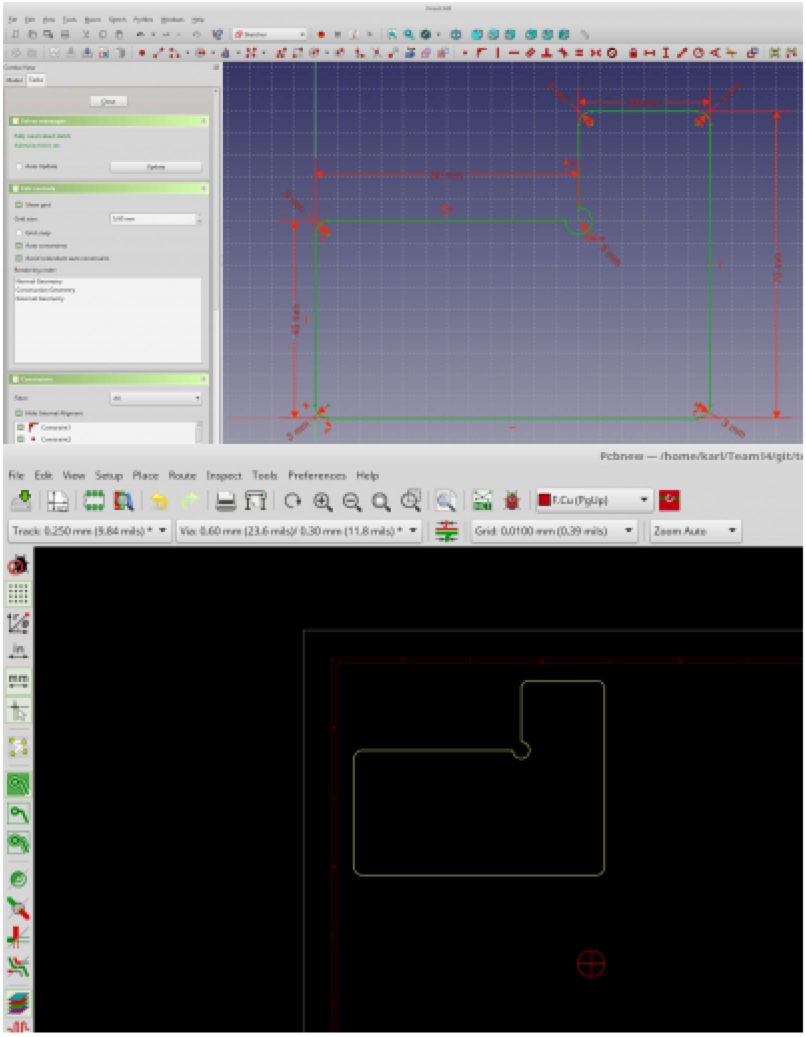
KiCad Tools Collection

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

FreeCAD for Complex PCB Outlines

It's very easy to import a DXF file, that was created with FreeCAD's sketcher:



Replicate Layout

Implemnts a repetitive place and route within PCBnew, when the schematic is based on identical hierarchical sheets:
https://github.com/MitjaNemec/Kicad_action_plugins

Interactive HTML BOM

https://hackaday.com/2018/09/04/interactive-kicad-boms-make-hand-assembly-a-breeze/
seems to be really great!

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

KiField

Similar to KiCad Partlist Editor, but seems to be maturer:
https://github.com/xesscorp/KiField

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

KiBoM

KiBoM is a configurable BOM (Bill of Materials) generation tool for KiCad EDA. Written in Python, it can be used directly with KiCad software without the need for any external libraries or plugins.
KiBoM intelligently groups components based on multiple factors, and can generate BoM files in multiple output formats.
BoM options are user-configurable in a per-project configuration file.
https://github.com/SchrodingersGat/KiBoM

svg2mod

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

Altium to KiCad Converter

This tool is available as an online service:
http://www2.futureware.at/KiCad/

or on GitHub: https://github.com/thesourcerer8/altium2kicad

kicadLibCreator

Merges existing symbols with metadata and footprint from Octopart.com

https://github.com/pioupus/kicadLibCreator

Action Plugins for KiCad 5+

- Tear Drops - add polygons to implement beautiful teadrop pad connections
- Layer View Set - manage layer sets in PCBnew

Tool-Collections of other People

* from Pointhi

english, software, kicad, collection, tools, linux, github