

# Kicad Activity Report

by ohloh.net

## Project Summary

Kicad is an open source (GPL) software for the creation of electronic schematic diagrams and printed circuit board artwork.

### In a Nutshell, Kicad...

... has had 5,198 commits made by 122 contributors representing 521,358 lines of code

... is mostly written in C++ with an average number of source code comments

... has a well established, mature codebase maintained by a very large development team with stable Y-O-Y commits

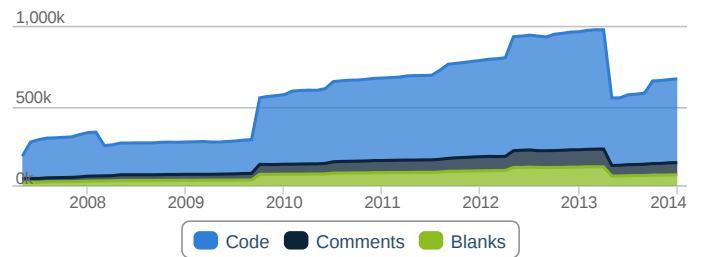
... took an estimated 138 years of effort (COCOMO model) starting with its first commit in May, 2007 ending with its most recent commit about 1 month ago

### Languages



C++	60%	XML	26%
C	12%	11 Other	2%

### Lines of Code



## Activity

### 30 Day Summary

Dec 20 2013 — Jan 19 2014

71 Commits

12 Contributors

including 2 new contributors

### 12 Month Summary

Jan 19 2013 — Jan 19 2014

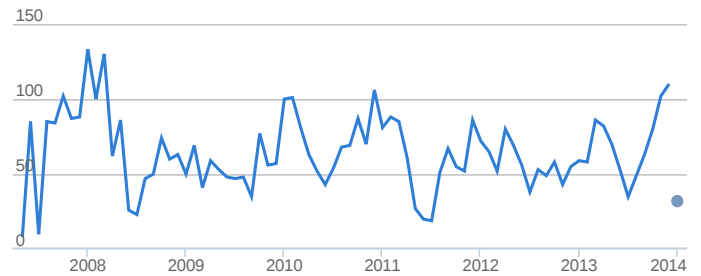
834 Commits

Up +148 (21%) from previous 12 months

46 Contributors

Up +9 (24%) from previous 12 months

### Commits per Month



## Community

### Ratings

7 users rate this project:

4.3/5.0

### Most Recent Contributors



Dick Hollenbeck



Wayne Stambaugh



...pierre charras



Brian Sidebotham



Maciej Sumiński



Cirilo Bernardo

### Contributors per Month

