Welcome to the most boring KiCad talk ever: How I Switched From Eagle To KiCad With No Surprises Whatsoever

Also lets make an art badge if there’s still time

Let's commiserate together over licensing models

2016: EAGLE contracts a terminal case of being acquired by Autodesk, which promises to continue perpetual licensing.

2017: and switches to a subscription-only model.

2020: and stand-alone is discontinued. Long live the Fusion Electronics bundle. Buy hey, don’t worry, you can still use your special EAGLE subscription.


I AM ALTERING THE DEAL
AUTODESK
PRAY I DO NOT ALTER IT ANY FURTHER
made on imgur

The new deal

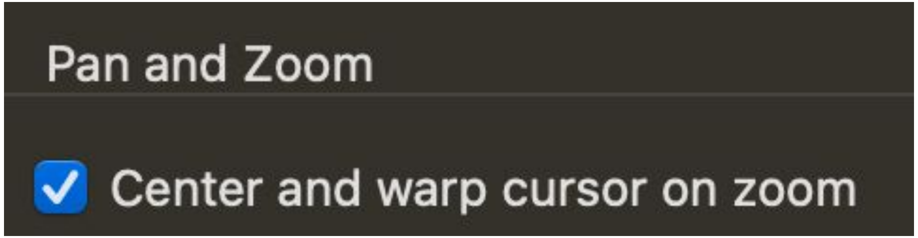
2023: EAGLE will be unavailable somewhere between 2024-2026 depending on your subscription. But don’t worry, you can still open those files with a new Fusion license with is only $680 a year (or whatever it is by then).

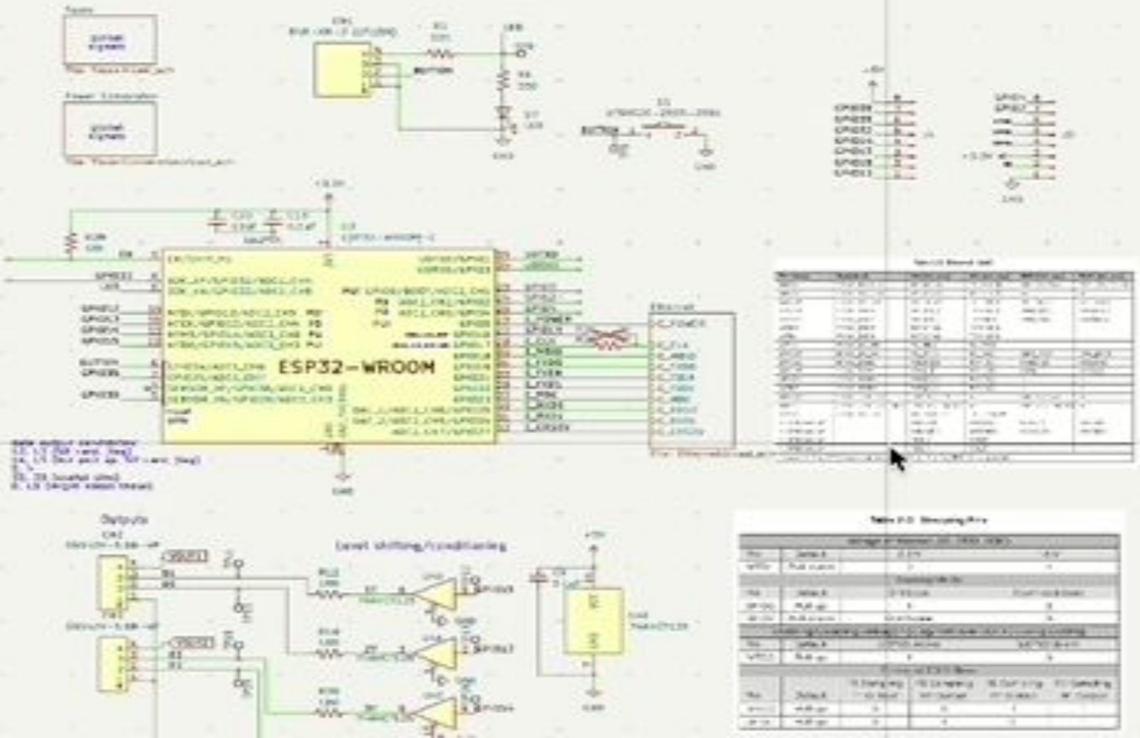
But not everything has to be a slow cloud-based subscription model bloatware!

KiCad is better than I thought it could be.

KiCad has gone from usable to excellent in recent years.

- Sane/decent line/vector handling, grid tools, alignment, images, Python!
- Much easier to discover features than Eagle.
- Default parts library is actually ***good***.
- Learning it was easier and more enjoyable than I thought possible
- Still has one major flaw that will bite every new user and make you question your sanity. This setting defaults to enabled:




ESP32-WROOM
Outputs
Level shifting/conditioning

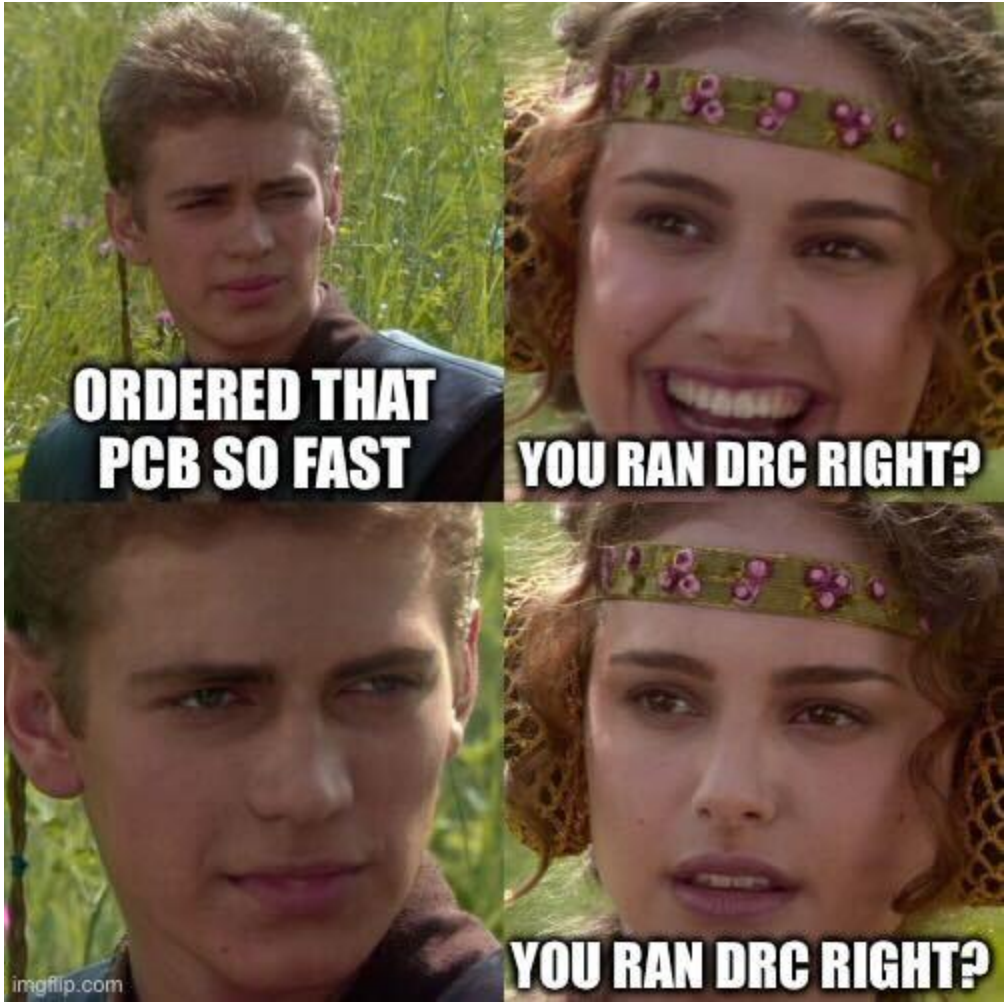
The hardest part of learning KiCad: How to pronounce KiCad.

Like KDE team and Apple got together and made a CAD child. Unless you wrote KiCad, in which case it's KeeCAD


Jokes about pronouncing GIF
Jokes about pronouncing KiCad

This workflow I spent two weeks on is better than what I had for my 10+ years of eagle experience

1. Use KiCad library for 90%
2. Find parts on LCSC
3. Import with easyeda2kicad
4. Draw many lines
5. Import graphics directly or use gingerbread.wntr.dev
6. Export to JLC with Fabrication Toolkit
7. Order PCBs
8. Remember to run Design Rules Checker to find any potential issues


ORDERED THAT PCB SO FAST
YOU RAN DRC RIGHT?
imgflip.com
YOU RAN DRC RIGHT?

Necromancy for EEs: You can import an eagle board but its way more fun to kill it and make a new one from the bones of the old version.

- KiCad parts are better
- 3D step files exported are better, even if you use Fusion 360

Anyone can make a PCB

(KiCading intensifies)