

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.

From:
<https://robosub-vm.eecs.wsu.edu/wiki/> - **Palouse RoboSub Technical Documentation**

Permanent link:
<https://robosub-vm.eecs.wsu.edu/wiki/ee/kicad/start>

Last update: **2016/12/05 10:42**



