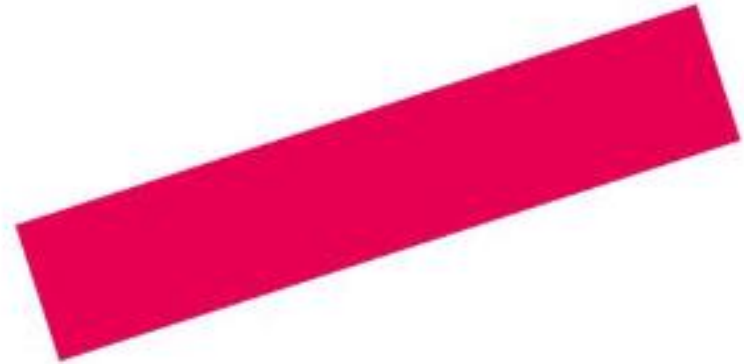




KiCad
workshop
for



HAN_UNIVERSITY
OF APPLIED SCIENCES

Introduction and Advanced Workshop

Created by: Casper R. Tak
Modified by: Emiel V

KiCad workshop introduction

Why go through the trouble of:

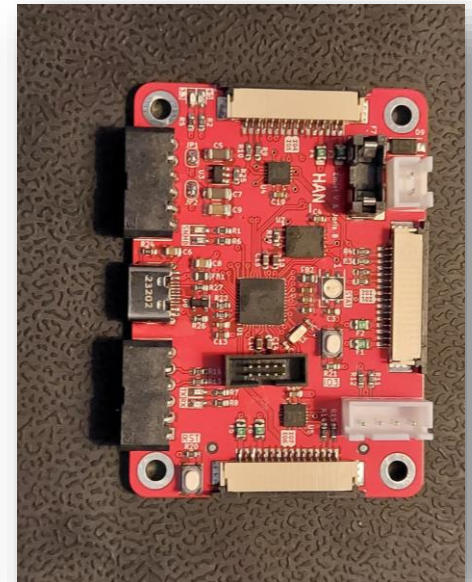
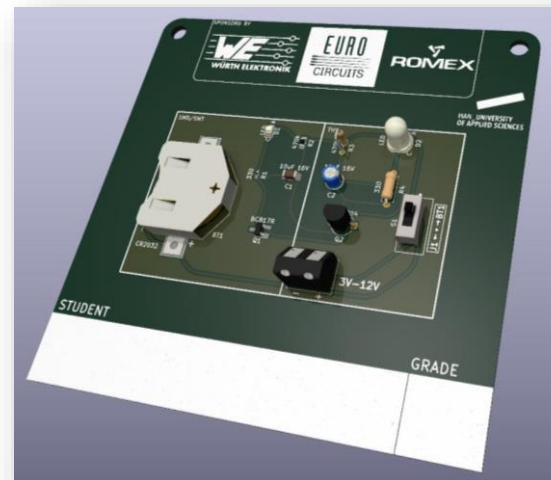
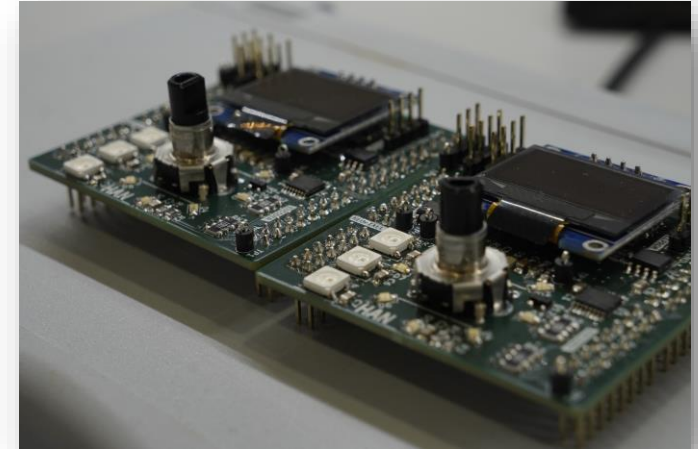
- Designing the PCB
- Waiting for it to arrive
- Figuring out and finding the (many) mistakes
- And then doing it all over again, while the deadline is approaching...

Timing: when to start designing a PCB?

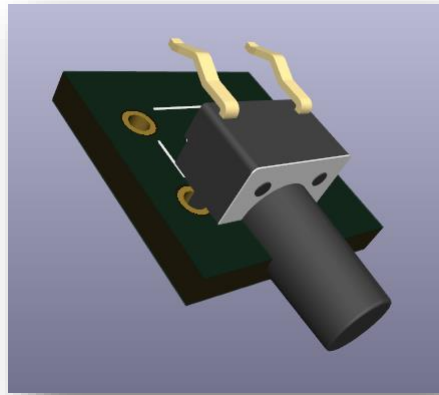
- Breadboard first (if possible)

Why not Altium/Eagle/etc.?

- Open source
- Regularly updated and added features
- Plugins
- Cross-Platform Compatibility
- Export options



The Workshops



1. Introduction Workshop

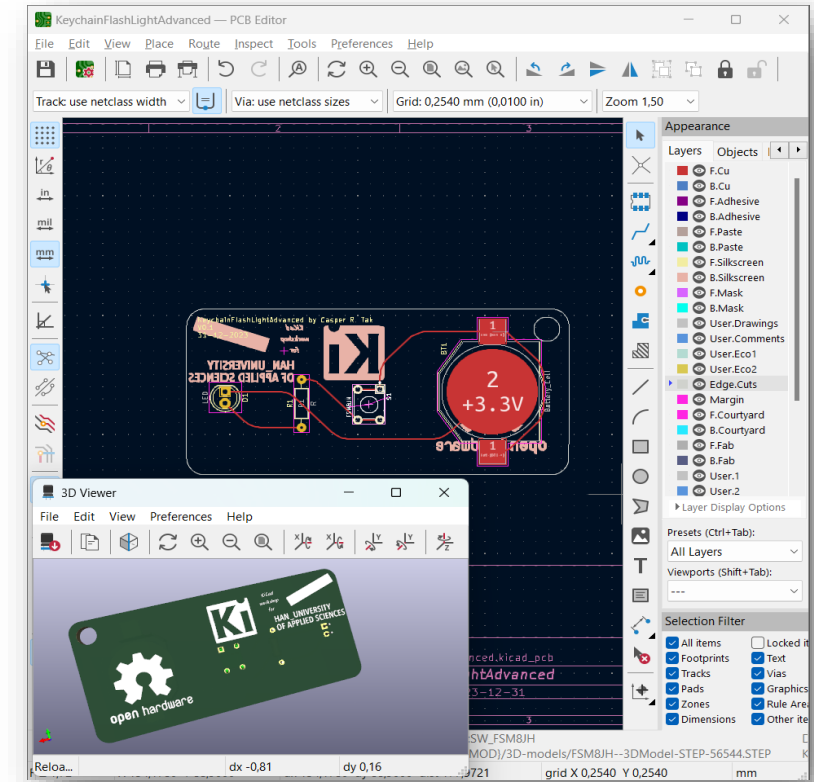
- project setup, component addition, wiring, footprint assignment, error checking, PCB design, board outline creation, routing connections, and concluding with a 3D review.

2. Advanced Workshop

- Personalized schematic touches, text annotations, advanced schematic organization, netlabels, library management, aesthetic PCB customization, rounded PCB corners, silkscreen additions, symbolic enhancements, library integration and KiCad add-ons.

3. Microcontroller Workshop

- Put what you learned into practice in a simple microcontroller board using a STM32, Tag-Connect and USB-C port. Includes some extra tips and tricks.



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

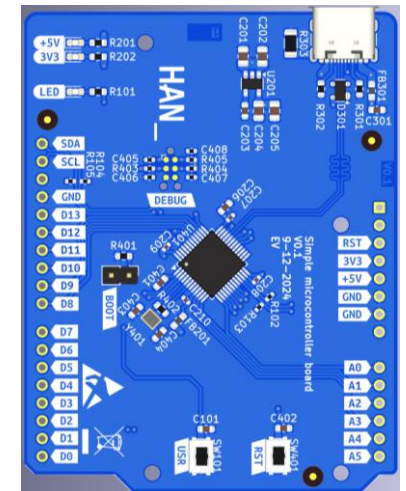
6. You can run a ERC, but don't have to.

7. Go back to the PCB editor and click the button (or F8) on the top toolbar

8. Click "Update PCB" and after closing the window, reconnect the button.

9. Your PCB should now look something like this:

10. 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.



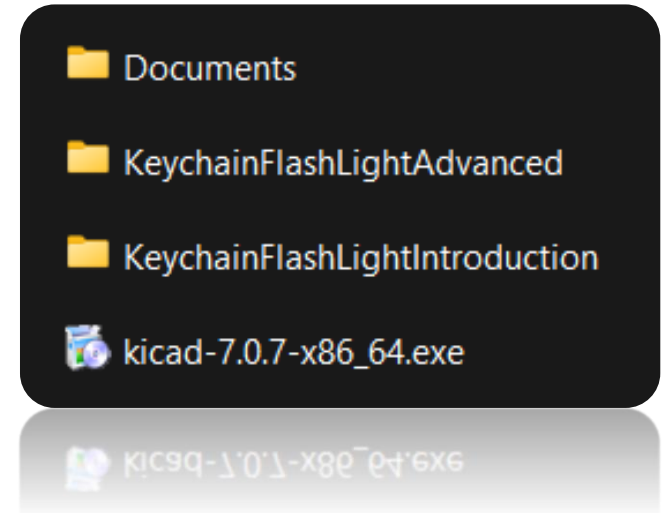
Requirements

Mandatory:

- A laptop (Preferably with Windows)
- The workshop files ([from GitHub](#))
 - Documents (Workshop specific)
 - Project files (Workshop specific)
 - KiCad V8 installation file ([from official KiCad website](#))

Highly Recommended:

- A computer mouse
- An internet connection



Good luck and have fun!

Need any help or have questions in general?

ASK! We are here to help each other learn new things because we like electronics!

Finished? Please try and surprise us with your own additions, make it YOUR design! (Maybe download a funny bitmap for on your PCB?)

Feedback? Yes please, we want to make this a fun experience for ALL!



*KiCad
workshop
for*



**HAN_ UNIVERSITY
OF APPLIED SCIENCES**

Useful links

<https://www.youtube.com/@PhilsLab>

<https://www.fedevel.com/blog>

<https://resources.altium.com/?tab=highlights>



*KiCad
workshop
for*



**HAN_ UNIVERSITY
OF APPLIED SCIENCES**