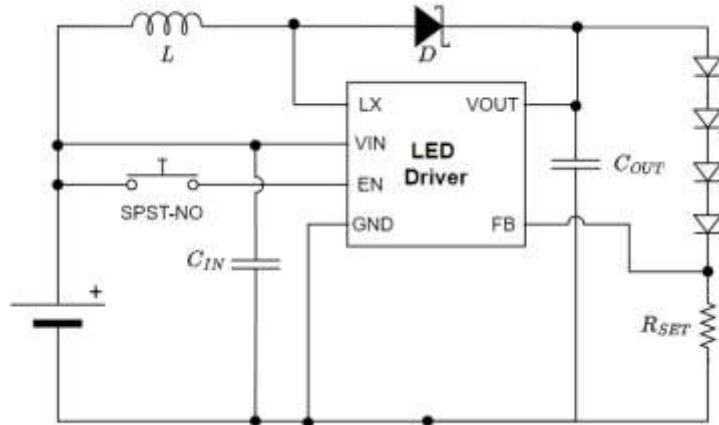


Hi, and welcome to part 4E of The Hive's PCB Design With KiCAD series. My name is Ben, and I'm your host today. Part 4 as a whole has been covering the entirety of the schematic creation, and we'll wrap this portion up with a discussion of the ERC, understanding and dealing with warnings and errors, and some miscellaneous schematic tools you might be interested in being vaguely aware of. Let's get into it.



## 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.



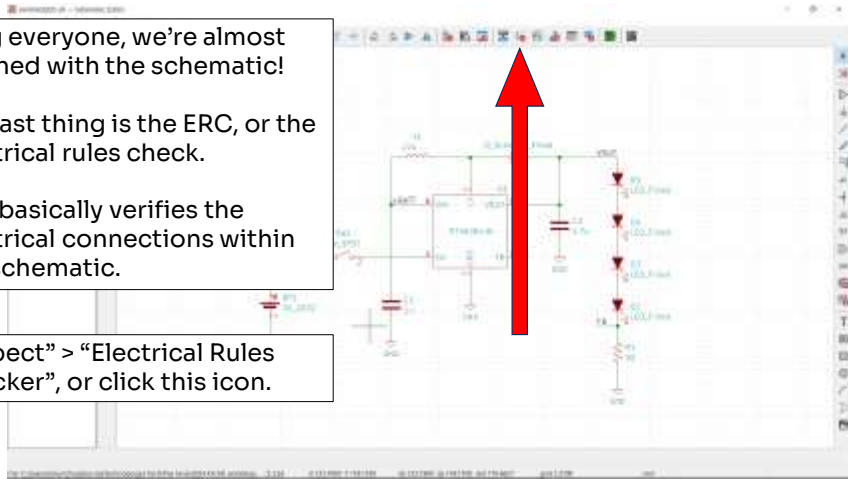
## ERC

Omg everyone, we're almost finished with the schematic!

The last thing is the ERC, or the electrical rules check.

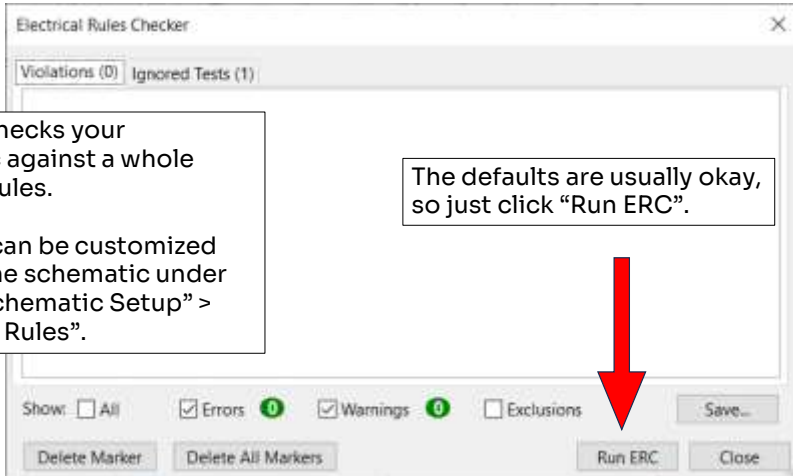
This basically verifies the electrical connections within the schematic.

“Inspect” > “Electrical Rules Checker”, or click this icon.





## ERC



The ERC checks your schematic against a whole bunch of rules.

The rules can be customized through the schematic under “File” > “Schematic Setup” > “Electrical Rules”.

The defaults are usually okay, so just click “Run ERC”.

The ERC should, ideally, have been run a few times throughout the schematic design process because it can be lengthy and iterative, so it's valuable to see if you've made errors early before they pile up and start building on each other.



# ERC

Electrical Rules Checker


Oh no! Two errors??

Violations (2) Ignored Tests (1)

- ▼ **Error: Input Power pin not driven by any Output Power pins**  
Symbol U1 Pin 2 [GND, Power input, Line]
- ▼ **Error: Input Power pin not driven by any Output Power pins**  
Symbol U1 Pin 6 [VIN, Power input, Line]

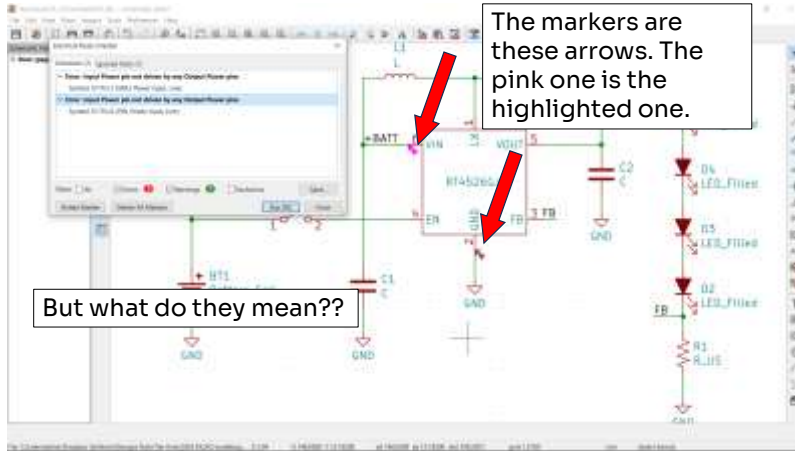
Clicking on them will highlight them in the schematic, which will give you insight into what this might mean.

Show:  All  Errors **2**  Warnings **0**  Exclusions





# ERC





# ERC

So they're pointing at pins, and say that the the input power pins aren't driven by output power pins.

Electrical Rules Checker

Violations (2) Ignored Tests (1)

- ▼ **Error: Input Power pin not driven by any Output Power pins**  
Symbol U1 Pin 2 [GND, Power input, Line]
- ▼ **Error: Input Power pin not driven by any Output Power pins**  
Symbol U1 Pin 6 [VIN, Power input, Line]

Frankly, like a lot of PCB CAD errors, this is pretty inscrutable until you have some experience with it and with the jargon. Google and the forums can be helpful here.

Show:  All  Errors **2**  Warnings **0**  Exclusions





# ERC

What this is saying is that there is a mismatch in pin types.

Electrical Rules Checker

Violations (2) Ignored Tests (1)

- ▼ **Error: Input Power pin not driven by any Output Power pins**  
Symbol U1 Pin 2 [GND, Power input, Line]
- ▼ **Error: Input Power pin not driven by any Output Power pins**  
Symbol U1 Pin 6 [VIN, Power input, Line]

Remember way back when we were making the symbol for the LED driver IC in video 4B? When we were creating the pins?

Show:  All  Errors **2**  Warnings **0**  Exclusions



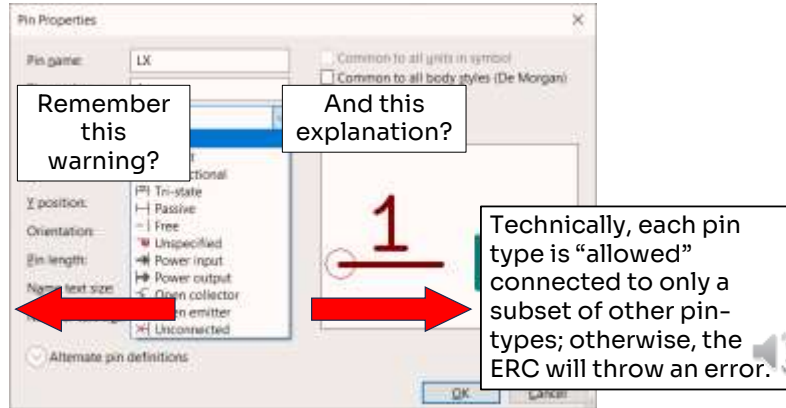




(For those who don't remember)

If you're not sure what it is, you can either set it to "Bidirectional" or "Unspecified" or "Passive".

The type won't cause the design to fail, but it might cause you headaches later with the ERC.



\*Remember this warning?

\*And this explanation? This probably didn't make any sense back then, but this is exactly what we're seeing.



## ERC

So I set the GND and VIN pins of the IC to "Power input" types because, well, they take in power, right?

Electrical Rules Checker

Violations (2) Ignored Tests (1)

▼ Error: Input Power pin not driven by any Output Power pins  
Symbol U1 Pin 2 [GND, Power input, Line]

▼ Error: Input Power pin not driven by any Output Power pins  
Symbol U1 Pin 6 [VIN, Power input, Line]

Unfortunately, the battery symbol's pins are actually set to "Passive". Which is not an allowable pin type to connect with a "Power input".

Oops.

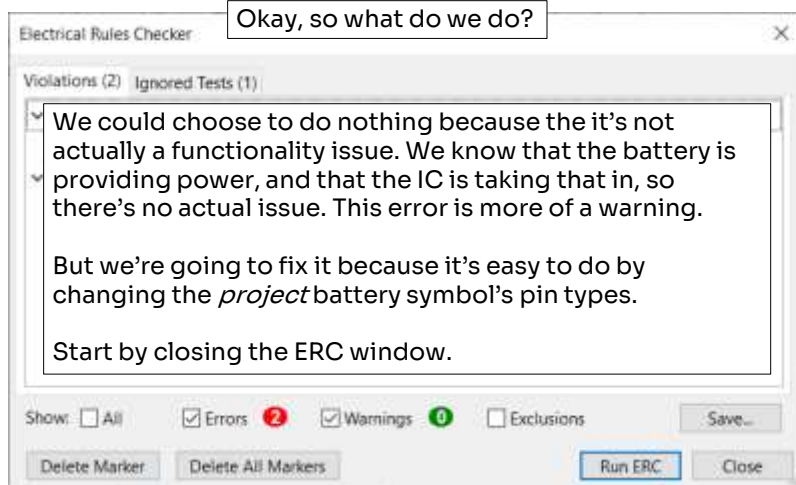
BT1  
Battery\_Cell

Passive Passive

How would you have known this? You wouldn't, necessarily, and I certainly didn't, but the error suggests that whatever is on the other side of the problematic VIN and GND pins (which is what the markers point at) is not correct. So I went and looked at the battery symbol and at its pin types to discover this mismatch.



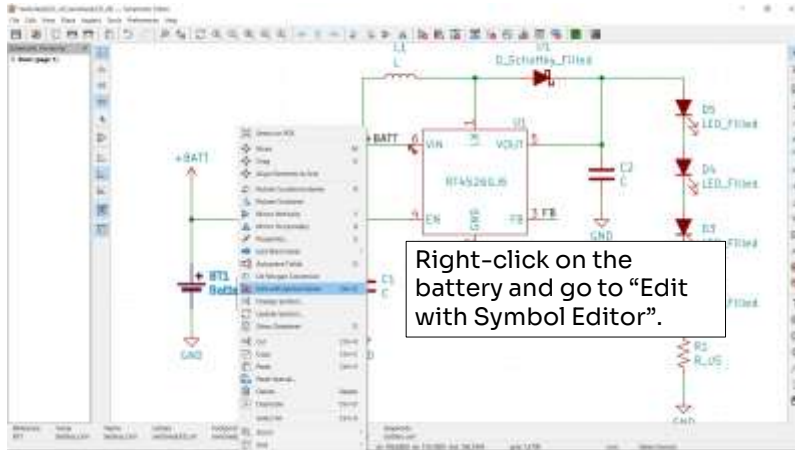
## ERC



By “project” here, I mean that we’re just editing the project’s version of the footprint, not the global part model. Therefore, we’re not interfering with any other battery symbols in other schematics, though it will update other batteries within this schematic.



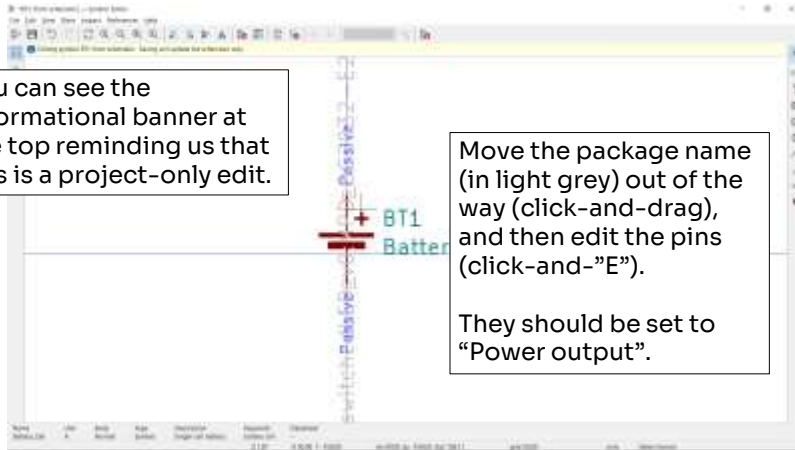
# ERC





# ERC

You can see the informational banner at the top reminding us that this is a project-only edit.



Move the package name (in light grey) out of the way (click-and-drag), and then edit the pins (click-and-“E”). They should be set to “Power output”.





# ERC



Not necessary to



# ERC



Number	Base Name	Alternate Assignment	Electrical Type	Graphic Style
1			Power output	Line
2			Power output	Line

You can check that it updated by going to the symbol's properties within the schematic, and look under the "Alternate Pin Assignments" tab at the top.

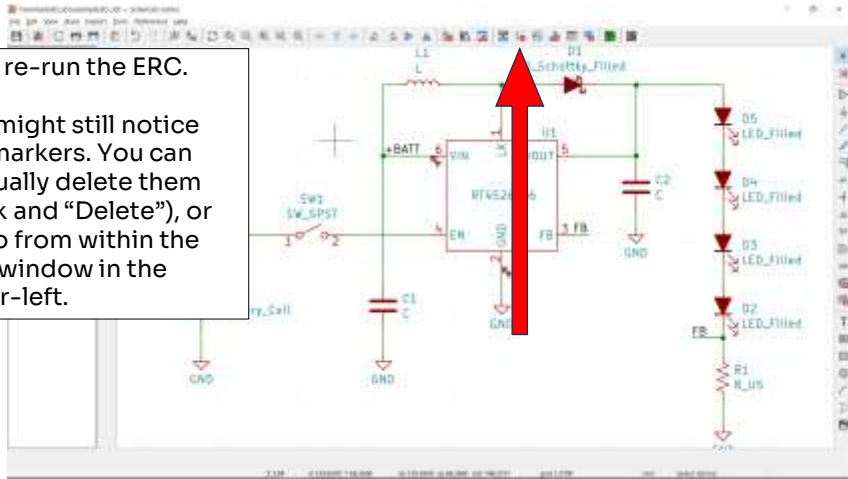




# ERC

Now re-run the ERC.

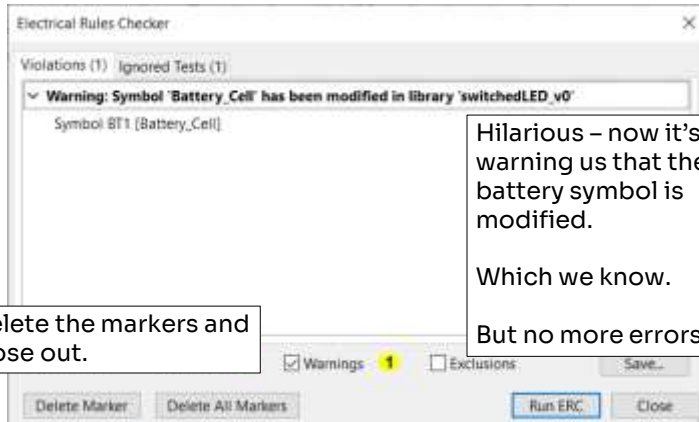
You might still notice the markers. You can manually delete them (click and "Delete"), or do so from within the ERC window in the lower-left.







# ERC



Delete the markers and close out.

Hilarious – now it's warning us that the battery symbol is modified.

Which we know.

But no more errors!

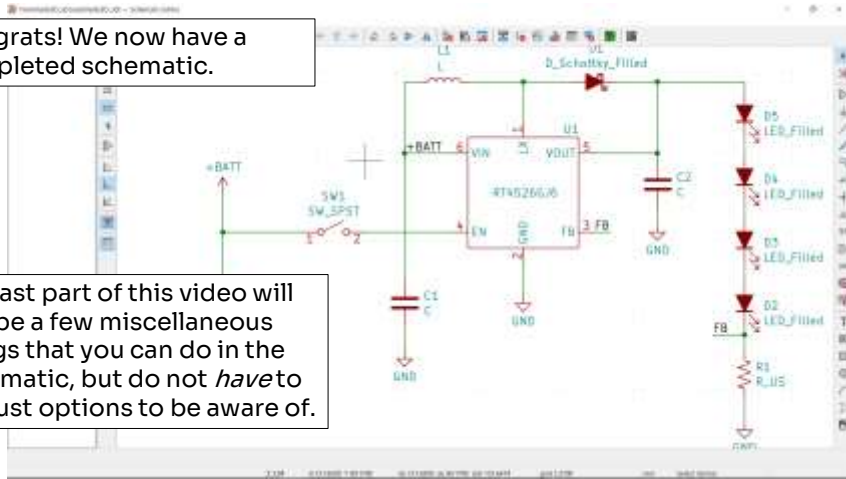




## Final schematic and misc.

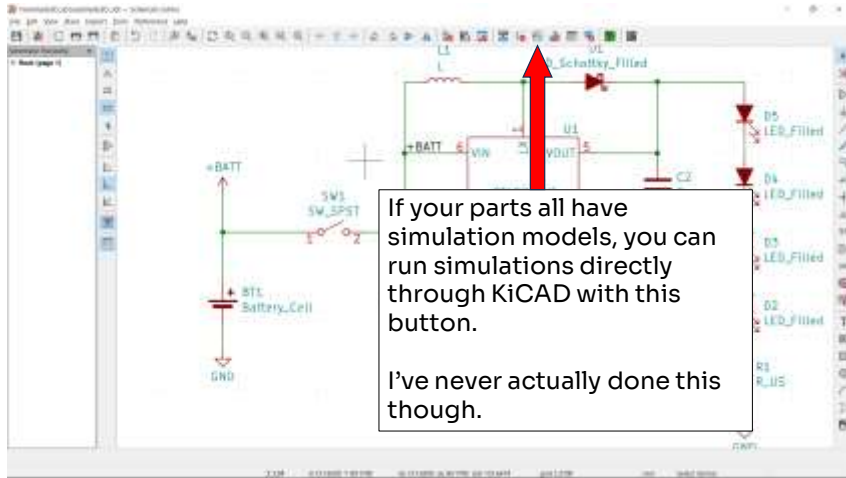
Congrats! We now have a completed schematic.

The last part of this video will just be a few miscellaneous things that you can do in the schematic, but do not *have* to do; just options to be aware of.



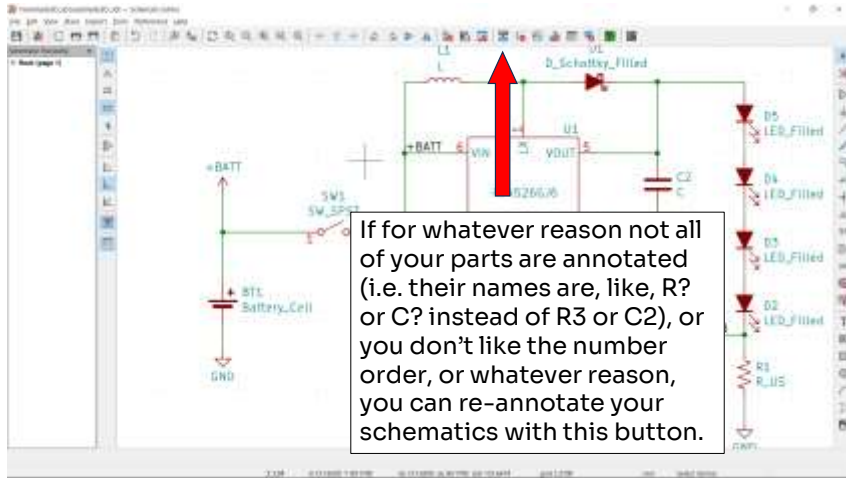


## Final schematic and misc.



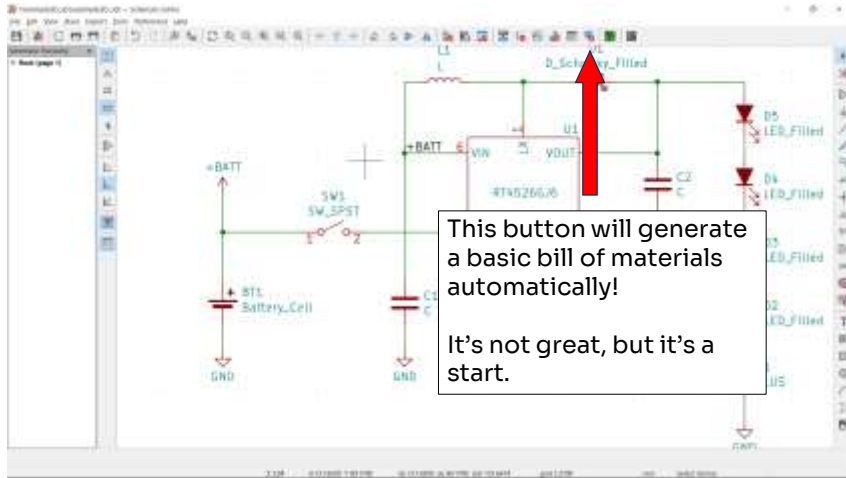


## Final schematic and misc.



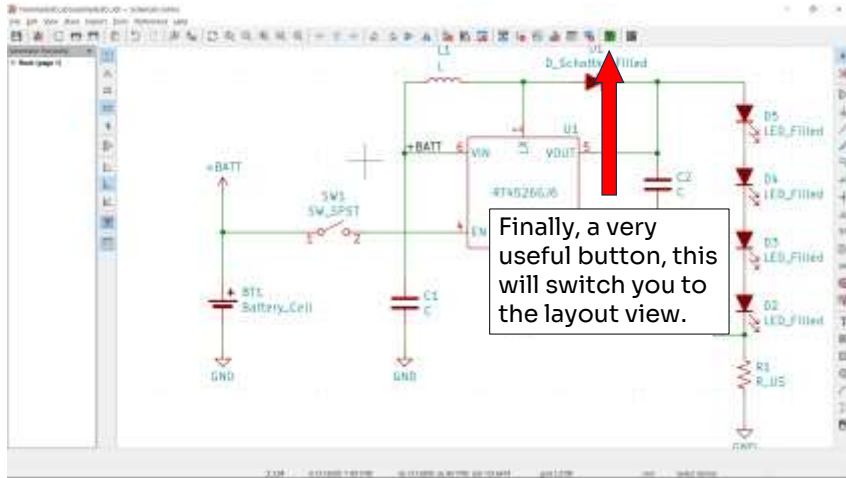


## Final schematic and misc.





## Final schematic and misc.





## End of Part 4E

And with that, we've completed part 4E, in which we covered ERC and some additional miscellany. A PDF of this video is available as well, linked in the description and hosted on The Hive's Wiki.

This also brings us to the end of the schematic capture portion of the design. Congratulations!

In the next video in our PCB Design with KiCAD series, part 5A, we'll move over to the layout, known in KiCAD as the PCB view, and begin with setting up some defaults and the design rules.