Afbeelding met tekst, Lettertype, logo, symbool

Automatisch gegenereerde beschrijving

Workshop KiCad7

Advanced workshop

WRITTEN BY: CASPER R. TAK

2023

Contents

[1. Workshop Version management 2](#_Toc154866284)

[Workshop Summary: What will you learn in this workshop? 3](#_Toc154866285)

[Step 1: Add information to the schematic page 5](#_Toc154866286)

[Step 2: Add informative text to the schematic sheet 6](#_Toc154866287)

[Step 3: Adding a hierarchical sheet 7](#_Toc154866288)

[Step 4: Importing symbols 12](#_Toc154866289)

|  |  |  |
| --- | --- | --- |
| 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. |

# Workshop Version management

## 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:**
2. **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.
3. **Advanced Schematic Organization:**
   * *Hierarchical Sheets:* Master the art of creating hierarchical sheets, allowing for a more organized and modular approach to complex designs.
4. **Library Management:**
   * *Component Imports:* Explore the process of importing components from external libraries, expanding your component options and design flexibility.
5. **In PCB Editor:**
6. **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).
7. **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.
   * *3D Models:* Add missing 3D models to enhance the 3D viewer experience, ensuring a comprehensive representation of your PCB design.
8. **Extra Enhancements:**
9. **Library Integration:**
   * *Snapeda/SamacSys Integration:* Explore how to seamlessly incorporate components from external libraries like Snapeda/SamacSys, accessing an extensive range of electronic CAD symbols and models.
10. **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.

*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.*

## 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. Afbeelding met tekst, schermopname, Lettertype, nummer

   Automatisch gegenereerde beschrijving
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. Afbeelding met tekst, schermopname, scherm, software

   Automatisch gegenereerde beschrijving
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. Afbeelding met Rechthoek, schermopname, lijn, diagram

   Automatisch gegenereerde beschrijving
3. A window will open, fill in the following text:
4. Afbeelding met tekst, scherm, schermopname, software

   Automatisch gegenereerde beschrijving
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. Afbeelding met tekst, diagram, lijn, schermopname

   Automatisch gegenereerde beschrijving

## Step 3: Adding a hierarchical sheet

1. Press the hierarchical sheet icon on the right tool bar (see image below)
2. Afbeelding met tekst, schermopname, Lettertype, diagram

   Automatisch gegenereerde beschrijving
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. Afbeelding met tekst, schermopname, software, scherm

   Automatisch gegenereerde beschrijving
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.Afbeelding met tekst, schermopname, Lettertype, lijn

   Automatisch gegenereerde beschrijving
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. Afbeelding met tekst, schermopname, Rechthoek, diagram

   Automatisch gegenereerde beschrijving
10. Alter the circuit as seen below:
11. Afbeelding met tekst, diagram, Lettertype, schermopname

    Automatisch gegenereerde beschrijving
12. Add a Hierarchical label by selecting the icon as seen below
13. Afbeelding met schermopname, diagram, ontwerp

    Automatisch gegenereerde beschrijving
14. Write in the window that opens the text: 3.3V input and after clicking “ok” add the item to the circuit as following: (make sure the label is touching the button cell symbol as shown)
15. Afbeelding met tekst, Lettertype, schermopname, lijn

    Automatisch gegenereerde beschrijving
16. Now do the same but for the ground (output) label, as shown below:
17. Afbeelding met tekst, Lettertype, schermopname, diagram

    Automatisch gegenereerde beschrijving
18. Now exit the hierarchical sheet by clicking the upward pointing arrow as mentioned before.
19. Hover your mouse into the hierarchical sheet box and click the right mouse button.
20. Afbeelding met tekst, schermopname, software, Webpagina

    Automatisch gegenereerde beschrijving
21. 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.
22. Afbeelding met tekst, schermopname, Lettertype, nummer

    Automatisch gegenereerde beschrijving
23. Press “P” on your keyboard. A windows will load and open.
24. Find the +3.3V symbol and place it on your schematic sheet.
25. Do the same for the GND symbol.
26. Connect the symbols you just added to the pins coming from the hierarchical sheet.
27. Afbeelding met tekst, schermopname, Lettertype, lijn

    Automatisch gegenereerde beschrijving
28. You have now created a box (the hierarchical sheet) that contains your flashlight. It only has an input and output.
29. 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.
30. Afbeelding met tekst, schermopname, scherm, software

    Automatisch gegenereerde beschrijving
31. Select A5 or add a custom size to shrink the page size.
32. 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