

KICAD EDA SOFTWARE SUITE

KiCad is an EDA (Electronic Design Automation) software suite for the creation of professional schematics and printed circuit boards up to 32 copper layers with additional technical layers. KiCad runs on Windows, Linux and Apple OS X and is released under open-source licence. KiCad is a mature EDA software tool under continuous development. It has a core development team and a dynamic and growing user community contributing regularly.

FEATURES

- Rich set of open-source libraries including 3D Models.
- Three step approach in PCB design via independent interconnected modules.
- All KiCad files are in ASCII. Facilitates manual manipulation and scripting. No vendor-lock in.
- Extensive documentation.
- GPL v.2 licence.

AREA OF EXPERTISE

- Electronics
- ICT

IP STATUS

- Open Source Software

TECHNOLOGY READINESS LEVEL

- Mature.
- Under continuous development.

CONTACT PERSON

nick.ziogas@cern.ch

Find out more at:

kt.cern

APPLICATIONS

- Design of PCBs.
- Educational tool to teach real-life electronics.

