

Students: Michal Madura, Dmytro Babitskyi

Company: MinebeaMitsumi, Ing. Tibor Rovenský, PhD.

Modern electronic devices require printed circuit boards (PCBs). To design PCBs, engineers use CAD/EDA software tools. Today, many different tools exist, and each one has different functions, prices, learning difficulty and workflow.

Because of this, students, hobbyists and professionals often face a problem: **Which CAD tool should they choose for their project?**

The goal of our project is to compare several popular CAD tools for electronic design and show their advantages, disadvantages and typical use cases.

Solution Architecture



Usage & Target Users

The compared CAD tools are commonly used in:

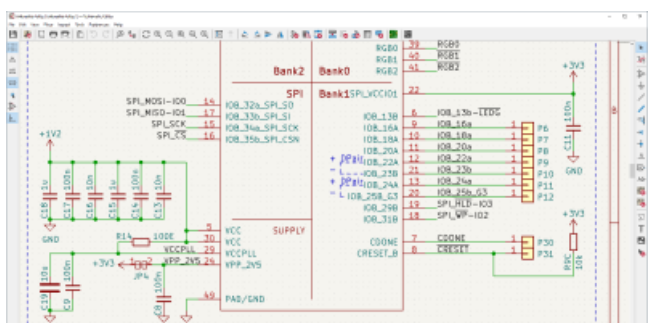
- University education (students learning PCB design)
- Hobby prototyping (makers and DIY electronics)
- Industry (professional PCB development)

The workflow typically includes:

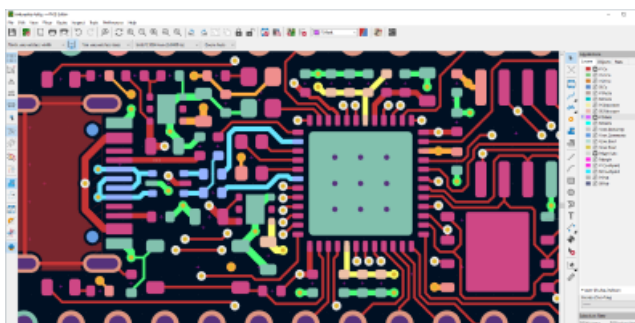
1. Schematic capture
2. PCB layout and routing
3. 3D visualization
4. Manufacturing documentation
5. PCB fabrication

Features & Capabilities

Feature	Altium Designer	KiCad	Autodesk Eagle
Schematic Capture	✓	✓	✓
PCB Layout	✓	✓	✓
3D View	✓	✓	✓ (Fusion 360)
Autorouting	✓	Limited	Limited
Library Management	Advanced	Manual	Basic
Simulation	✓	External	External
Licensing	Paid	Free	Subscription



Schematic Capture



PCB Layout / Routing



3D Board View

Evaluation & Conclusion

Altium Designer is best for professional and complex PCB projects.

KiCad is free, open-source and well suited for students and hobby users.

Eagle fits small projects and Fusion 360 users.

Choice depends on budget and workflow requirements.