KiCad
workshop
for

HAN_UNIVERSITY
OF APPLIED SCIENCES

Workshop KiCad7

ADVANCED WORKSHOP

WRITTEN BY: CASPER R. TAK

Contents

1. Workshop Version management

Version	Date	Changes/notes
V0.1	05-12-2023	Setup of document
V0.2	30-12-2023	Addition of step 1, step 2 and step 3.
V0.3	31-12-2023	Addition of: Step 4: Importing symbols, Step 5: Integrating the new button, Step 6: Redesigning the board outline and adding a hole, Step 7: Adding text and pictures to the PCB Step 8: Adding 3D-models Added Bonus Chapter: Add-ons
V0.4	28-01-2023	Made chapter 4&5 optional. Added tips on grid settings. Fixed wiring error. Added some plugin information and extra's in chapter 3 and 3.1.

Workshop Summary: What will you learn in this workshop?

Advanced Workshop Summary: Elevate Your KiCad Skills

Are you ready to take your KiCad proficiency to the next level? Join our Advanced Workshop where we delve into advanced techniques and features in both the schematic and PCB editors. By the end of this workshop, you'll be equipped with the skills to add personal touches to your designs and optimize your workflow.

1 In Schematic Editor:

1. **Personalization:**

   - *Add Your Name:* Learn how to add a personal touch by including your name on the schematic paper.

   - *Text Annotations:* Utilize the text feature to include useful information on the schematic, enhancing clarity and documentation.

2. **Advanced Schematic Organization:**

   - *Hierarchical Sheets:* Master the art of creating hierarchical sheets, allowing for a more organized and modular approach to complex designs.

3. **Library Management:**

   - *Component Imports:* Explore the process of importing components from external libraries, expanding your component options and design flexibility.

2 In PCB Editor:

1. **Aesthetics and Customization:**

   - *Rounded PCB Corners:* Learn how to round off the corners of your PCB, adding a professional and polished look to your designs.

   - *Silkscreen Additions:* Add a .bmp logo file to the PCB silkscreen and incorporate text with details such as name, date, version, project name, and designer information.

   - *Symbolic Additions:* Enhance the visual representation with useful symbols like polarity indicators (+ and - symbols).

2. **Advanced PCB Features:**

   - *Custom Holes:* Create holes in the PCB for unique features, such as keychain attachments, expanding the range of applications for your designs.

   - *Add* .BMP files to your board design

   - *3D Models:* Add missing 3D models to enhance the 3D viewer experience, ensuring a comprehensive representation of your PCB design.

**3 Extra Enhancements:**

1. **KiCad Add-ons:**

- *Downloadable Add-ons:* Discover and leverage KiCad downloadable add-ons to enhance your software capabilities and customize your KiCad experience according to your project requirements.

- *Trying out the smooth lines/round tracks plug*-in.

Embark on this advanced workshop to elevate your KiCad expertise, infusing your designs with personalized elements and taking advantage of advanced features for a more professional and customized PCB design experience. Whether you are a seasoned user or looking to expand your knowledge, this workshop is tailored to refine your skills and boost your design capabilities.

2. The KeychainFlashLight Advanced Workshop

Step 1: Add information to the schematic page

1. Open the *KeychainFlashLightAdvanced.kicad_pro* file in KiCad

2. Open the schematic file.

3. Click on the paper icon shown below.

4. **ıe 1)**
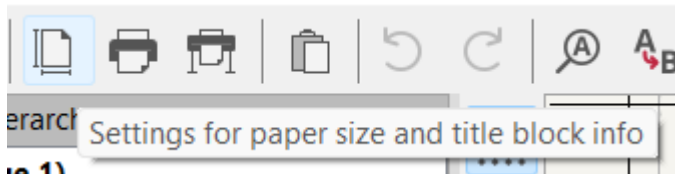
5. Add some information to the page (see example below). You may put a blue tik in the boxes if you want to export the same information to other sheets.
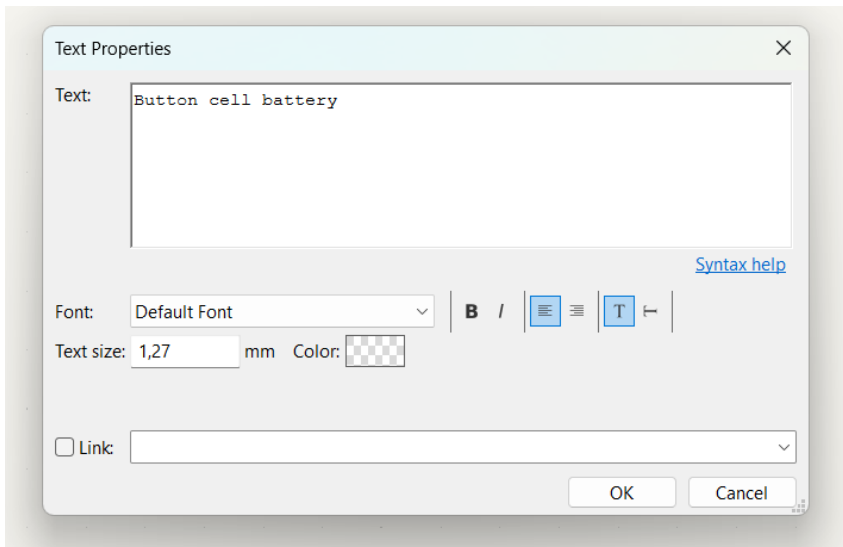

6.
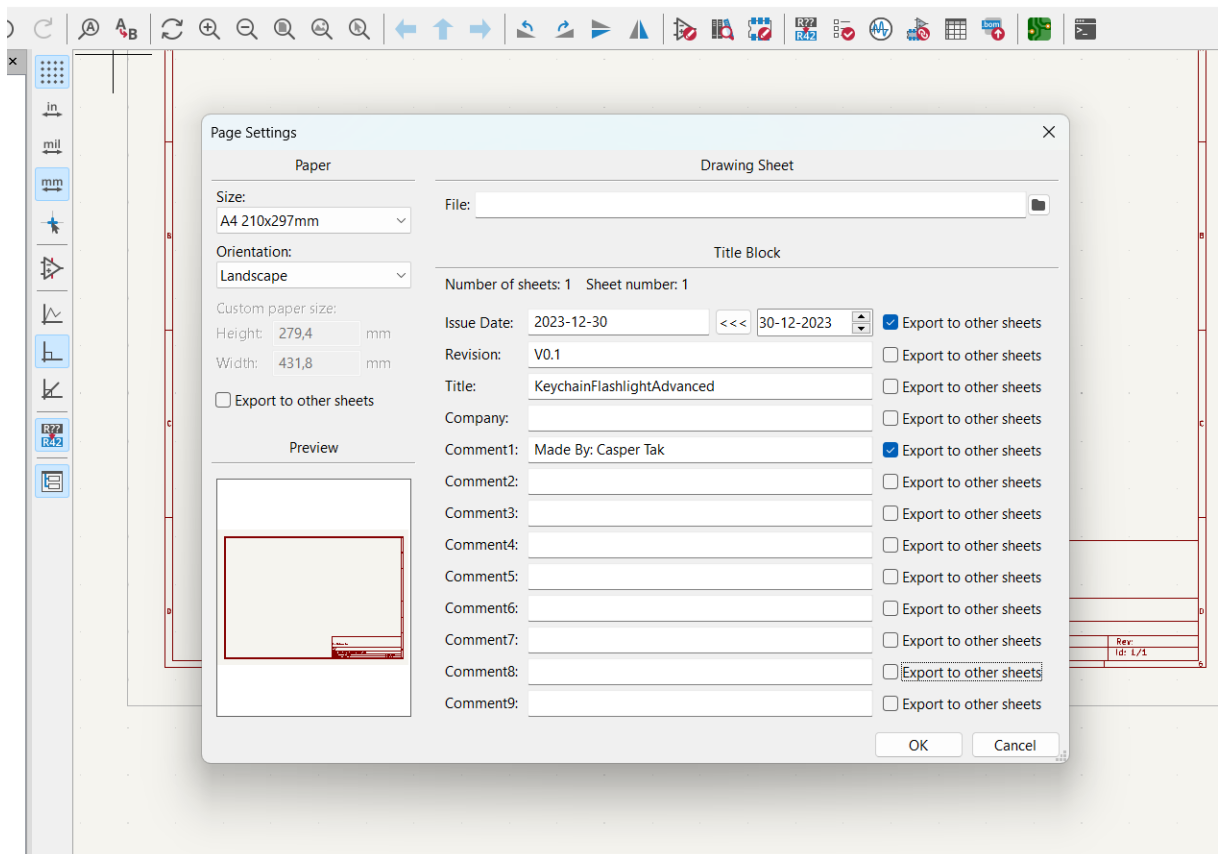
7. You should now see the information added to the red "box" in the bottom right corner.

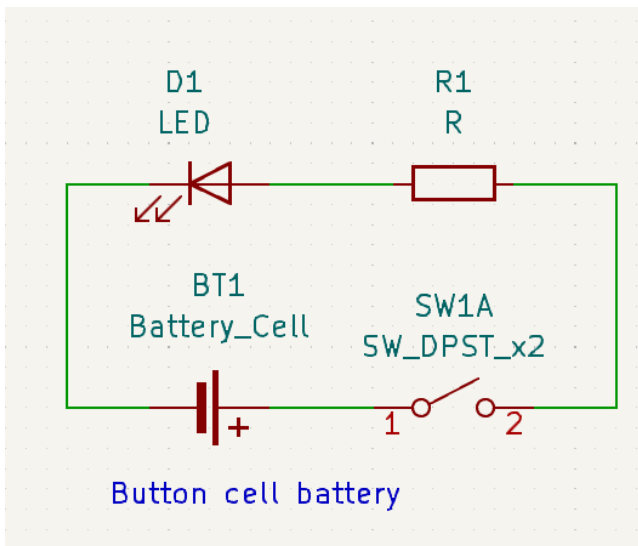## Step 2: Add informative text to the schematic sheet

1. On the schematic sheet, select the flowing icon in the toolbar on the right.

2.

Add text (T)
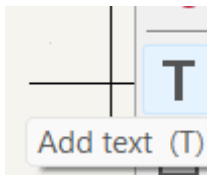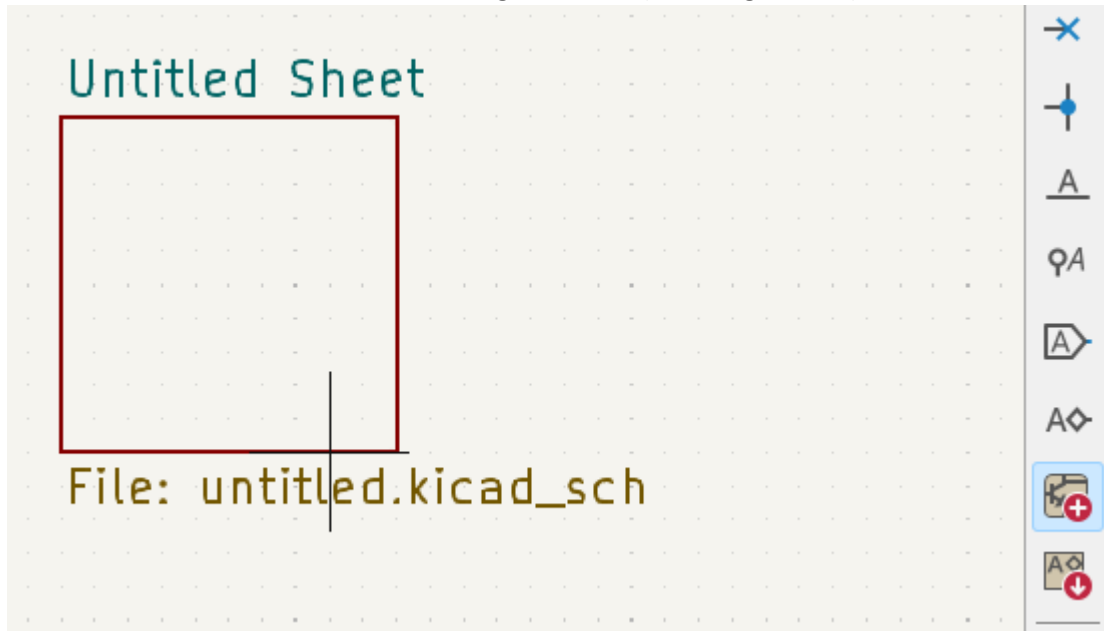
3. A window will open, fill in the following text:

4.

5. When pressing "ok" you will see that the text will follow your cursor. Place the text near the BT1 icon. You can try to experiment with the settings you see in this window.
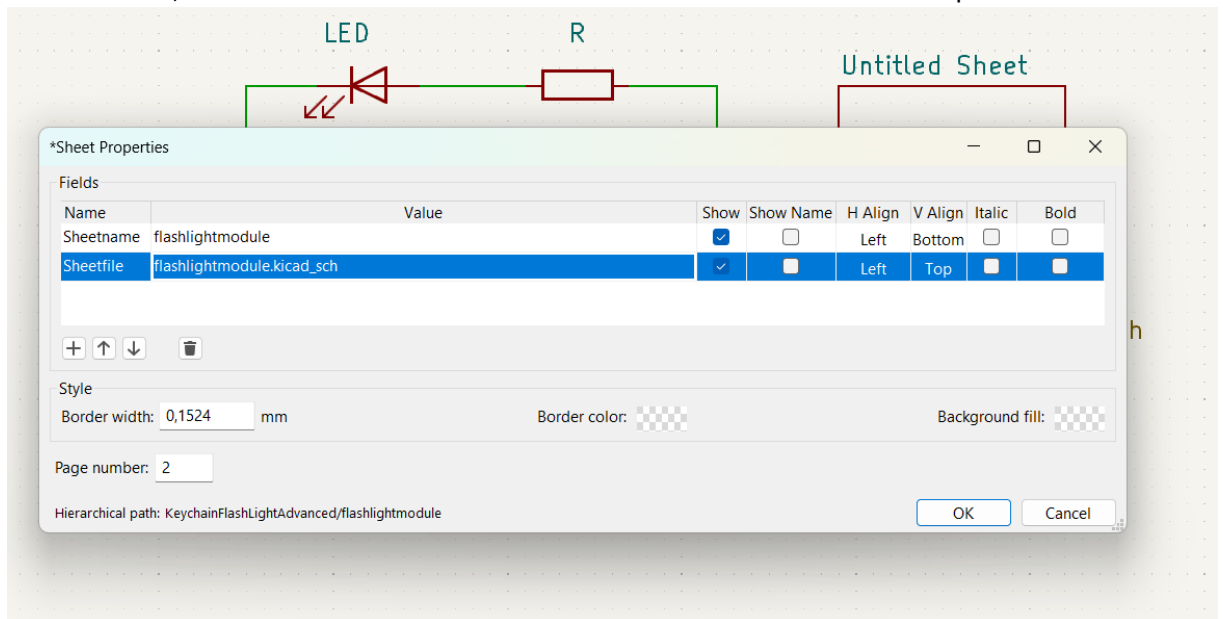

6.

Step 3: Adding a hierarchical sheet

1. Press the hierarchical sheet icon on the right tool bar (see image below)

2.
3. By clicking and dragging the mouse, you will create a box shape, when clicking the mouse for a second time, it will show a new window. Fill in the window as seen below and press "ok".


4.
5. Your hierarchical sheet is now made, you may now enter it by double clicking inside the box with your mouse.
6. When inside the sheet, please exit it again by pressing the arrow as seen below.
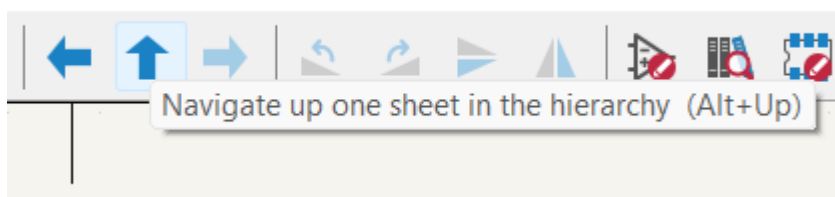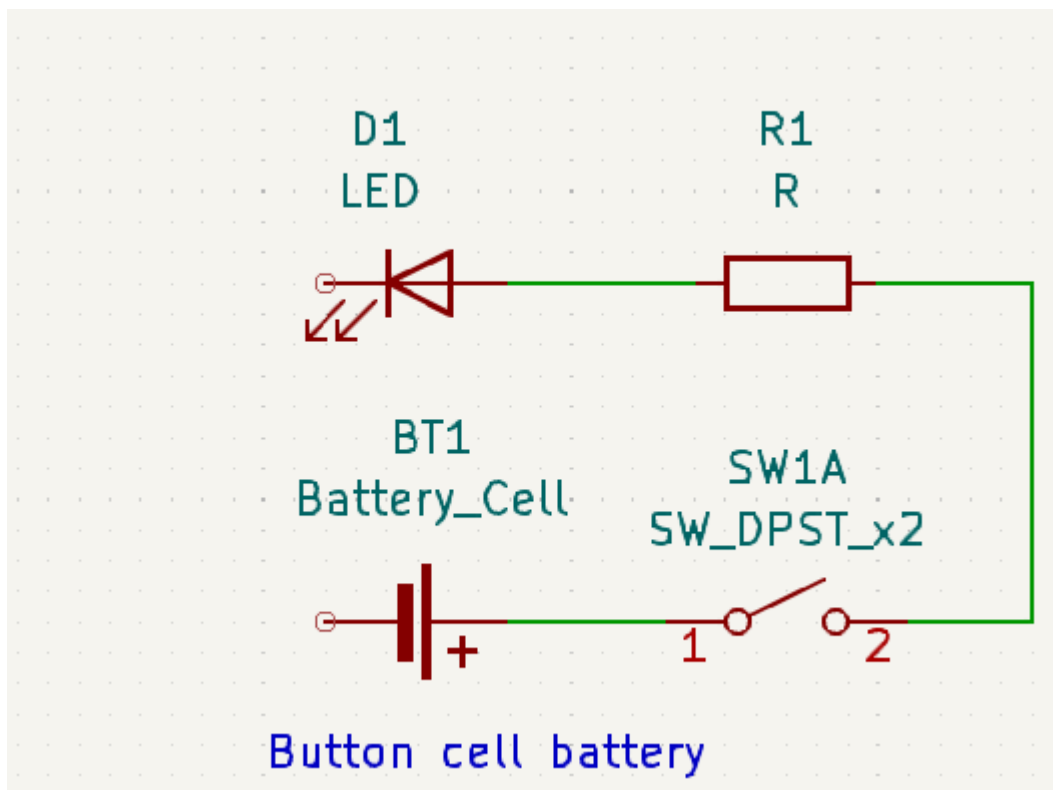

7. Now select the circuit diagram and "cut" it by pressing CTRL + X on your keyboard.

8. Go back into the hierarchical sheet and paste the circuit you use cut by pressing CTRL + V. Click the mouse to place it on the sheet. See below.
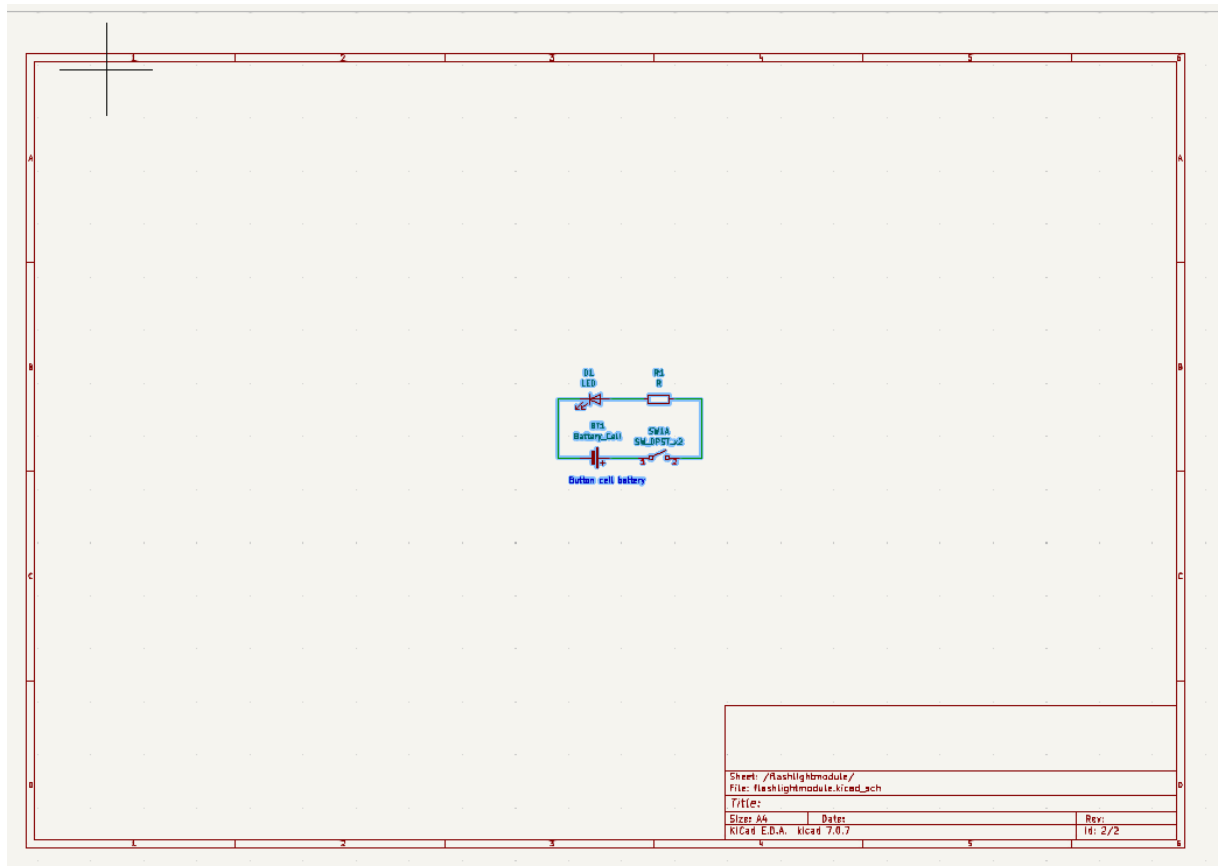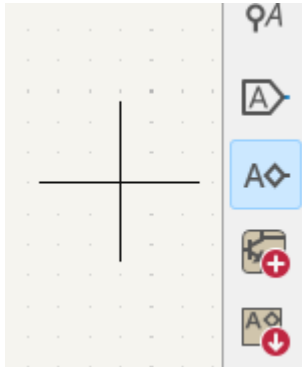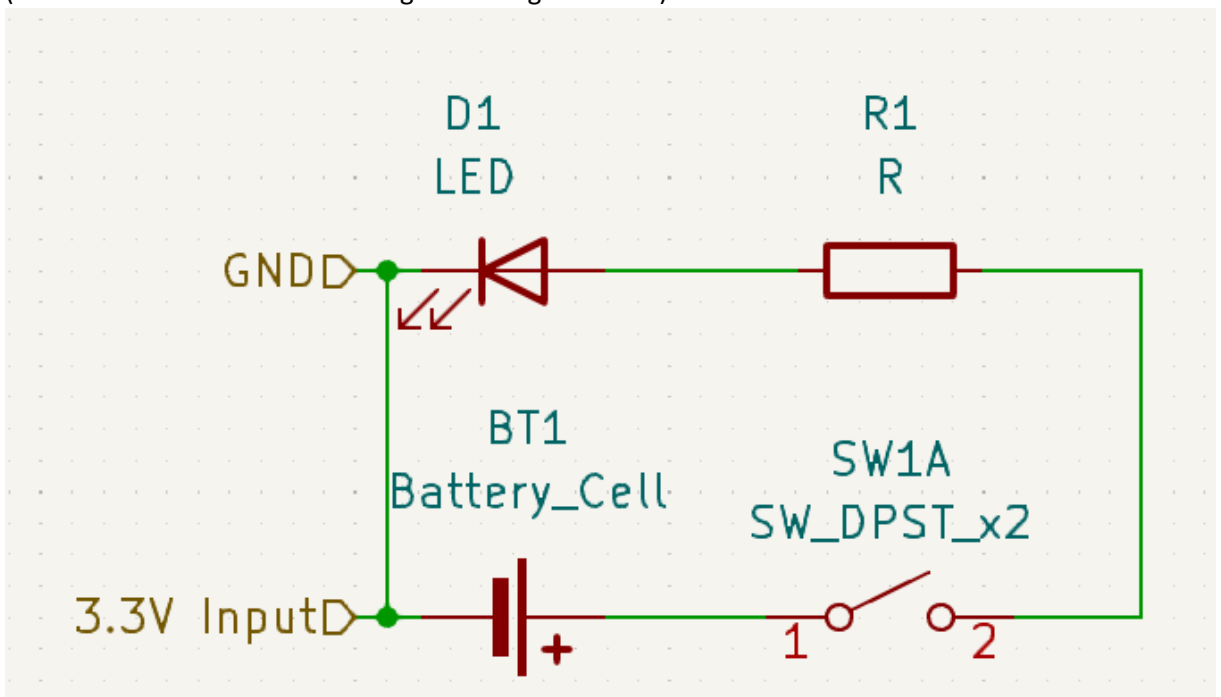

9.
10. Alter the circuit as seen below:

11.

12. Add a Hierarchical label by selecting the icon as seen below

13.

14. Write in the window that opens the text: 3.3V input and after clicking "ok" add the item to the circuit. Please do the same for the GND label and draw a wire between GND and 3.3V (make sure the labels are touching the wiring as shown).


15.

16. Now exit the hierarchical sheet by clicking the upward pointing arrow as mentioned before.

17. Hover your mouse into the hierarchical sheet box and click the right mouse button.
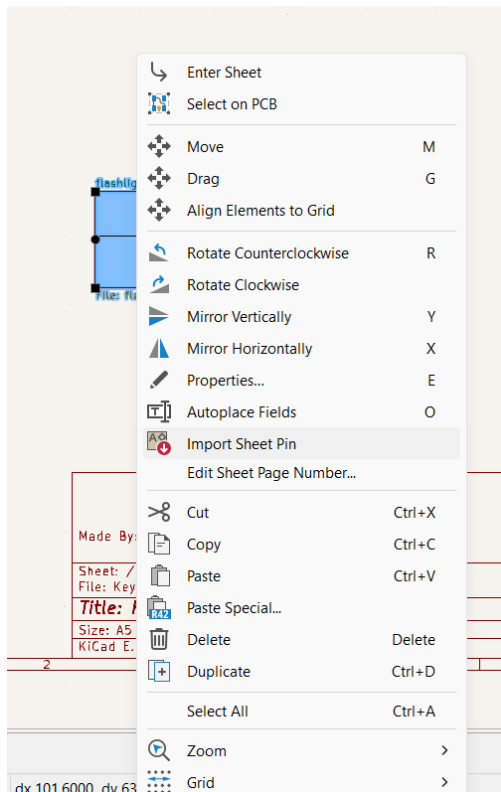
18. `dx 101,6000  dy 63`

19. Select the import sheet pin option. A GND or 3.3V input text will begin to follow your mouse within the box. You can click your mouse to place them one after the other. You will end up with something like you can see below. You can still move the pins within the box.
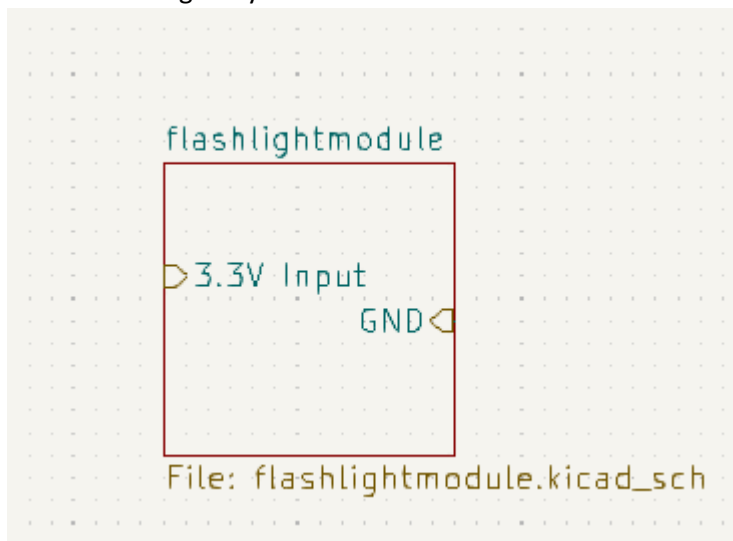
20.
21. Press "P" on your keyboard. A windows will load and open.
22. Find the +3.3V symbol and place it on your schematic sheet.
23. Do the same for the GND symbol.
24. Connect the symbols you just added to the pins coming from the hierarchical sheet.
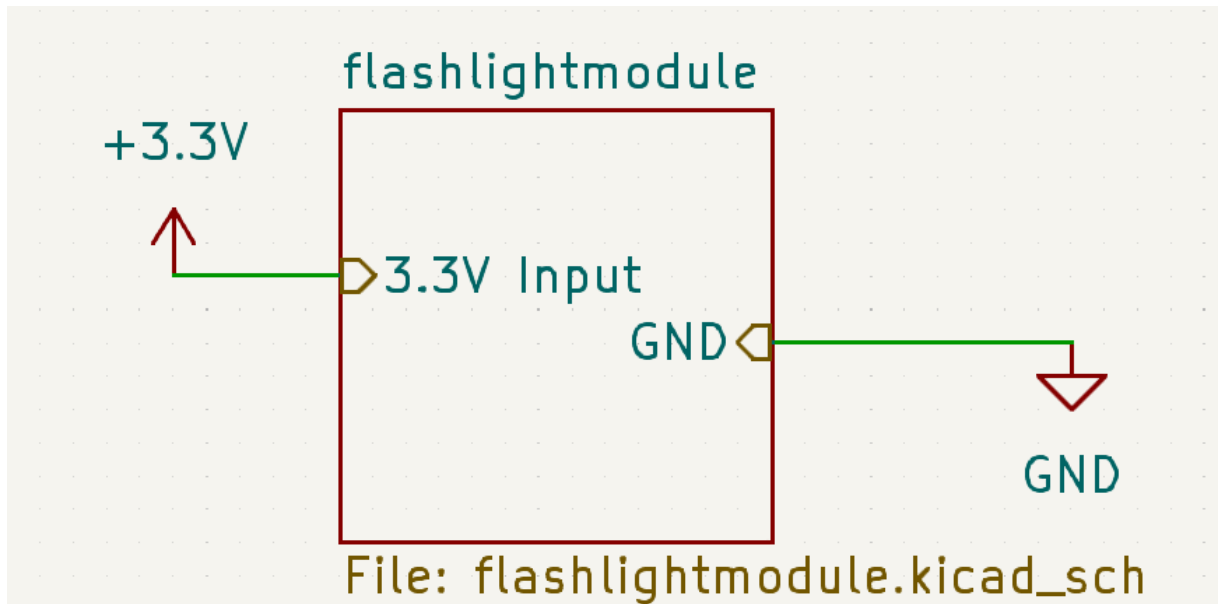
25.

26. You have now created a box (the hierarchical sheet) that contains your flashlight. It only has an input and output.

27. Since we just shrunk down our circuit; let's reduce the size of the schematic page by clicking the paper symbol as we did earlier.
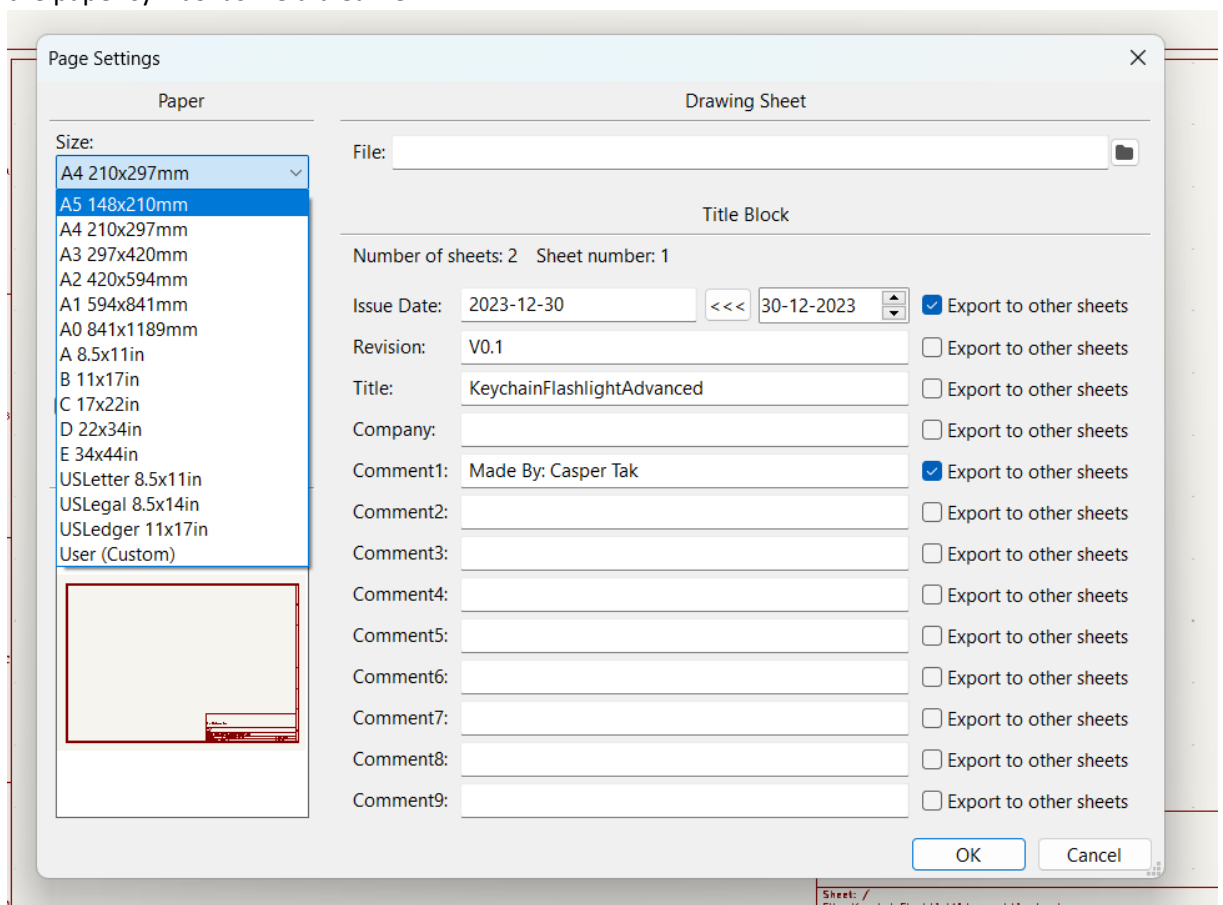
28.

29. Select A5 or add a custom size to shrink the page size.

30. Move the hierarchical sheet box back within the borders if required.

An hierarchical sheet may not be really useful for this project, but image having multiple complicated circuits that you would like to have into a "black box" with just the input and output pins coming out of it. That is what this is mostly used for.

Step 4: Importing symbols (OPTIONAL)

This chapter is optional, you can skip to step 6 if you are not interested in learning this. You will keep the original button, which will look slightly different from the one that will be used in these steps.

Sometimes you want a particular part in your design that is not in the standard KiCad Library's. You could create your own symbols and footprints, but this can take a lot of time. There are many websites as: https://www.snapeda.com/ or https://nl.mouser.com/ that either offer the symbol, footprint (combination) or even a 3D-model of a part!

For your own designs: always double check (peer review) that a imported footprint is matching the exact pinout of your physical component!

PLEASE NOTE: the FSM8JH is already in the project folder! You can continue from step 4.

1. Go to https://www.snapeda.com/ and search for "tactile button" or "**FSM8JH**"
2. Now press the orange download symbol and footprint button. For format, choose KiCad.
3. You will need to make an account, you can do that, than choose the kicad 6 or later version of the file. **Or you can choose to get the .zip file from the project folder.**
4. Follow the following steps to get the symbol imported to your library:
1. Extract the content of the downloaded **.zip* file.
2. In KiCad (Schematic editor), go to **Preferences.**
3. Click on **Manage Symbol Libraries.**
4. On the **Global Libraries** tab, click on **Browse Libraries** (the **small folder icon**)
5. Select the *.kicad_sym* file, then click **Open. (**If you get an error saying something like "read only", make a personal library to store your personal assets.)
6. The library will appear, click **OK.**
7. Click on **Symbol Editor**.
8. Type on the filter search field, and navigate to the symbol you imported.
   Double-click over it to open the file.

5. By searching the symbol you will find the newly imported symbol! Click the symbol and place it inside the hierarchical sheet you made earlier.
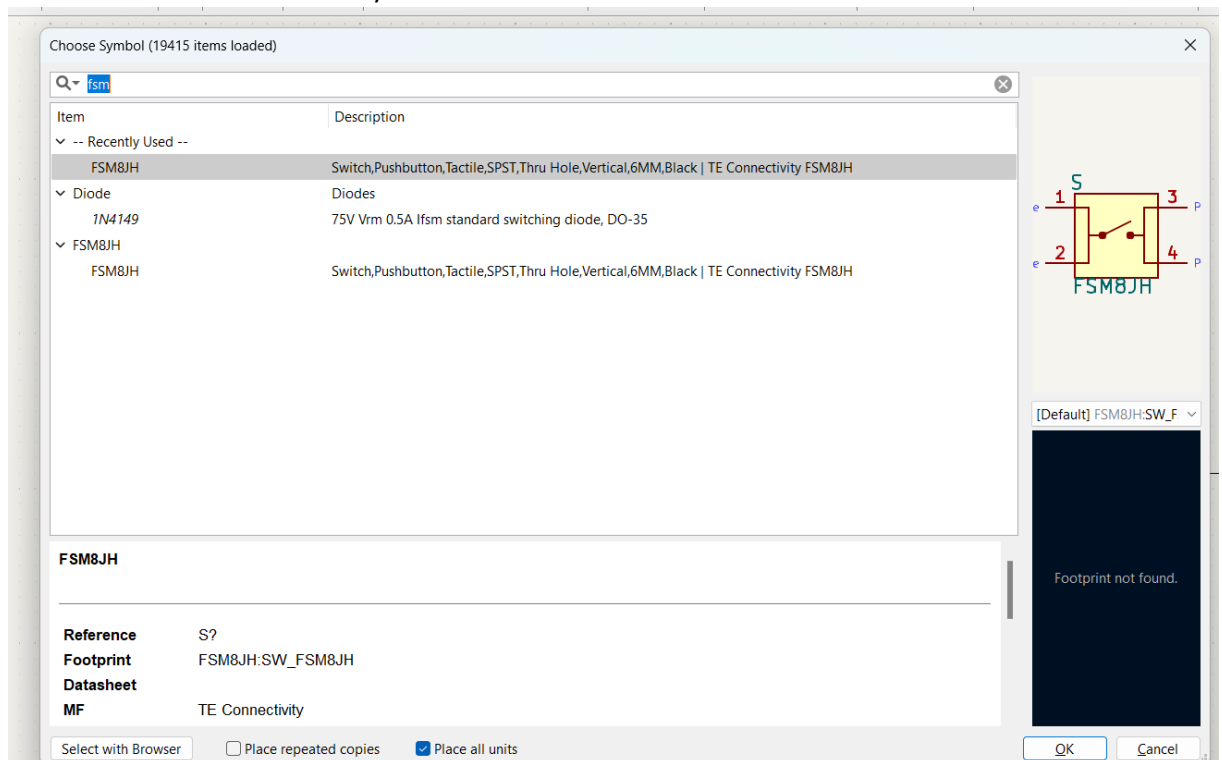

6.
7. To add the footprints follow these steps:

1. Extract the content of the downloaded *.zip* file.
2. In KiCad (PCB editor), go to Preferences.
3. Click on Manage Footprint Libraries.
4. On the Global Libraries tab, click on Browse Libraries (the small folder icon)
5. Navigate to the Folder where the *.kicad_mod* file is located. Then click Select Folder.

> Note: You will not normally see the *.kicad_mod* file on this step because you need to select the folder where it is located.

6. The library will appear, click OK.
7. Right click on the SW1 footprint on your PCB design.
8. Click the Open in footprint editor icon (seen below).

9. Open in Footprint Editor   Ctrl+E

10. Go to file, import, footprint…
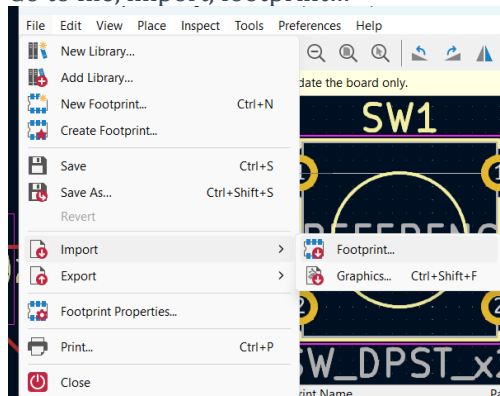
11.
12. Search fort he SW_FSM8JH.kicad_mod file and double click it or press "open". It is located in the project folder.
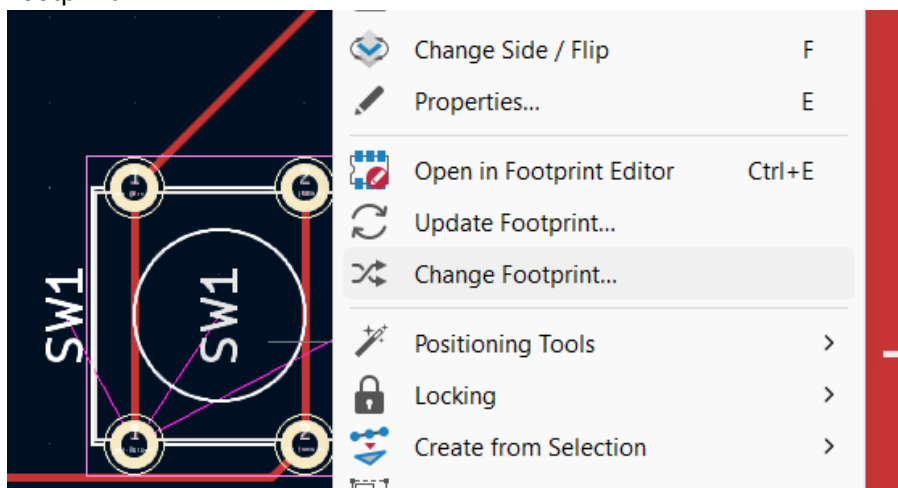
13

13. You will see a slightly altered version of the button we had earlier. Safe the file in the selected library while you exit the popup window
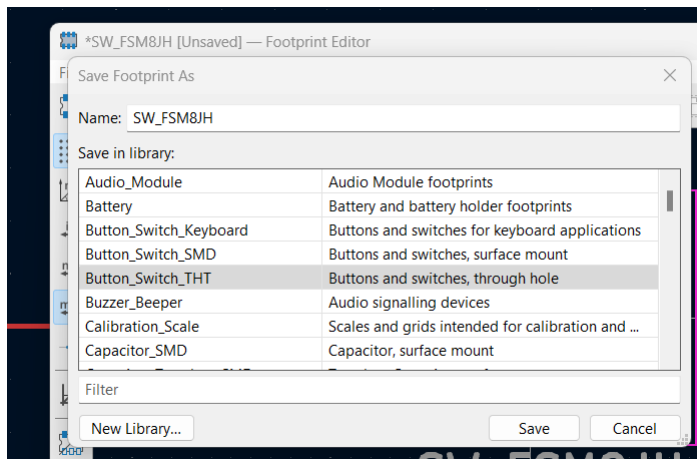
14.

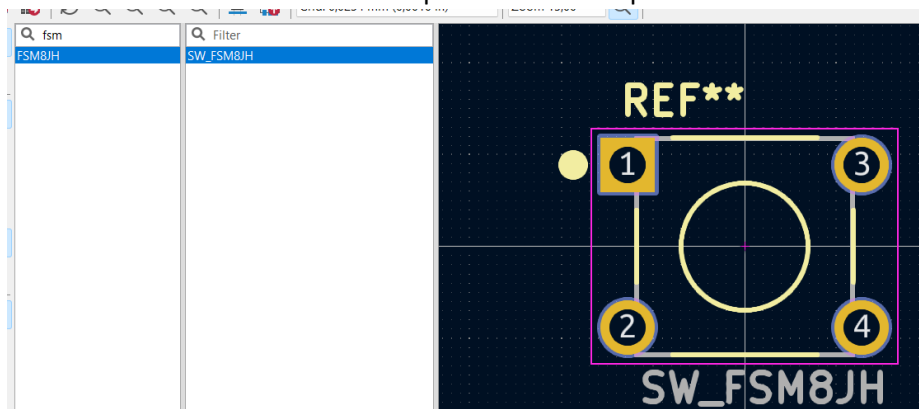15. The button is now saved to in the Button_Switch_THT library.

8. Lets add the new footprint to the board by selecting SW1 and then selecting "Change Footprint…"

9.

10. In the popup window, click the "second" library icon for the new footprint library id.

11. Search for the FSM8JH. You will be presented with a preview.

12.

13. Click twice on the right SW_FSM8JH (now highlighted in blue) you will see the new footprint id in the box after the popup window closes.

14. Click on "Change" and exit/close the window.

15. You have now got a new footprint for the button.

An important side-note as that this process can be done many ways and could and should be "automated". There is for example a program for the parts distributor Mouser that automatically import the files you want to add to your library. For Mouser it is SamacSys, which allow you to import the footprints automatically you find along components on their website. This may safe you a lot of time.

## Step 5: Integrating the new button (OPTIONAL)

1. Go to the PCB editor if you aren't there already.

Your PCB probably looks something like this:
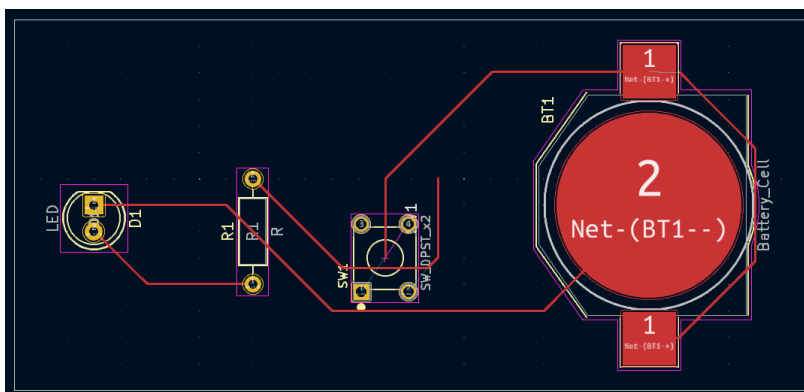
2. Lets fix the disconnected button first. You should be able to do that with the skills you gained/now have.
3. But there seems to be a problem… 2 of the pads are not connected to anything.
4. We forgot to add in the new symbol at the schematic.

5. Go back to the schematic and connect the button you have placed before within the hierarchical sheet (with 4 connections) as shown below:
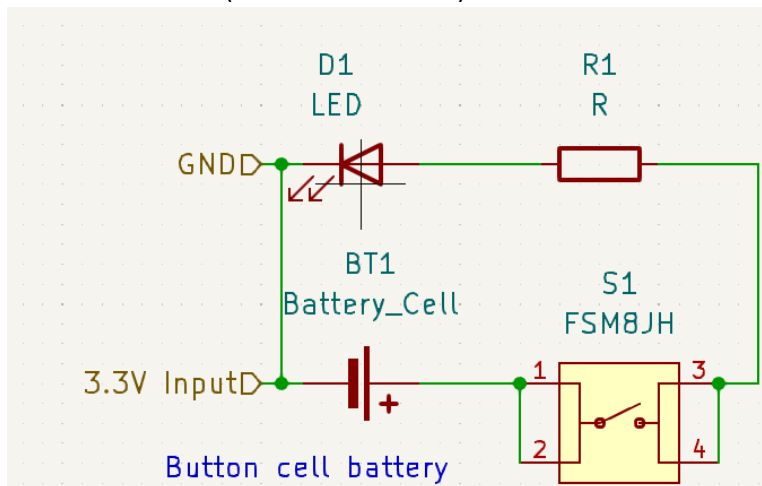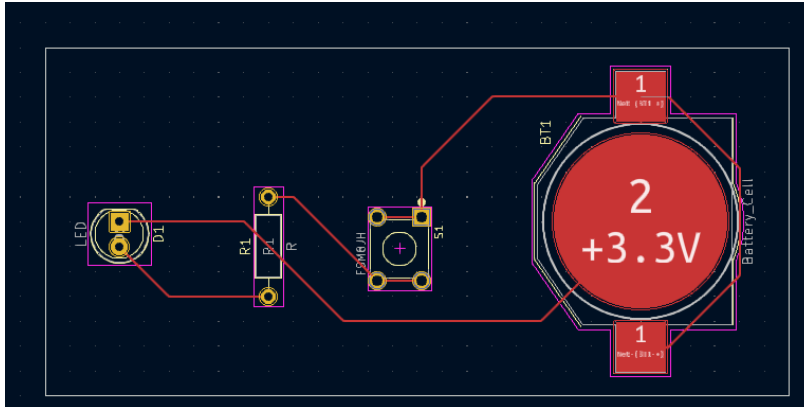

6.
7. You can run a ERC, but don't have to.
8. Go back to the PCB editor and click the button (or F8) on the top toolbar
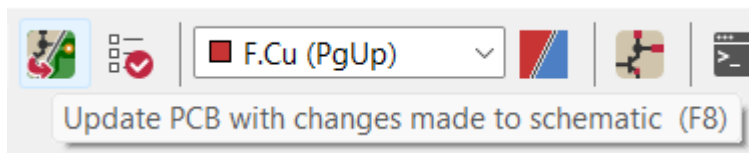
9.
10. Click "Update PCB" and after closing the window, reconnect the button.

Your PCB should now look something like this:

(please ignore the +3.3V footprint error for bt1, it should be GND in your design)

11. Let's add some basic information to the page settings here. This is just the same as on the schematic page settings. Also change the paper size to A5 format.

Step 6: Redesigning the board outline and adding a hole

The PCB as it is now is a little boring and not really a keychain, because of the absence of a hole. We will now make the PCB **more tactile and visually appealing**, ensuring it is nicer to touch. Select the Egde.Cuts layer on the right.

1. Then select the "circle" tool and draw a circle at the upper right corner (as shown below). You can change the Grid settings in the top toolbar, this allows you to move the circle with more (or less) precision.
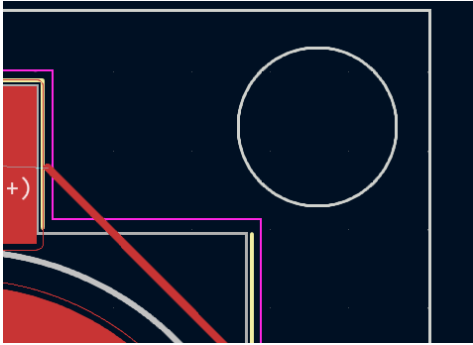

2.

3. Now let's do something about those sharp corners by selecting the "arc" tool.

4. Try to draw an arc in the upper right corner of the PCB. You can try to use the grid settings to get more precision (0.2540, worked for me).

If you are having trouble with the placement of the lines, please try changing your grid settings to a different size. This will either increase or decrease your grid size and therefor precision.
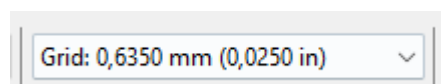
5. You can do this for all for corners, by moving the arcs and connecting the lines to them. This may be a bit of trial in error.
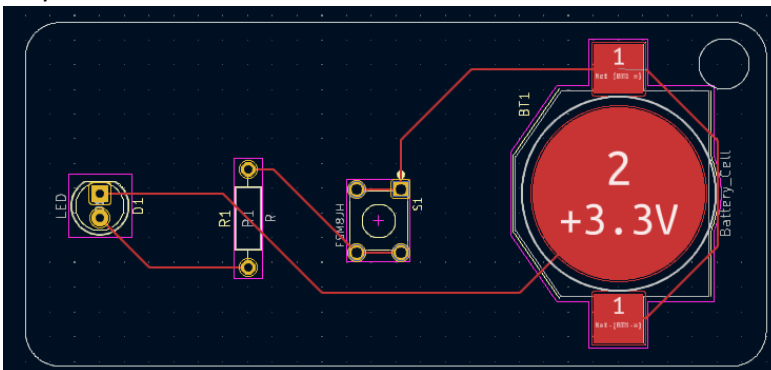
6.

Step 7: Adding text and pictures to the PCB

Of course, the PCB is missing some information on it; there should be the name of the developer, the version number of the board and the date as a starter. Maybe you can think of more things to add. Apart from that, we should add an image to the back of the board so it is clear what the board was/is used for.

1. Now let's add some text to the board by using the "text" item tool.
2. Make sure to select the Layer: F.Silkscreen
3. Write some text in the text box and click "ok".

4. Place the text somewhere on the board, for example, the upper left corner.

KeychainFlashLightAdvanced by Casper R. Tak
V0.1
31-12-2023

5. We will now add a .bmp file.
6. Go to the main page of KiCad and select the Image Converter:

Image Converter
Convert bitmap images to schematic symbols or PCB footprints
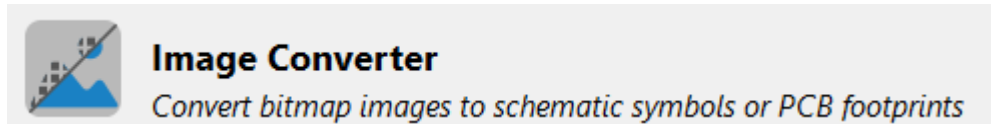
7. Click: "Load Sourcde Image" and select the *logo for workshop.bmp file.*
8. *Choose the following settings and click: "Export to Clipboard"*
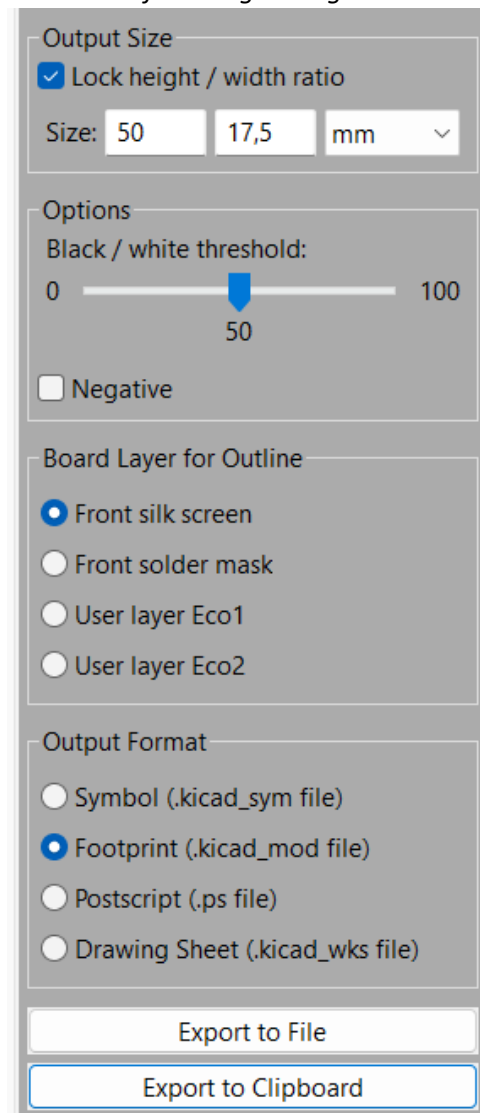
9.
10. Press CTRL + V while you are in the PCB editor and press "f" before placing the logo somewhere on your PCB, this switches it to the back side of the PCB.
11. You can remove those *** by double clicking the image on your PCB and unticking the "show" box with the reference designator.

12. *You have successfully added your own.bmp file!*
13. *You can add some more logos by choosing a footprint (or press A). I will be adding the: OSHW-Logo2_24.3x20mm_SilkScreen to my board.*
14. *You can see the look of the frond and back of the PCB in the 3D-viewer (press ALT+ 3)*
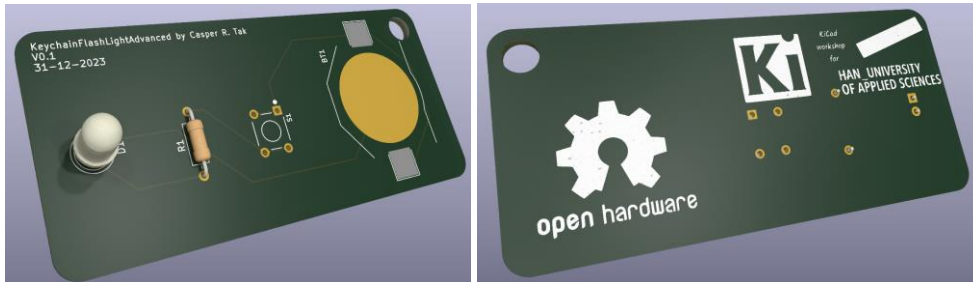
15.
16. *Now in the final step we will add the missing 3D-models.*

Step 8: Adding 3D-models

For your final step, we will be adding the missing 3D-models to the model. This may seem useless for now, but a very useful feature of KiCad is the ability to export your 3D model (in .step files for example). Tis 3D-model can be imported into other CAD programs like SolidWorks or Fusion360 and such. Your IDE co-worker/peer will thank you later.

1. *Select the whole footprint of BT1 in the PCB editor, right click and select "properties" Footprint properties will open.*
2. *Go to the 3D Models tab and the following 3d model:*

${KICAD7_3DMODEL_DIR}/Battery.3dshapes/BatteryHolder_Keystone_3034_1x20mm.step

3. *You can also locate it manually in the project folder, where it is called:*

BatteryHolder_Keystone_3034_1x20mm.step

4. *The 3D model should appear on the preview. Use the 3 boxes on the left (Scale, Rotation and Offset) to rotate the holder in a way that the battery will be inserted from the right side of the keychain (so the battery does not get blocked by the button)*
5. *Open the 3D Viewer (ALT+3) to check if it resembles this:*
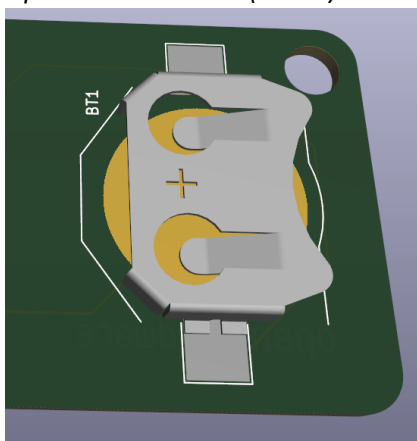
6.
7. *Now we will add the button as the final 3D model*
8. *We could download the 3D model from the snapeda website, but again, I downloaded it for you into the project folder. It is called:*

FSM8JH--3DModel-STEP-56544.STEP

9. The model will most likely look like this when you import it:
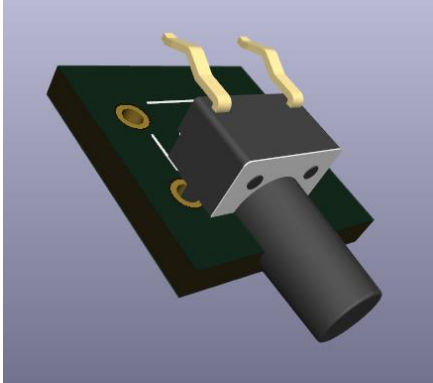
10.
11. Of course, you can just rotate it. Rotate it on the X: and the Z: axis for the correct orientation.
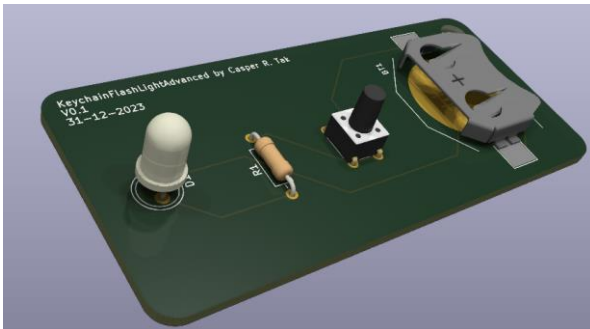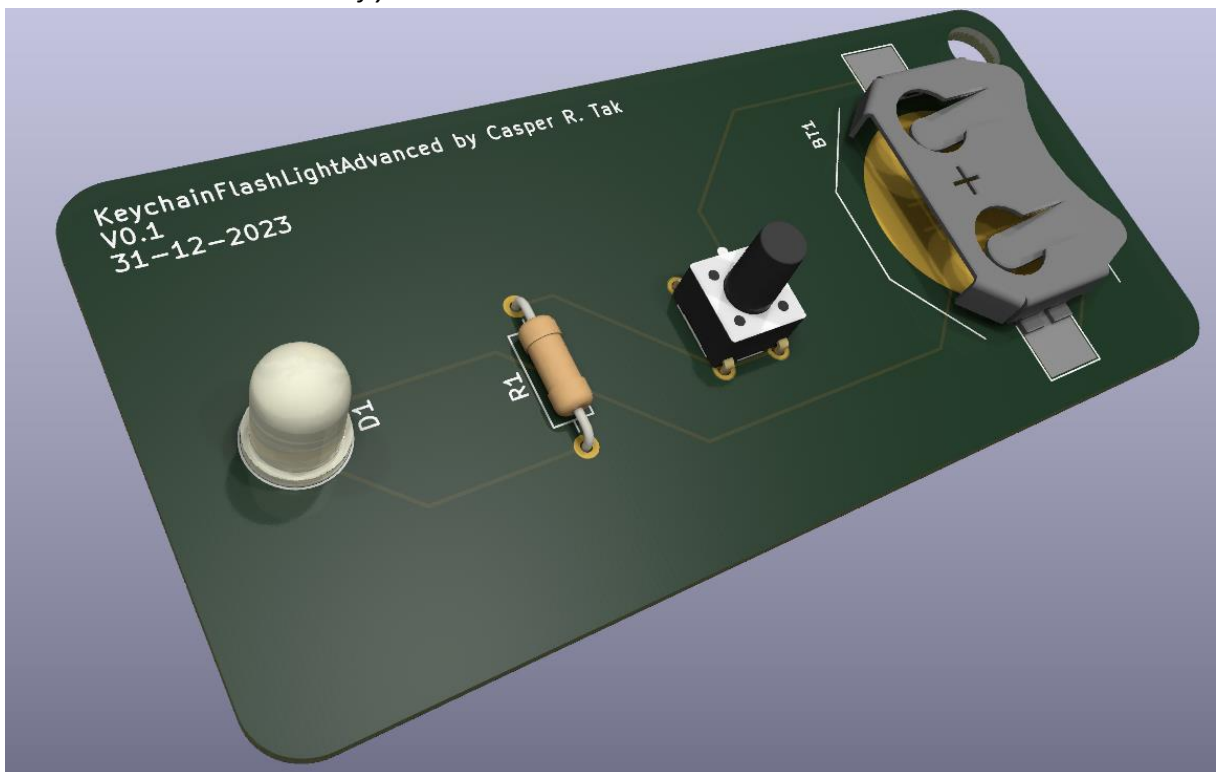12. Press "ok", when you are finished.
13. You should now have a 3D viewer model that looks like this:

14.
15. Now we just need to fix the floating LED. You should be able to do this on your own now.
16. This would be the end result of your PCB!

17.

Congratulations, you finished the KiCad Advanced workshop!

Congratulations on completing the KiCad Advanced Workshop! You've now acquired advanced skills in both the schematic and PCB editors, allowing you to personalize your designs and optimize your workflow. From adding your name to the schematic to incorporating 3D models, you've explored a range of features that elevate your KiCad proficiency.

Remember, these advanced techniques open up possibilities for more professional and customized PCB designs. Whether you're adding personalized touches, creating hierarchical sheets for complex circuits, or importing symbols and footprints, you've gained valuable insights into the world of advanced PCB design.

Continue to explore and apply these skills in your projects, and don't hesitate to experiment with the features you've learned. As you further integrate these techniques into your design process, you'll find yourself creating more polished and visually appealing PCBs.

Thank you for joining us in this workshop, and we hope these advanced KiCad skills empower you in your future electronic design endeavors. Happy designing!

3. Fun Bonus Chapter: Plugins

In the ever-evolving landscape of PCB design, KiCad 7 introduces a powerful feature—Plugins—that extends the capabilities of the software. Add-ons are downloadable modules that enhance and customize your KiCad experience, offering new functionalities and tools tailored to specific design requirements. Some add-ons safe you a lot of time and effort. But let's delve into this exciting realm and explore with a fun plugin: the Round Tracks Plug-in.

Discovering and Managing Add-ons

1. **Accessing the Add-ons Manager:** In KiCad 7, the Add-ons Manager serves as the gateway to a wealth of additional features. To access it, navigate to the main menu and select "Tools" > "Add-ons Manager."

2. **Browsing Add-ons:** Browse through the available add-ons categorized by functionality, from design utilities to manufacturing aids. Each add-on comes with a description, version information, and compatibility details.

3. **Installation Process:** Installing an add-on is a breeze. Simply select the desired add-on, click "Install," and KiCad will seamlessly integrate it into your workflow. You can also add add-ons found on websites, this is beyond the scope of this bonus chapter.

Enhancing Aesthetics with Round Tracks

The Round Tracks add-on is particularly appealing for those keen on elevating the aesthetic appeal of their PCB layouts. This plug-in introduces rounded traces, a departure from the traditional sharp-edged tracks. Follow these steps to integrate this add-on into your design:

1. **Accessing the Plug-in:** After installing the add-ons, open your PCB editor and navigate to the top menu. Click on "Tools" and then select "Round Tracks Plug-in."

2. **Configuring Round Tracks:** The Round Tracks configuration window will appear. Here, you can set parameters such as the track width, corner radius, and the segments per 90-degree turn. Experiment with these settings to achieve the desired visual effect for your traces.

3. **Applying Round Tracks:** Once configured, select the traces you want to round. This can be done individually or in groups. Click "Apply Round Tracks," and watch as your sharp corners transform into smooth, rounded curves as seen below.
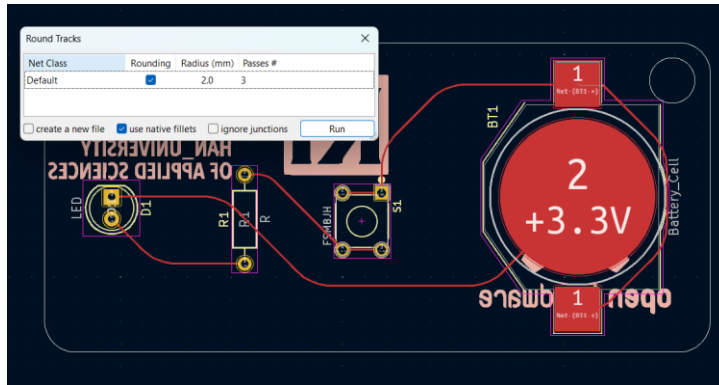
4. **Fine-tuning:** The add-on allows for real-time adjustments. If needed, revisit the configuration window to fine-tune the parameters until you achieve the perfect balance between aesthetics and functionality.

Benefits of Round Tracks

- **Visual Appeal:** Rounded tracks contribute to a more polished and visually pleasing PCB layout. This aesthetic enhancement can be particularly beneficial for projects where design plays a crucial role. Incorporating rounded tracks can give your PCB design a more modern look, making it stand out in presentations or when showcased to clients and collaborators.

- **Reduced Stress Points:** Rounded corners reduce stress concentrations on the PCB, potentially enhancing the overall reliability of the design, especially in applications where mechanical stress is a concern.

As you explore and integrate these tools into your projects, you'll find that KiCad continues to be a dynamic and versatile platform for all your PCB design needs.

3.1 A few noteworthy plugins:

Some useful and fun plugins. Please explore these and more in the Plugin and Content Manager on the main page of KiCad.
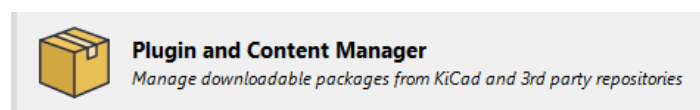
Download more from the internet.