

# KiCad Tools Collection

On this site you can find many tools, which are written by KiCad users to improve the workflow with KiCad.

## KiPart

Create uniform schematic symbols from a CSV file.

Due to a missing property editor in KiCad, this seems to be a good workaround! Supports pin-type, side (left/right), automatic pin-sort, units (for large devices) and many more

<https://github.com/xesscorp/KiPart>

## KiCost

Collects prices based on manufacturer part numbers from the webstores of Digi-Key, Mouser and Farnell (Element 14).

<https://github.com/xesscorp/KiCost>

## svn2mod

create your logo design in Inkscape and convert it then to a KiCad module, so you can place it on your PCB.

## kicad2csvbom

simple c-programm that extracts a BOM out of a netlist.

This is written by me, an should perhaps be converted into a python script for the BOM generator in EESchema

<https://github.com/KarlZeilhofer/kicad2csvbom>

## footprint wizard

The wizard is built into KiCad's PCBnew (or more precisely the footprint editor)

- plugin for the wizard: [https://github.com/xesscorp/xess\\_fp\\_wizard](https://github.com/xesscorp/xess_fp_wizard)

## KiCad Librarian

manage, compare and modify symbol and footprint libraries.

[http://www.compuphase.com/electronics/kicadlibrarian\\_en.htm](http://www.compuphase.com/electronics/kicadlibrarian_en.htm)

## VisualPlace

Assistance tool for manual pick and place

[http://www.compuphase.com/visualplace/visualplace\\_en.htm](http://www.compuphase.com/visualplace/visualplace_en.htm)

## KiCad Sheet Rearranger

Change the order of sheets in hierarchical schematics (only flat hierarchy is supported)

[kicad-manual-page-order](#)

## KiCad Partslist Editor

It's a Python tool with GUI for exporting and back importing various fields of hierarchical schematics to and from CSV.

[https://github.com/KarlZeilhofer/KiCAD\\_Partslist\\_editor](https://github.com/KarlZeilhofer/KiCAD_Partslist_editor)

From:

<http://www.zeilhofer.co.at/wiki/> - **Verschiedenste Artikel von Karl Zeilhofer**

Permanent link:

[http://www.zeilhofer.co.at/wiki/doku.php?id=kicad\\_tools\\_collection&rev=1485701765](http://www.zeilhofer.co.at/wiki/doku.php?id=kicad_tools_collection&rev=1485701765)

Last update: **2017/01/29 15:56**

