One Tool, One Journey

How KiCad Builds Professional Skills AND Engineering Mindsets

Why the tool you learn first should be the tool you use forever

Peter Dalmaris

Author: "KiCad Like a Pro" & "Maker Education Revolution"

Five Days to Proficiency

Eagle CAD
Years of experience

→

5 Days
Learning KiCad

→

1 Year Later
Primary tool, no regrets

From **licensing anxiety**
to **tool ownership**

What if you never had to switch?

What if the tool you **learned on**...

...was the tool you'd **use for life**?

One Tool, One Journey

1. **Professional Skills:** No relearning required
2. **Engineering Mindset:** What open source teaches
3. **Maker Impact:** Real products, real companies
4. **The Ecosystem:** Everyone using the same tool
5. **Growing Pains:** Honest assessment

The Traditional Learning Path: Tool Ladder

Learning Debt:

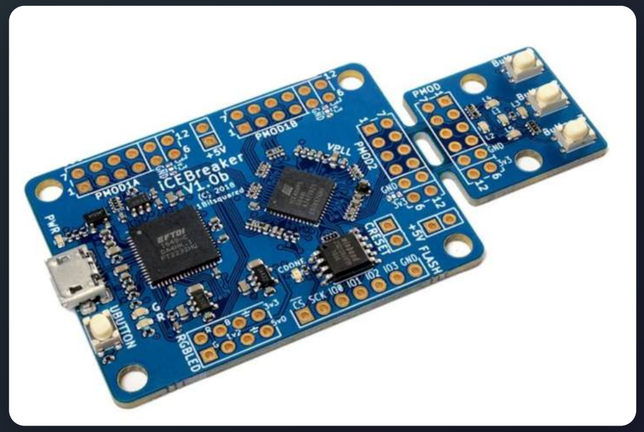
Time lost at each transition

One Tool: Concept to Product

Piotr Esden-Tempski

1BitSquared

- Dec 2017: Discussion with Clifford Wolf at CCC
- Jan 2018: First design in KiCad
- 5 hardware revisions
- All in KiCad



Same Files, Three Purposes

📚 Nov 2018: **Educational** workshop (Hackaday Supercon)

🚀 Dec 2018: **Commercial** launch (Crowd Supply)

🌍 Today: Used worldwide in **education & professional** FPGA development

Same KiCad files throughout

Purism

Privacy-Focused Hardware Manufacturer

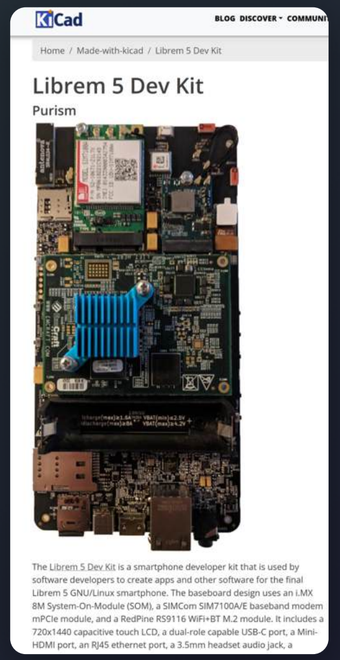
100% Free Software Commitment

KiCad used to design the Librem 5 smartphone base board

Website: puri.sm

Hardware Repository: source.puri.sm/Librem5

Why KiCad? Freedom and Capability



Purism's KiCad Journey:

- Chose KiCad over proprietary tools
- GNU GPLv3+ license alignment
- Librem 5 smartphone (commercial product)
- Development kit designed from scratch
- All designs released under free licenses

"KiCad outperforms several pricey proprietary alternatives"

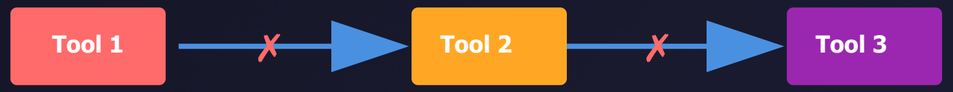
Maker Giant Uses KiCad

- ✓ Actively maintains KiCad libraries
- ✓ Publishes designs in KiCad format
- ✓ Contributes to ecosystem

When you learn KiCad,
you learn **SparkFun's tool**

One Journey: No Learning Debt

Traditional Path:



KiCad Path:

Your learning compounds forever

Professional Portfolio from Day One

- ✓ Senior project = Professional portfolio
- ✓ Skills transfer directly to jobs
- ✓ Files open forever (no license expiration)
- ✓ Knowledge you own, not license

The World's Leading Physics Lab Chose Open Source

2013: CERN invests in KiCad development

assigned developers to contribute

1,400+ hours contributed

Why?

Use ← → arrows or Space to navigate

Knowledge Without Barriers

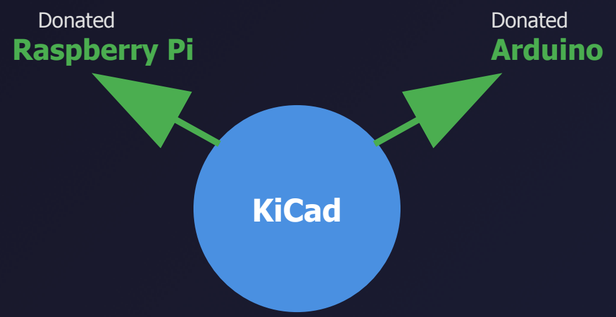
> *"KiCad can do to PCB design what GCC did to software: ensure there are no artificial barriers to sharing"*
>
> — Javier Serrano, CERN

What CERN Built

- Push-and-shove router
- Differential pair routing
- Integrated SPICE simulation

Advanced features now available to **everyone**

Who Else Invested?



Supporting the ecosystem that supports them

What Students Learn (Beyond PCB Design)

Commercial Tool Issue:

Submit ticket → Wait

Learn: Dependence

KiCad Issue:

Search forums → Read docs → Ask community → File bug → See fix

Learn: Engineering Agency

A Student's Perspective

> *"In Eagle, when something didn't work, I just assumed I was doing it wrong. In KiCad, I could look at the code and see if it was a bug or my mistake. That changed how I think about software entirely."*
>
> — Engineering Student

This mindset **transfers**

Engineering Agency in Action

A teacher using KiCad in his electronics classes:

- Built the same board from my course
- Added interfaces for his teaching needs
- Encountered problems with the original hardware
- Debugged, traced the schematic, identified issues
- Found workarounds, shared with his students, and documented them

Then he wrote to share his findings—

contributing knowledge back

His students learn from watching him troubleshoot real problems

The Maker Movement Impact

Makers aren't just learning

They're building **companies**

Creating **products that matter**

Changing **industries**

MNT Reform: From Frustration to Product

Lukas Hartmann

MNT Research, Berlin

- 2017: Started design out of frustration with closed hardware
- **7 different PCBs** - all in KiCad
- All open source (CERN-OHL)

MNT Today: Thriving Company

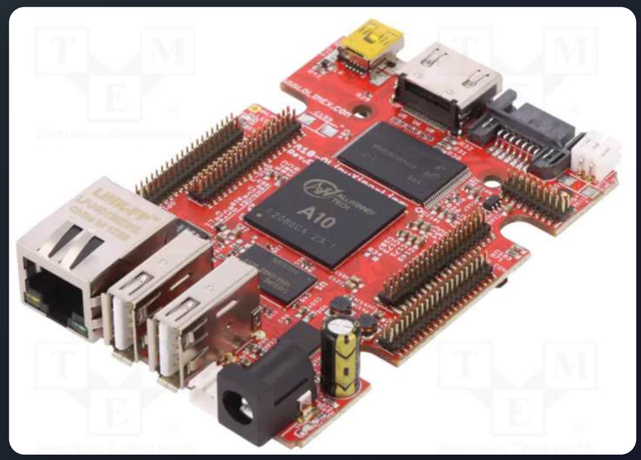
- ✓ Thousands of Reform laptops shipped worldwide
- ✓ Pocket Reform (mini laptop) launched
- ✓ Reform Next (8-core Rockchip)
- ✓ Used in universities for teaching electronics

From Maker frustration
to **commercial success**
to **educational tool**

Olimex: Industrial-Grade Open Hardware

Olimex, Bulgaria

- Switched from Eagle to KiCad in 2016
- Never looked back
- OLinuXino family: Industrial Linux SBCs
- Rated -40°C to +85°C



Real Products, Real Commitment

- 💪 Used in factories, automation systems, harsh environments
- 📚 Students can buy for €24 and study the KiCad files
- ♾️ Guaranteed longevity - manufacture as long as there's demand
- 🤝 Agreements with chip manufacturers for long-term supply

Only possible because KiCad is

open and permanent

"Made with KiCad" Showcase

Hundreds of projects:

- 📹 DIY thermal cameras
- 🤖 Robot controllers
- 📡 IoT devices
- ⚡ Motor drivers
- 🎓 Educational platforms

All started with Makers who had an idea

and a **free, professional tool**

The Ecosystem Effect

Something remarkable is happening:

Universities

Commercial companies

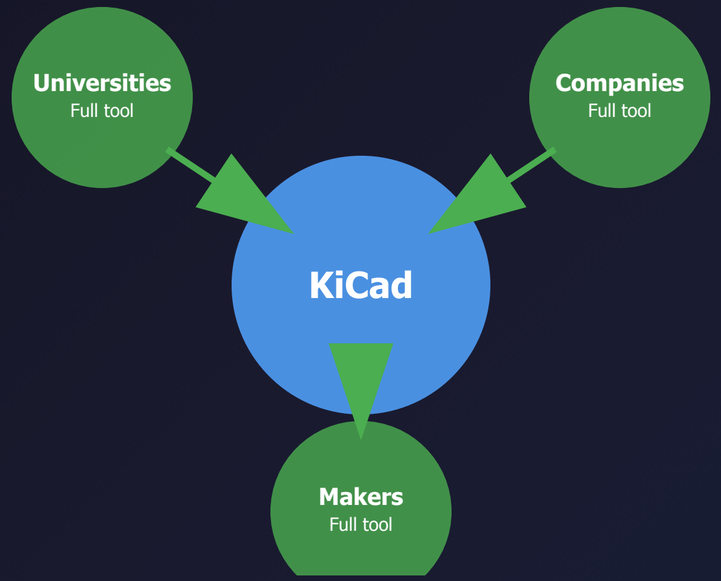
Maker community

All using the **same tool**

Traditional Software Ecosystem

Everyone speaks a different language

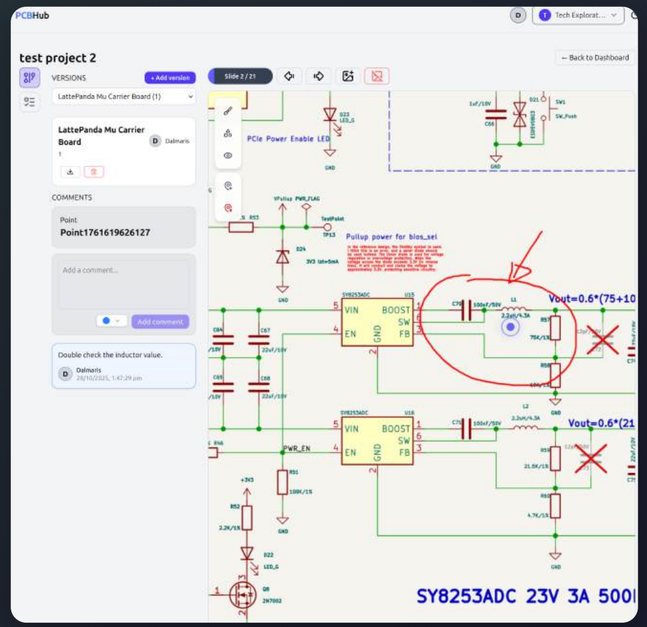
KiCad Ecosystem Convergence



The Portfolio Effect

- 💼 Show actual design files in job applications
- 🔍 Employers can open and explore your designs
- 📈 Track your growth over time
- 🌍 Collaborate globally without license barriers
- 💰 No $5,000 license needed to view your work

pcbhub.org — Currently in development



- 📤 Upload KiCad projects to your account
- 🤝 Share with collaborators worldwide
- ✏️ Annotate schematics and PCB layouts directly
- 💬 Discuss designs where they matter

Because KiCad files are **open and well-documented**,
the community can build collaborative platforms

No special licensing agreements needed

Active, Growing Ecosystem

KiCad 9 (February 2025)

📊 4,870 commits

👥 Hundreds of contributors

🔧 1,500 new symbols

📐 750 new footprints

📚 Library team doubled in size

Development shows no signs of slowing

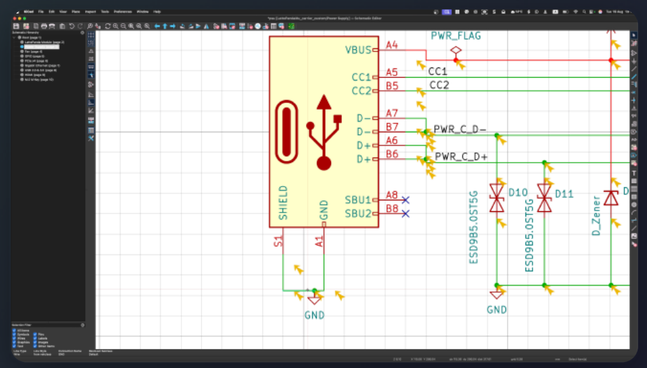
Let's Be Honest: The Growing Pains

KiCad isn't perfect

Pretending it is would do you a disservice

Let's talk about what doesn't work well **yet**

Problem #1: Grid Misalignment



⚠️ The Problem:

- Grid appears to work correctly
- Components seem aligned
- Later: discover hundreds of off-grid errors
- Devastating for students

Status: Being improved in KiCad 9

Problem #2: SPICE Simulator

- 📚 Requires learning SPICE syntax (cryptic, unforgiving)
- 💥 Frequent crashes and cryptic error messages
- 🔧 SPICE model compatibility issues
- 📉 Steep learning curve, unclear payoff

Reality: Most educators use external tools

It's improving, but not yet ready for easy education use

Problem #3: Library Challenges

Common student frustrations:

- "Where's this common part?"
- "How do I link this footprint?"
- "Why can't I find an Arduino?"

✓ Good news:

Library team doubled, adding 1,500 symbols + 750 footprints

Problem #4: Advanced Design Limits

- **RF & High-Speed:** Missing specialized tools, more manual work needed
- **HDI/Microvia:** Limited to outer-to-adjacent-inner layers only
- **Rigid-Flex:** Possible via workarounds, not natively supported
- **20+ Layer Boards:** Stackup management less sophisticated

For cutting-edge specialized work, commercial tools may still be needed

Educational Impact

These problems hit **students hardest**

Experienced engineers can work around issues

Beginners don't have that context

But Here's the Crucial Difference

Commercial Tools:

Submit ticket → Hope

Can't see issue tracker

Can't see the code

Can't contribute

Just... wait

KiCad:

Report on GitLab → See tracking

Problems openly discussed

Solutions are public

Can contribute fixes

Learning professional workflows

Every Tool Has Limitations

The question isn't

"Is the tool perfect?"

The question is:

"What do you learn from that experience?"

Two Different Lessons

Commercial Tools

↓

Learn dependence

KiCad

↓

Learn agency

A Vision: Learner Mode

What would make KiCad even better for education?

- 🎯 Simplified interface for beginners
- ⚡ Same powerful engine underneath
- 📚 Guided workflows for common tasks
- 💡 Built-in hints and tooltips
- 🎓 Clear path to "graduate" to full interface

Keeping the "One Journey" Promise

If we lose **accessibility**

in pursuit of **advanced features**

We break the promise

Students will start elsewhere again,
creating learning debt

To Educators

- 📚 Teach KiCad, but teach critically
- ⚠️ Acknowledge its limits
- 🔧 Turn rough edges into learning opportunities
- 💭 Advocate for features like Learner Mode

To Students & Early-Career Engineers

- 📝 Document your frustrations
- 🪲 File issues on GitLab
- 💬 Ask questions in forums
- 🔨 You're not just learning - you're improving the tool

We're All In This Together

When **CERN and others** improves KiCad,

everyone benefits

When **students** report bugs,

everyone benefits

When **makers** create libraries,

everyone benefits

One Tool.
One Journey.
One Community.

Thank you

Thank You!

Peter Dalmaris

techexplorations.com

Questions?