

# KiCad EDA

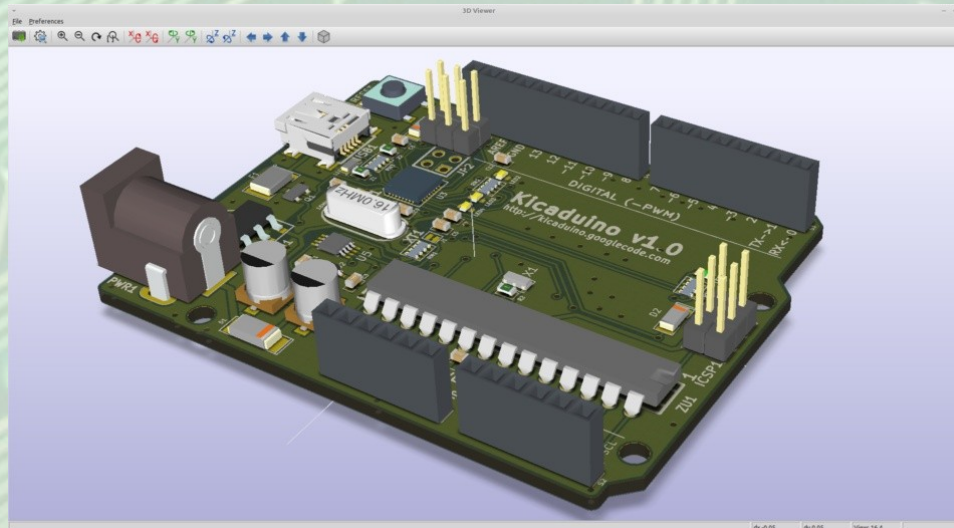
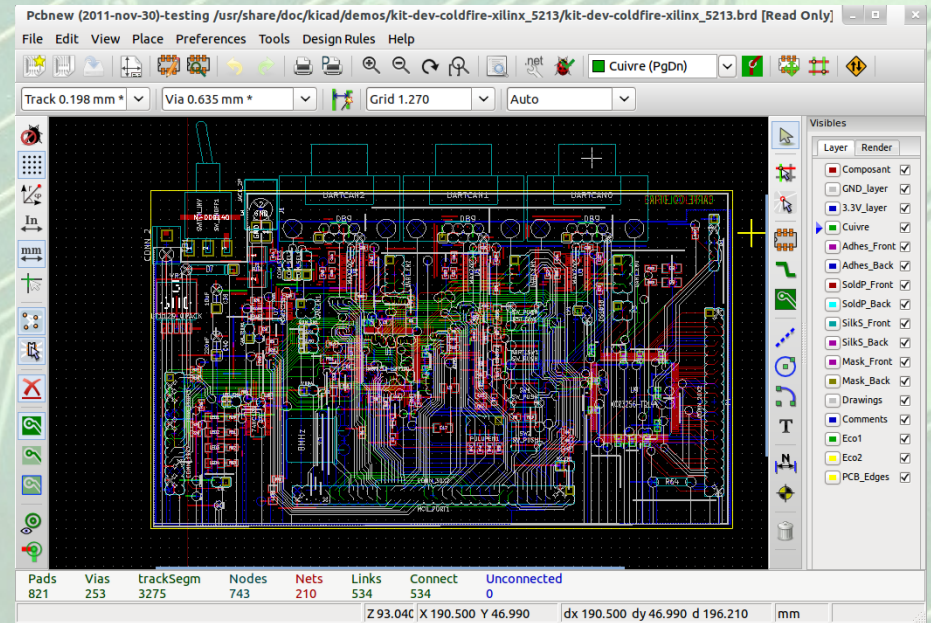
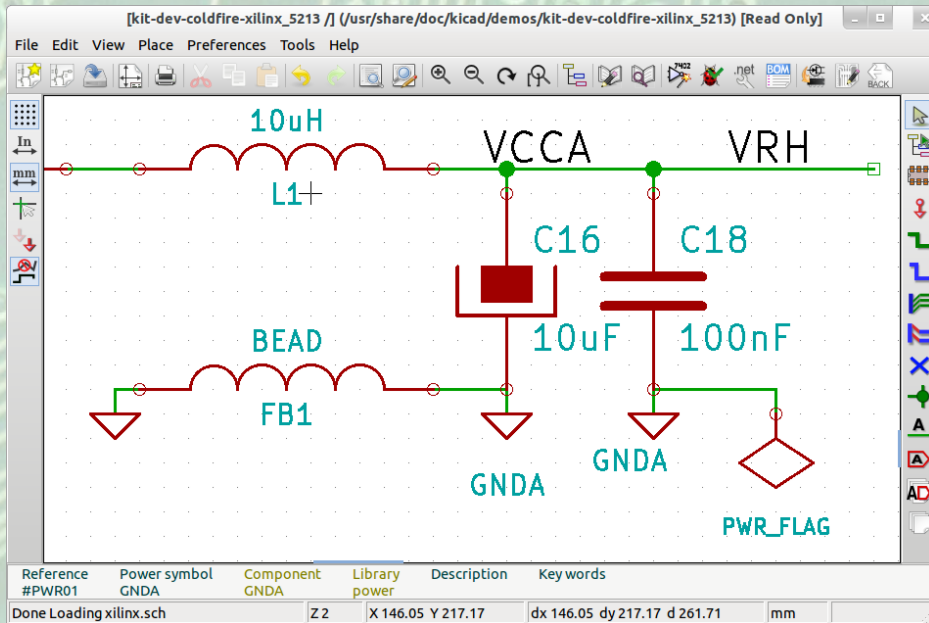
Joel Matějka

- About KiCad
- Technical parameters
- Project organisation
- My semestral work



# About KiCad EDA

## Schematic Capture and PCB Layout





# About KiCad EDA

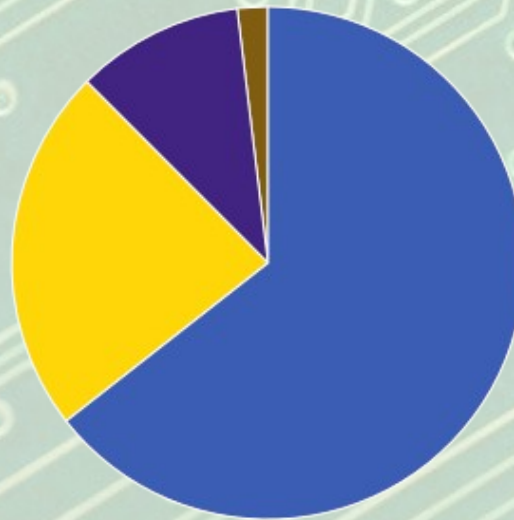
- <http://www.kicad-pcb.org>
- License GNU GPL v2
- Windows / Linux / OS X
- First release date in 1992
- Supported by CERN,  
University of Grenoble,  
SoftPLC



# Technical solution

- Written mostly in C++
- GUI interface using wxWidgets
- Stable and Product branch

63 % - C++  
24 % - XML  
11 % - C  
2 % - Other





# Developers team

- 6,853 commits made by 179 contributors
- 602,508 lines of code
- Code and bugtracker hosted on launchpad
- Bazaar version control
- Mail list or merge request

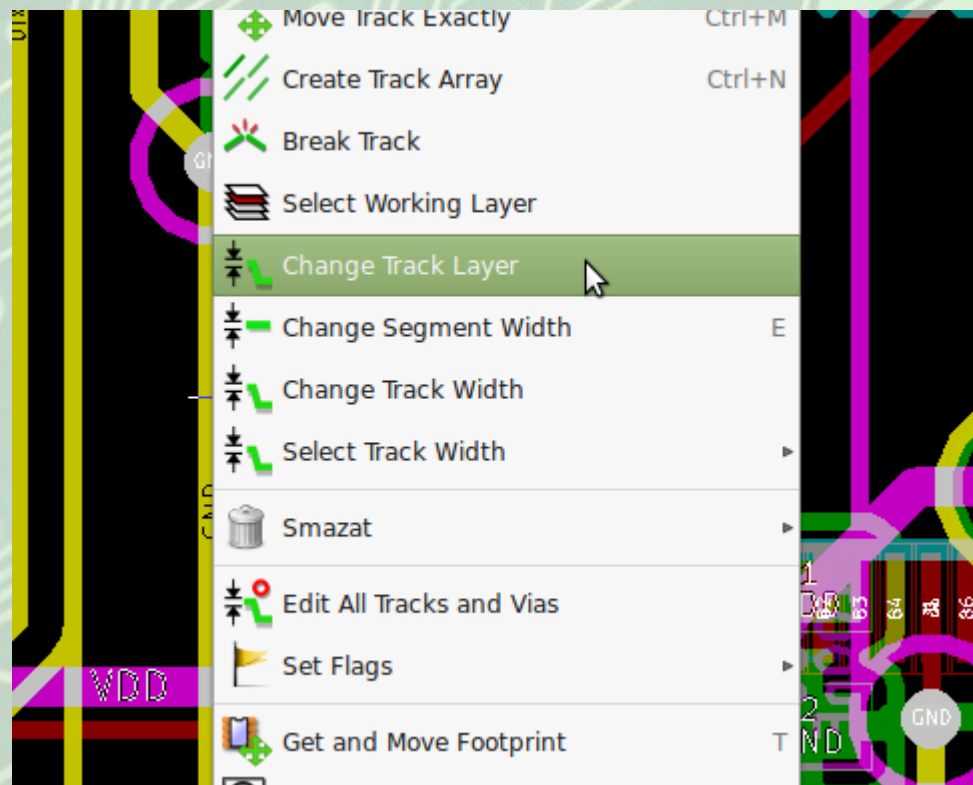




# Change layer of existing track

- New feature – 11 people want this
- Bug reported 2007

- Lock trace
- Change layer (check DRC)
- Update dependencies
- Unlock trace
- Add to undo
- Redraw





# Resources and links

Project homepage:

<http://www.kicad-pcb.org>

Source code:

<https://launchpad.net/kicad>

Developers mail-list:

<https://launchpad.net/~kicad-developers>

Bazaar version control system:

<http://bazaar.canonical.com/en/>

OpenHub site:

<https://www.openhub.net/p/kicad>

Selected bug:

<https://bugs.launchpad.net/kicad/+bug/593953>