KiCad Tools Collection

On this site you can find many tools, which are written by KiCad users to overcome the short comings of KiCads user interface.

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

KiCad Part Library Editor

It's a Phython tool with GUI for exporting and back importing various fields of hierarchical schematics to and from CSV. At the date of writing, it couldn't handle my schematics on linux.
https://github.com/BPJWES/KiCAD_Partslist_editor