KiCad

KiCad is a EDA for designing PCBs. It is free, cross-platform and open source!

Getting startet with KiCad: External.

Some things that might be good to know (a V4)

1. BOM generation
   Take a look at this (https://github.com/SchrodingersGat/KiBoM)
2. Via stitching
   Take a look at this (https://forum.kicad.info/t/protip-nicer-via-stitching/1103)
3. Add default fields to the parts
   In the Eeschema go to Preferences-> Schematic Editor Options-> Template
   Add. Then write in what you want.

kicad-cheatsheet.pdf

silica.io/wp-content/uploads/2018/06/kicad-cheatsheet.pdf