KiCad Tools Collection

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

KiCad Partslist Editor

It's a Phython tool with GUI for exporting and back importing various fields of hierarchical schematics to and from CSV.
https://github.com/KarlZeilhofer/KiCAD_Partslist_editor

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

KiCad Librarian

manage, compare and modify symbol and footprint libraries.
http://www.compuphase.com/electronics/kicadlibrarian_en.htm

VisualPlace

Assistance tool for manual pick and place
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