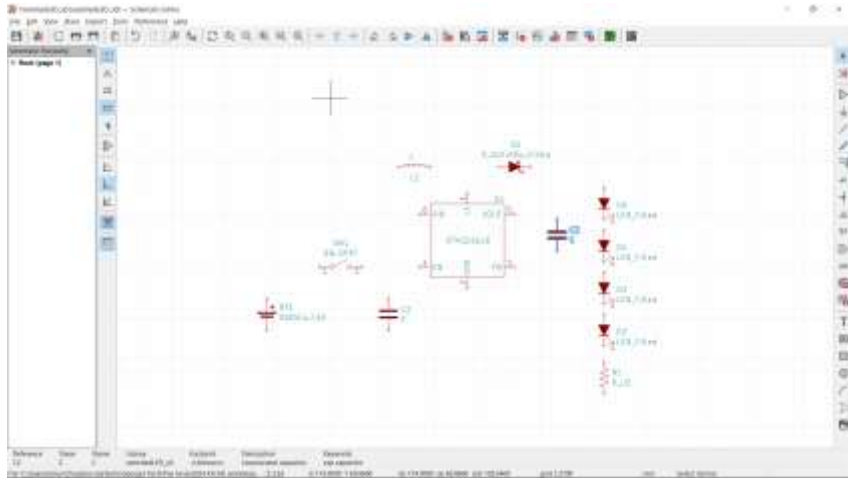
Circuit Reminder



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



Schematic Reminder



And this is a reminder of the schematic as it stood at the end of part 4B. All the components were down and arranged to look like the schematic in the previous slide (and like the one in the datasheet). If you've forgotten anything, I suggest you at least skim through that video (or the associated PDF).



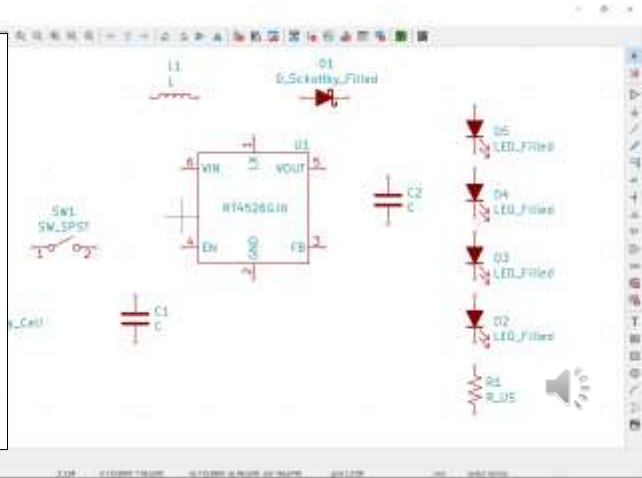
Wires and Labels and Nets – Oh My!

Components in a schematic can be connected either directly or indirectly.

Directly means connecting two pins with a *wire*.

Indirectly means using *labels* to name the nodes.

Any pin attached to nodes that have the name labeled name are electrically connected through a *net* of that name.





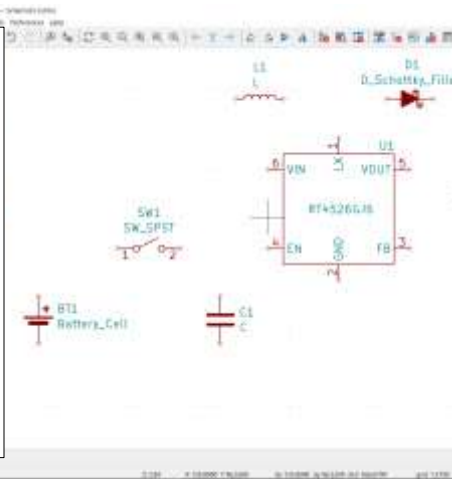
Wires and Labels and Nets – Oh My!

We'll start with wires.

Wires are created in one of four ways:

1. Click (and release) on a symbol's pin
2. Tap "W" to start a wire at the cursor's current location.
3. Click the "Wire" icon on the right (fifth from the top) to enter "Wiring" mode, then click to start a wire.
4. Click "Wire" in the "Place" menu.

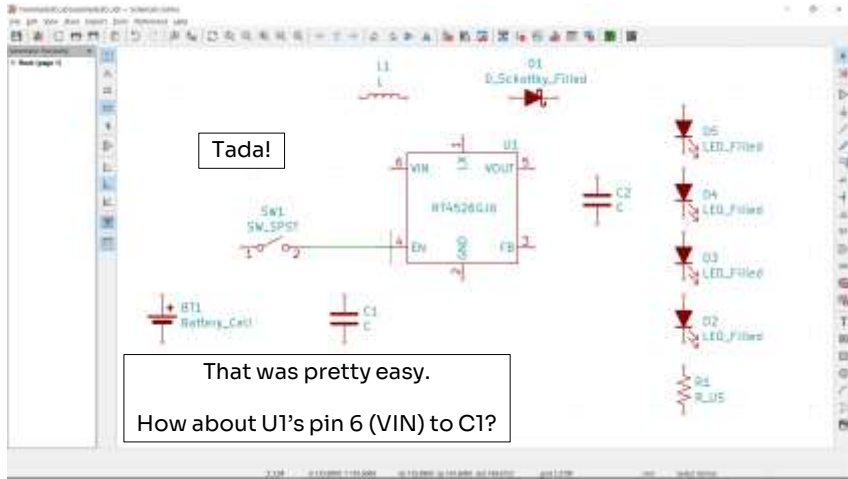
Let's connect the right terminal of the switch (pin 2) to the IC's EN pin (pin 4).



You might pause the video before continuing.

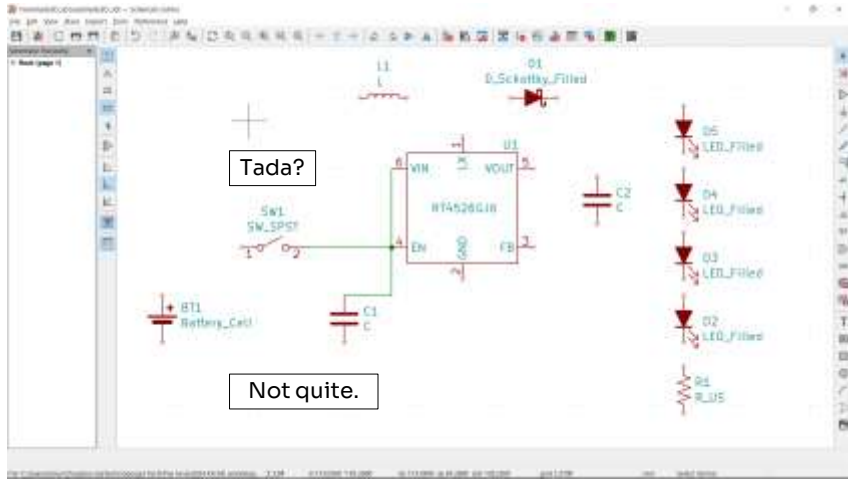


Wires and Labels and Nets – Oh My!





Wires and Labels and Nets – Oh My!



How's this look?

*Not quite.



Wires and Labels and Nets – Oh My!

Two issues, one visual and one electrical.

First, the wire coming off the pin at an angle looks funny. Better to have a bit straight off before bending.

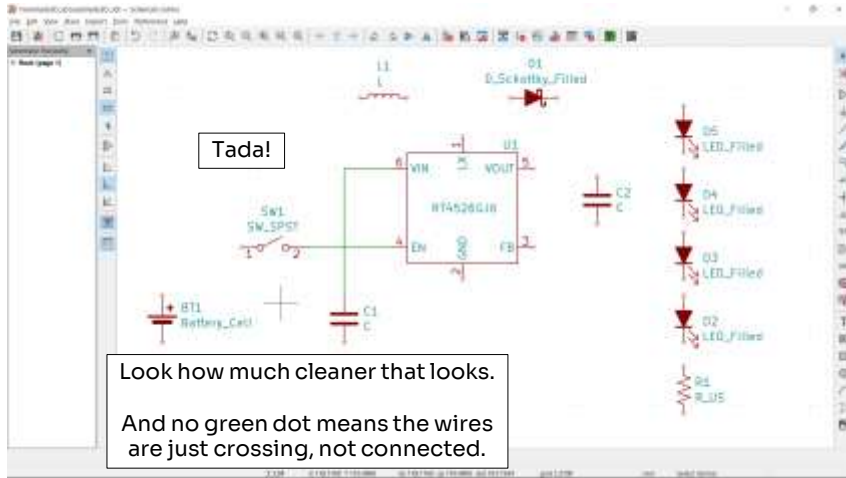
Second, this dot here, called a *junction*, means the crossed wires are electrically connected. No good for this.

Click wire segments and “Delete” to remove them. Or CTRL+Z to undo the segment. Then re-make it.

The screenshot shows a PCB layout with various components like a switch (SW_SPS1), a battery (Battery_Cell), and a capacitor (C1). Red arrows point to specific wire segments and a junction point. The junction point is marked with a small black dot where two wires cross.



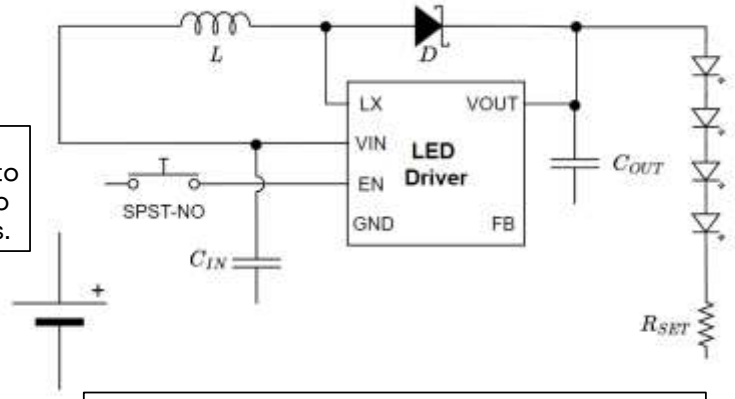
Wires and Labels and Nets – Oh My!





Take a few minutes to wire the passives to the IC.

Hint: You can connect pins to other wires, not just other pins.



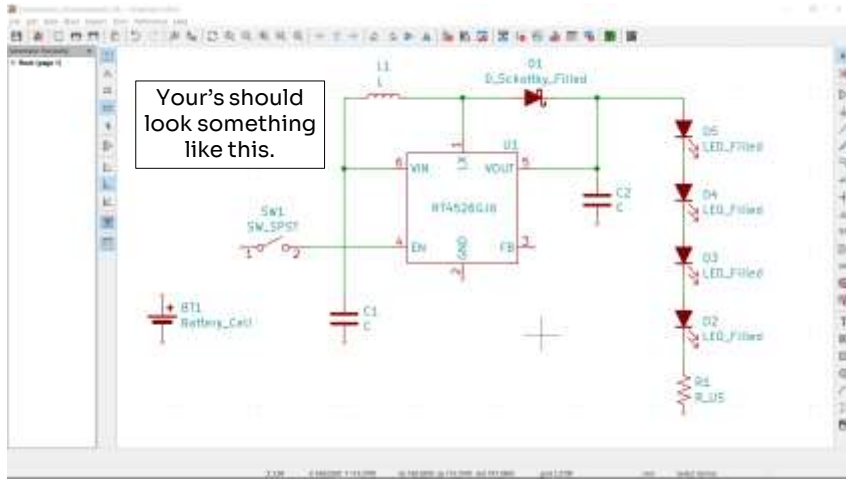
We're going to wire the FB pin, ground (GND), and the battery afterwards, so ignore those connections for now.



I suggest pausing the video here to try this on our own before continuing.



Wires and Labels and Nets – Oh My!



Again, it doesn't /have/ to look like this at all. But as long as the components are connected, you're fine.

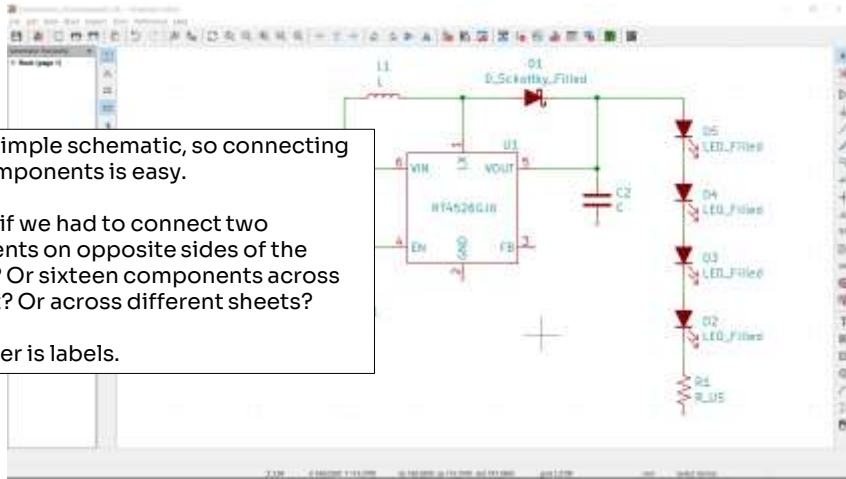


Wires and Labels and Nets – Oh My!

This is a simple schematic, so connecting these components is easy.

But what if we had to connect two components on opposite sides of the diagram? Or sixteen components across the sheet? Or across different sheets?

The answer is labels.





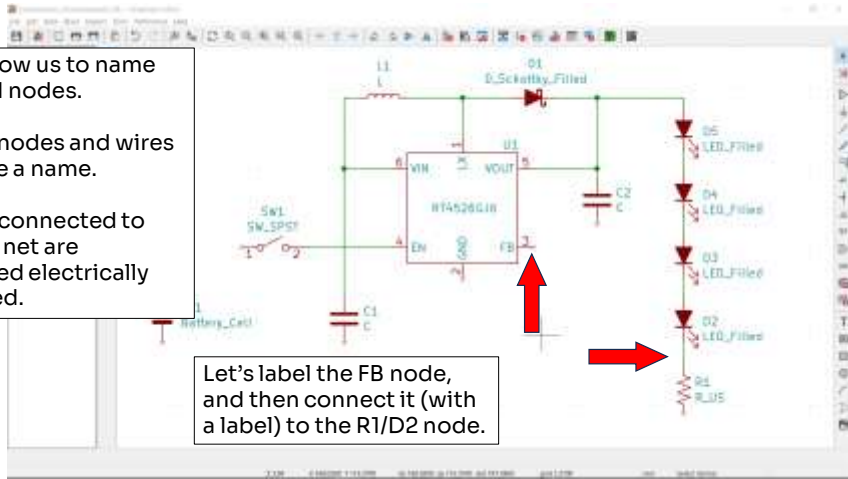
Wires and Labels and Nets – Oh My!

Labels allow us to name wires and nodes.

Nets are nodes and wires that share a name.

Any pins connected to the same net are considered electrically connected.

Let's label the FB node, and then connect it (with a label) to the R1/D2 node.

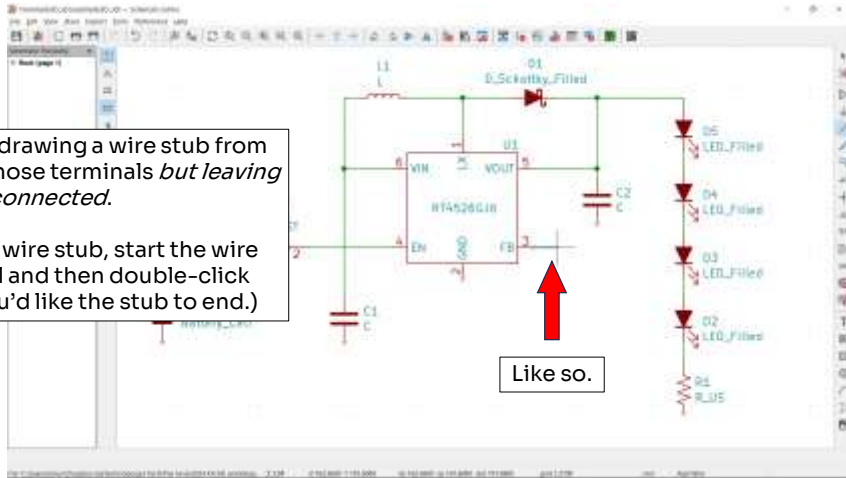




Wires and Labels and Nets – Oh My!

I start by drawing a wire stub from each of those terminals *but leaving them unconnected*.

(To end a wire stub, start the wire as normal and then double-click where you'd like the stub to end.)



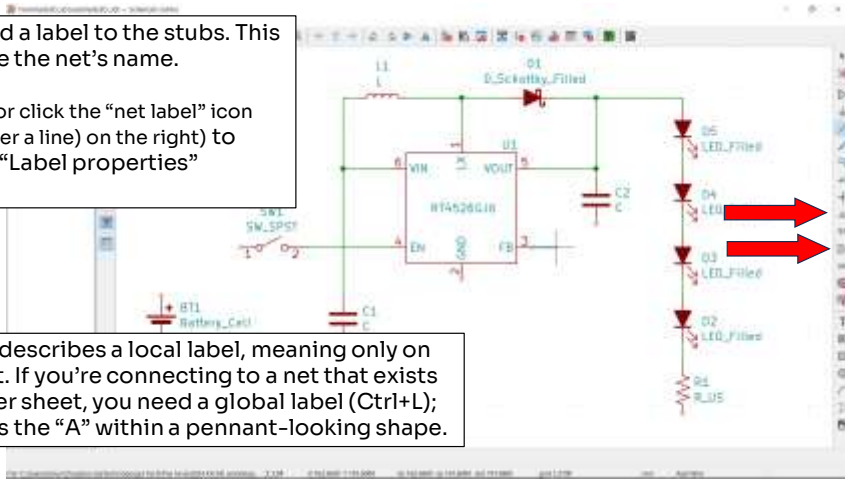


Wires and Labels and Nets – Oh My!

Next, I add a label to the stubs. This will define the net's name.

Tap “L” (or click the “net label” icon (the “A” over a line) on the right) to open the “Label properties” window.

Note: “L” describes a local label, meaning only on this sheet. If you’re connecting to a net that exists on another sheet, you need a global label (Ctrl+L); the icon is the “A” within a pennant-looking shape.



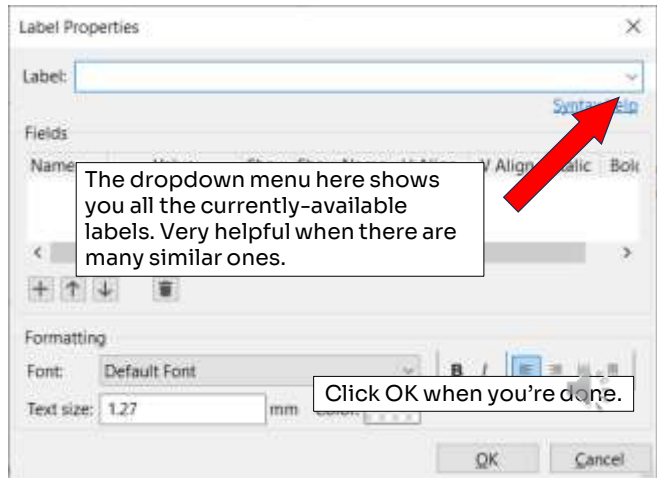


Wires and Labels and Nets – Oh My!

Type the name of the connecting net into the “Label” field and click “OK”.

The restrictions on names center around some symbols, and some globally-defined names like “gnd”.

(You can get fancy if you want here, but boring labels are fine, too.)



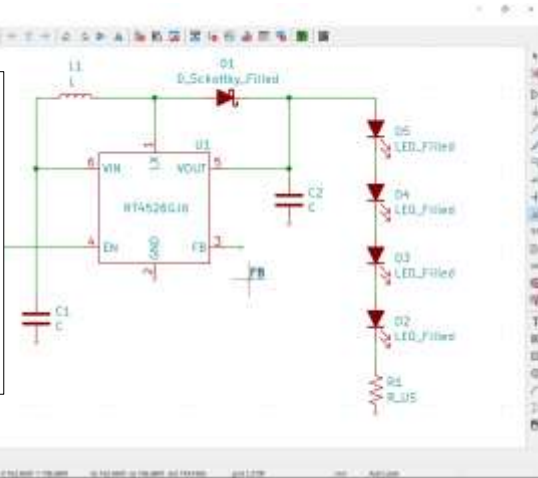


Wires and Labels and Nets – Oh My!

I named my net “FB” for “feedback”, which is fine for this contrived situation, but pretty bad for a larger schematic where there might be many feedback signals.

Anyway, left-click on the wire stub you’re naming to assign the name to the wire. Anywhere on the wire is fine, no need to put it at the end.

Then click “Esc” to exit “label”-mode.



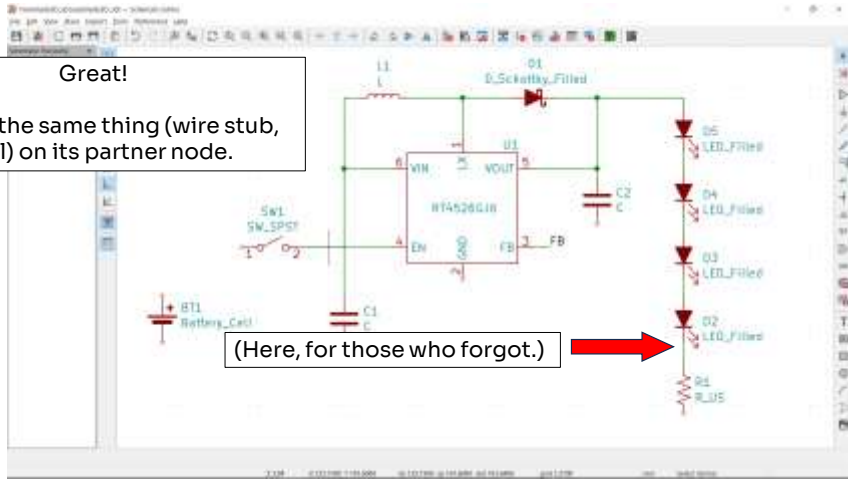


Wires and Labels and Nets – Oh My!

Great!

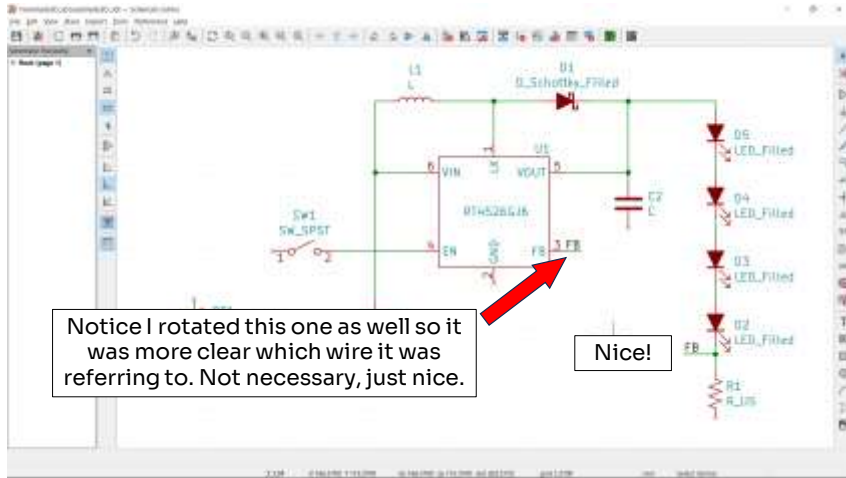
Now do the same thing (wire stub, label) on its partner node.

(Here, for those who forgot.)





Wires and Labels and Nets – Oh My!



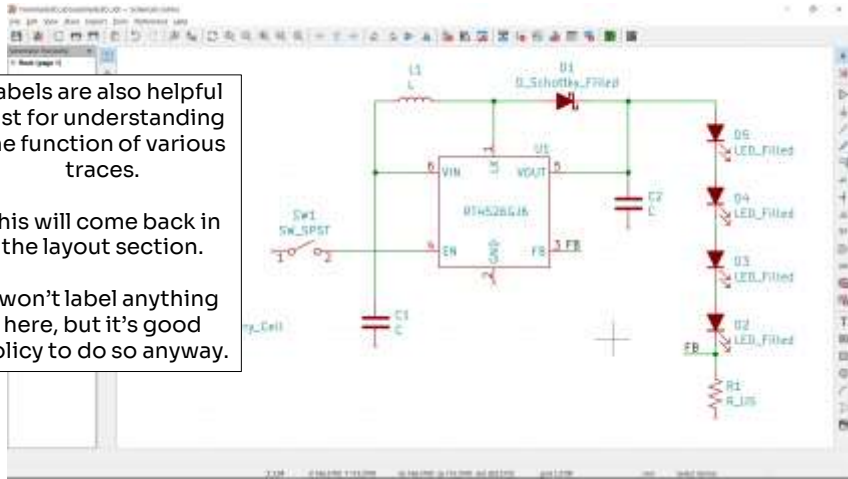


Wires and Labels and Nets – Oh My!

Labels are also helpful just for understanding the function of various traces.

This will come back in the layout section.

I won't label anything here, but it's good policy to do so anyway.



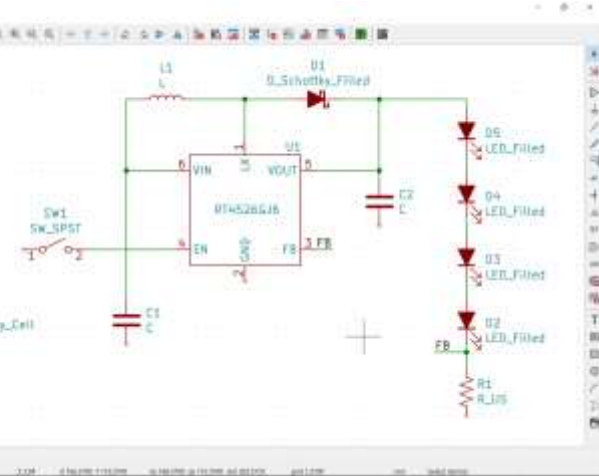


Wires and Labels and Nets – Oh My!

There are special nets for power and ground connections as well.

These come with their own symbols, meaning anything the symbol is connected to is automatically connected to the net of that name.

Exercise caution here! There is some subtlety with these that I'm not covering that can cause ERC failures or mistakes.

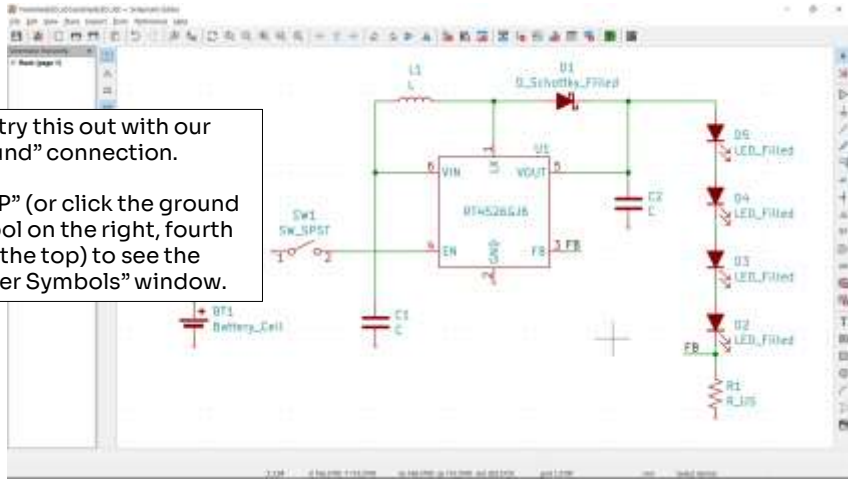




Wires and Labels and Nets – Oh My!

Let's try this out with our "ground" connection.

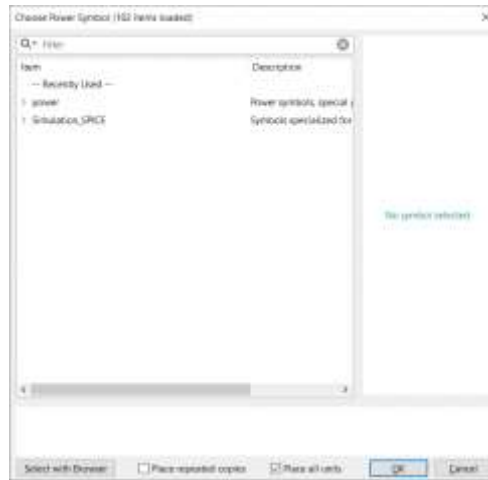
Tap "P" (or click the ground symbol on the right, fourth from the top) to see the "Power Symbols" window.





Wires and Labels and Nets – Oh My!

Basically the same as the
“Add Symbol” window.



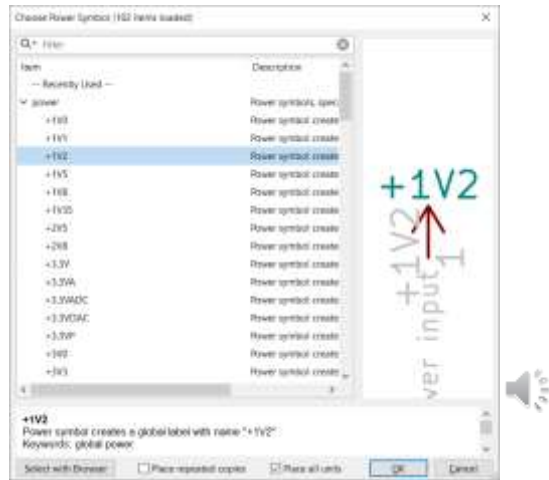


Wires and Labels and Nets – Oh My!

Opening the “power” library shows a bunch of different symbols for different voltages. Most look like this.

Note that these will also create a *global net* with the same name as the symbol, e.g. “+1V2”.

This is one of those label naming restrictions – errors will be thrown if you use these names for your other nets, or confusion will reign.



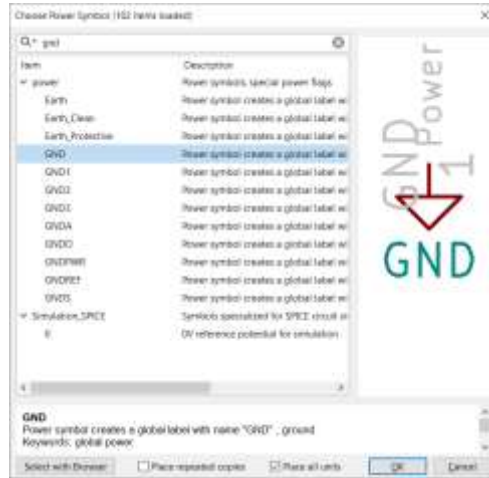


Wires and Labels and Nets – Oh My!

If we filter by “gnd” or “ground”, we’ll see a number of different ground net/symbol options.

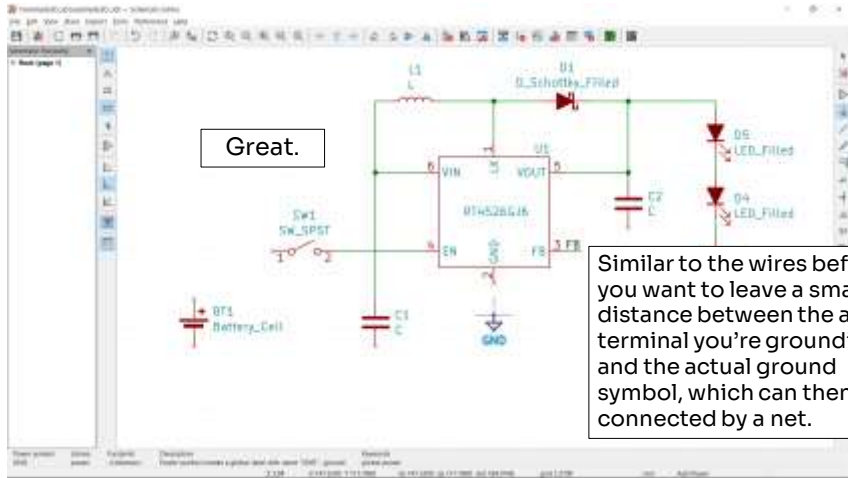
For this schematic, with only a single common return path, the regular GND is what we want.

Place it below (but not connected to) the battery’s negative pin.





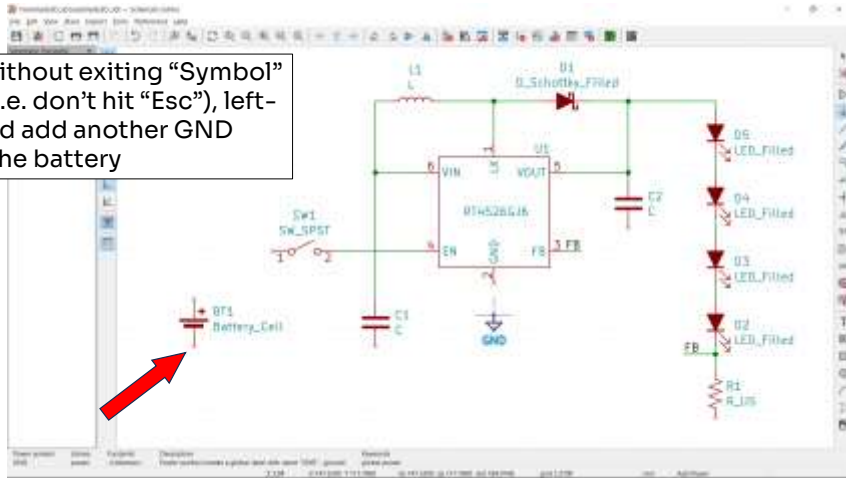
Wires and Labels and Nets – Oh My!





Wires and Labels and Nets – Oh My!

Then, without exiting “Symbol” mode (i.e. don’t hit “Esc”), left-click and add another GND below the battery

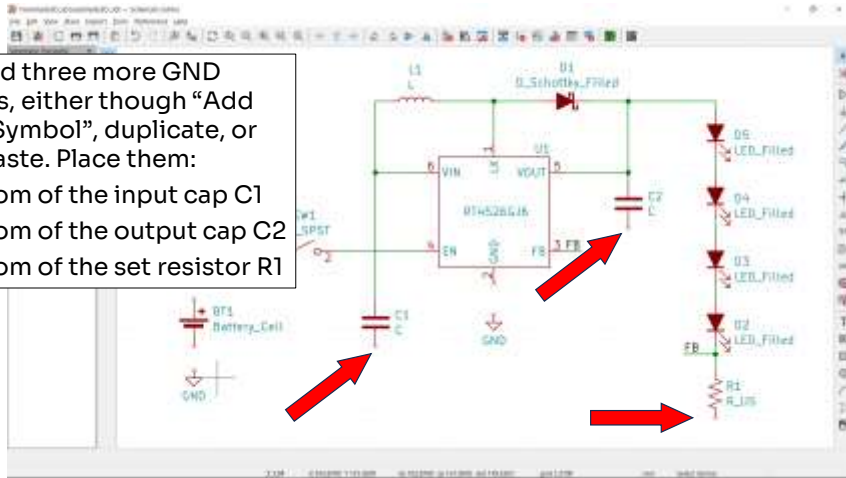




Wires and Labels and Nets – Oh My!

Now add three more GND symbols, either through “Add Power Symbol”, duplicate, or copy/paste. Place them:

- Bottom of the input cap C1
- Bottom of the output cap C2
- Bottom of the set resistor R1

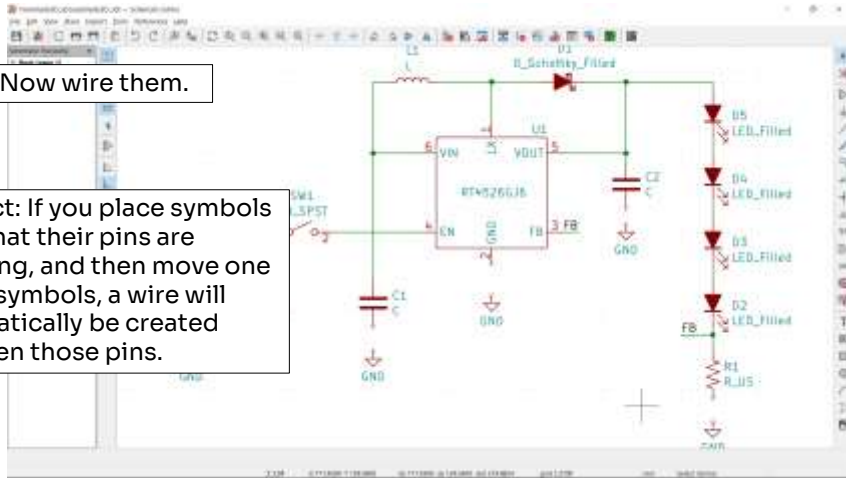




Wires and Labels and Nets – Oh My!

Great! Now wire them.

Fun fact: If you place symbols such that their pins are touching, and then move one of the symbols, a wire will automatically be created between those pins.



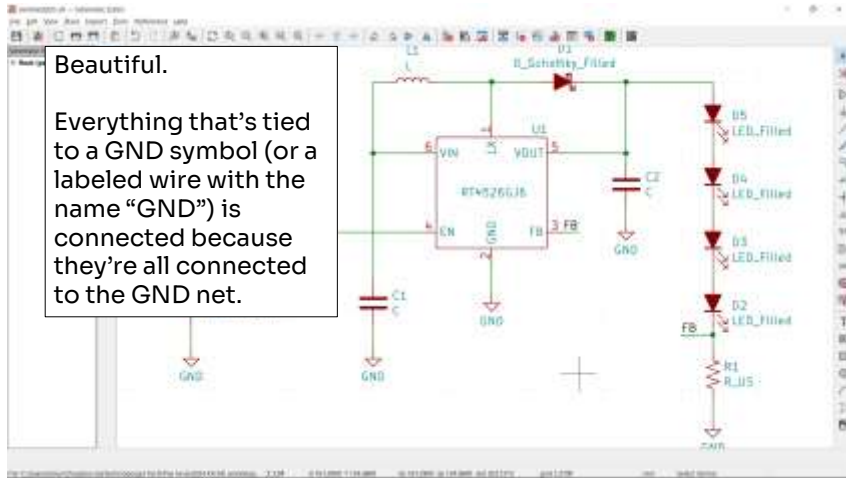
Not necessary to



Wires and Labels and Nets – Oh My!

Beautiful.

Everything that's tied to a GND symbol (or a labeled wire with the name "GND") is connected because they're all connected to the GND net.



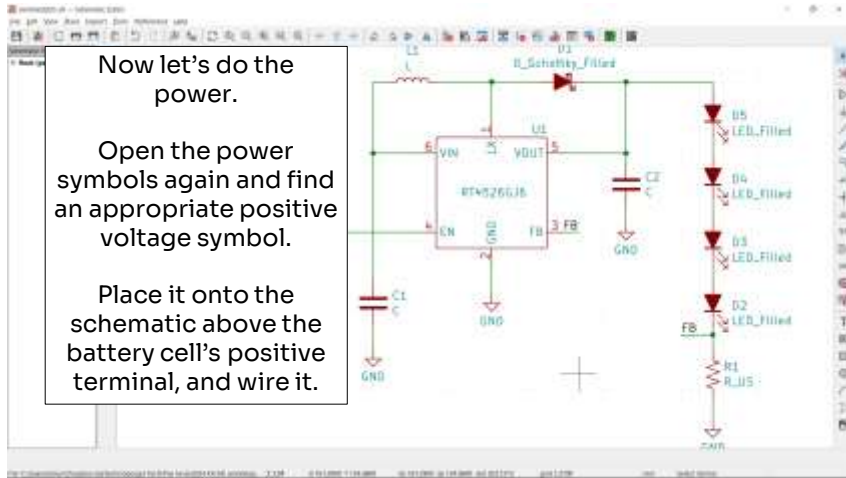


Wires and Labels and Nets – Oh My!

Now let's do the power.

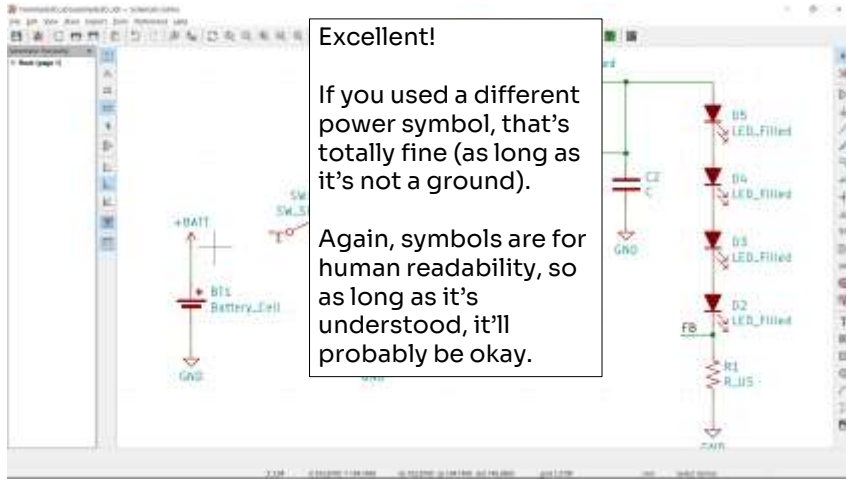
Open the power symbols again and find an appropriate positive voltage symbol.

Place it onto the schematic above the battery cell's positive terminal, and wire it.





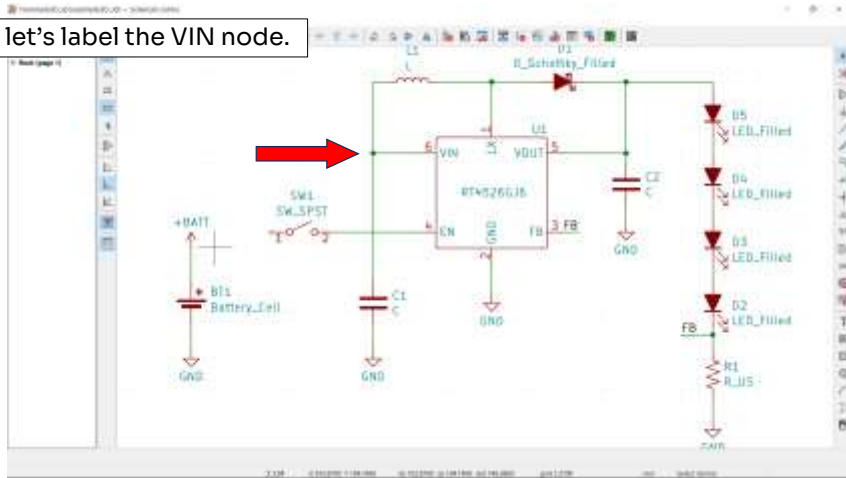
Wires and Labels and Nets – Oh My!





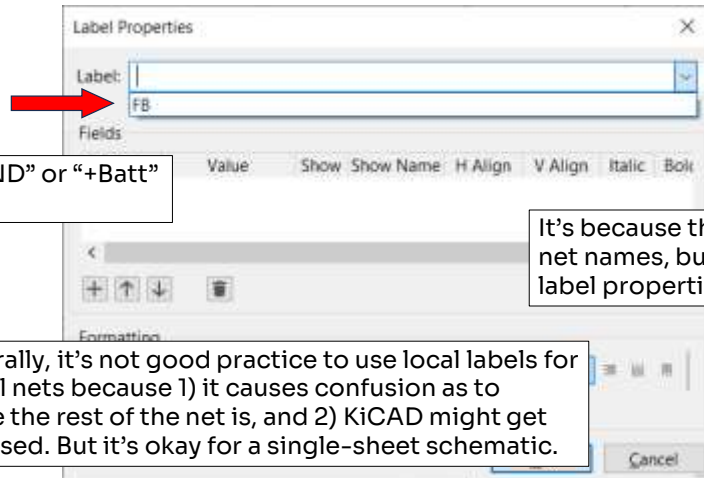
Wires and Labels and Nets – Oh My!

Now let's label the VIN node.





Wires and Labels and Nets – Oh My!



Why isn't "GND" or "+Batt" listed here?

It's because those are *global* net names, but this is the *local* label properties window.

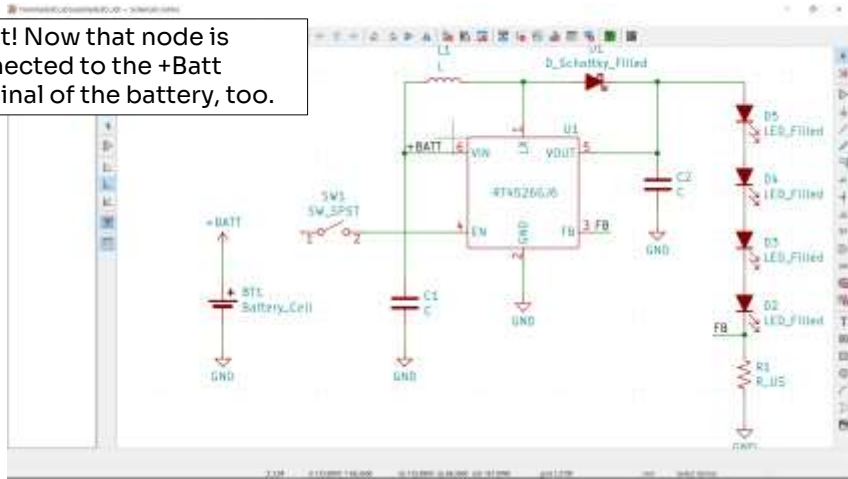
Generally, it's not good practice to use local labels for global nets because 1) it causes confusion as to where the rest of the net is, and 2) KiCAD might get confused. But it's okay for a single-sheet schematic.

You might notice in the label properties window that GND and your power net name aren't listed in the drop down. Why?



Wires and Labels and Nets – Oh My!

Great! Now that node is connected to the +Batt terminal of the battery, too.

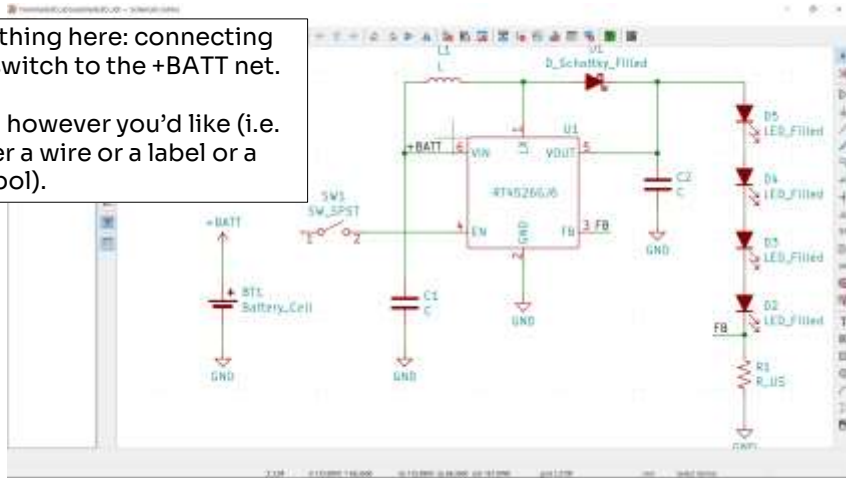




Wires and Labels and Nets – Oh My!

Last thing here: connecting the switch to the +BATT net.

Do it however you'd like (i.e. either a wire or a label or a symbol).

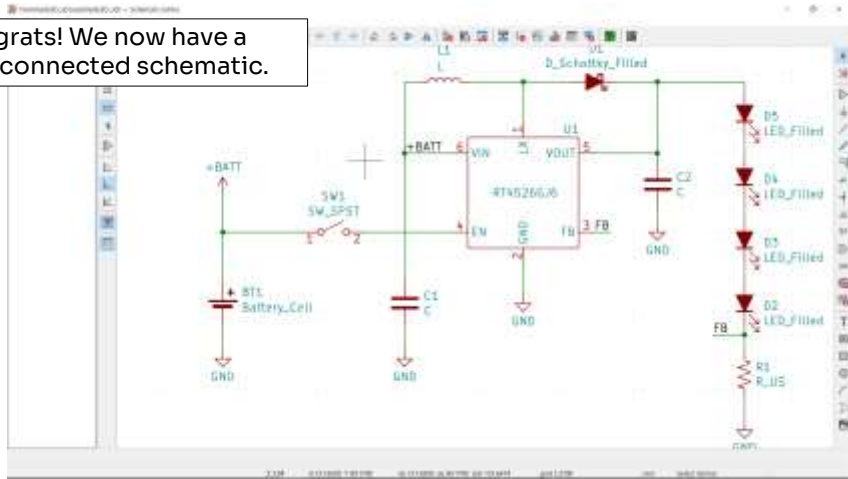


Take a second here to make this connection before continuing.



A fully connected schematic!

Congrats! We now have a fully connected schematic.





End of Part 4C

And with that, we end part 4C of this video series on KiCAD and PCB design in which I covered wiring and nets. A PDF of this video is available as well, linked in the description and hosted on The Hive's Wiki.

In the next part, 4D, we'll look at assigning footprints to the various symbols.

See you there.