Kicad

Kicad is the software we use for designing PCBs.

Installation

For linux, just run

```
sudo add-apt-repository --yes ppa:js-reynaud/kicad-4
sudo aptitude update
sudo aptitude install kicad kicad-library
```

For Windows,you can download it here.

Introduction

Below is a video with a short introduction to KiCAD.

Video

Common Issues

On linux, when you try to open up stuff, it may say you're missing a ton of libraries. To fix this, launch EESchema, then go to “Preferences→Component Libraries”. Add \`/usr/share/kicad/library\` to the user defined search path.