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](#_Toc152684324)

[Workshop Summary: What will you learn in this workshop? 3](#_Toc152684325)

|  |  |  |
| --- | --- | --- |
| Version | Date | Changes/notes |
| V0.1 | 05-12-2023 | Setup of document |

# 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.
   * *Netlabels:* Understand how to use netlabels effectively to simplify and streamline your schematic connections.
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.*