What's new in Kicad?
CERN contribution status and plans

ICALEPCS Open Hardware Workshop
San Francisco, October 6th, 2013
Outline

- CERN's involvement in Kicad
- Our contribution so far
- Plans for the future
- Summary
- A little demo
Why support Kicad?

• No F/OSS PCB design tools of sufficient quality and ease of use
• Proprietary tools thwart collaboration between different institutes and companies
• Licensing costs are very high
• Academic licenses prohibit commercialization of our projects
Our involvement

- We chose Kicad over gEDA:
  - truly cross-platform
  - easy to use, integrated design environment
  - written in C++
- Prepared the work packages
- Created official fund raising web site through *CERN Technology Transfer*
- Hired a talented C++ programmer
Our goals

- Kicad on par with mid-range proprietary tools
- Be able to design most of our PCBs solely in Kicad
- Make a truly multiplatform EDA suite
- Support OH community with quality libraries
Work packages

- Geometry library
- New graphics engine
- MCV architecture
- Push & Shove router
- Smart Drag & Drop
- Tool Framework
- Page templates
- Modularization
- Nice library browser

- GUI redesign
- Improved DRC
- Fwd/Back-annotation
- Pin/part swapping
- More PCB layers
- Property editor
- Net attributes
- Simulation support
- High speed routing

- Split power planes
- Mechanical layers
- Dimensioning
- STEP/IGES support
- File format plugins
- Python scripting
- Code cleanup
- Reusable IP blocks
- *Mr. Paperclip* 😊
New graphics engine

- Modern OpenGL-based (VBO, shaders)
- Clean view abstraction (MVC)
- Fast rendering of large PCBs on entry-level graphics cards
- Goodies: transparency, animations, interactive highlighting
- Cairo backend for software fallback and printing
New graphics engine

12-layer WR Switch v3 PCB
Geometry library

- A tiny 2D geometry library
- Mostly integer math, fixed rounding error
- Provides basic shapes used by the router: rects, circles, segments, polylines
- Fast spatial indexing & collision search (R-trees)
Tool framework

- Simple API for developing interactive tools
- Coroutine or state-machine based
- Internal event engine that behaves consistently across all supported platforms
- First tools available: Smart Drag & Drop and Push & Shove Router
Smart Drag & Drop

- Basic tool for item selection and edition (move, rotate, flip, delete, copy, paste)
- Single/multiple item editing
- Easy to use: look and feel as in typical graphics applications
- Smart selection algorithm to avoid asking for clarification in dense PCB areas
Push & Shove Router

- A must-have in any PCB program
- Based on original research (no literature!)
- Initial version available: pushes and sometimes shoves, fine-tuning needed
- More features coming soon (via plowing, dragging, less intrusive optimization)
- To be integrated with the Drag & Drop tool
Push & Shove Router

T. Włostowski

ZIO: The Ultimate Linux I/O Framework

What's new in Kicad?
Future plans

- Improve the P&S and D&D tools
- Port remaining pcbnew tools to new framework
- MCV-ize and cleanup Kicad schematic editor
- Increase number of layers
- Split code base into DLLs
- Ensure high quality on all OS platforms
- Take care of management & planning
- *Fix all bugs in the above...*
Summary

- Kicad becoming useful even for large projects
- More developers are welcome
- Hackathon planned in Q1 2014
- More cool stuff from CERN to come soon!

Check out `lp:kicad/kicad-cern-testing`

*Time for a demo!*