

AI-Powered PCB Inspector

Comprehensive PCB inspection and management platform with AI analysis, design rule checking, and component lifecycle tracking. Supports Gerber, KiCad, Altium, and Eagle formats with interactive layer viewing and BOM cost estimation.

Key Features

- Gerber, KiCad, Altium, Eagle file parsing with interactive layer viewer
- Design Rule Check (DRC) for trace width, clearance, and via rules
- Component library with lifecycle tracking (active / NRND / EOL)
- BOM cost estimation with supplier pricing integration
- **Board revision diff** — side-by-side comparison of added/removed components

Technology Stack

Gerber Parser

DRC Engine

FastAPI

React + TypeScript



PCB design errors and component obsolescence escaping review →

AI-checked before production

Live Demo: <https://pcbok.vercel.app>

Tools to Nought Error — Engineering precision into every system.