UW-Madison Chemistry
Introduction to KiCad

Blaise Thompson

2019–04–13

1   electronic computer-aided design

Various options, including:

- → EAGLE (part of autodesk family)
- → ExpressPCB
- → KiCAD
- → Altium
- → Cadence OrCAD

Prefer KiCad.

Most ECAD programs have two main interfaces to the circuit:

1. The schematic
2. The PCB

2   introduction to KiCad

download from http://kicad-pcb.org/download/

When you first launch KiCad, you will be met with a totally blank application.

KiCad is actually made up of several sub-programs. You will see these listed along the top of the page. They include:

  → schematic layout editor
  → symbol library editor
  → PCB layout editor
  → footprint library editor
  → gerber viewer
  → import bitmap
  → calculator
  → worksheet layout editor

2.1   creating a project

Before we create our first project, a word about staying organized. A KiCad project is not a single file. Instead, a KiCad project is a *folder* with different files corresponding to different pieces of your project, like the schematic and (when appropriate) PCB. Furthermore, you will find yourself using and modifying *libraries* of electronic footprints and symbols—these will also become part of your project. It is imperative that the contents of this folder remain organized, or you risk "breaking" your project.

Use **File → New → Project** to create a new project.

3 schematic capture using eeschema

4 pcb layout using pcbnew