Editor Needed !

You need a schematic editor to produce a netlist file for VeeCAD to import.

Default Schematic Editor

For a quick start, **download the TinyCAD schematic editor**, then draw your schematics using the special TinyCAD symbol libraries that are supplied with VeeCAD.

Alternative Schematic Editors

Most schematic editors can work with VeeCAD, provided you copy the basic VeeCAD library file and rename the outlines to match the footprints used in your schematic library.

VeeCAD works with these netlist formats:

- **Protel :** Produced by Altium, Protel, TinyCAD, Target3001, Multisim7, Rimu Schematic, Electronics Workbench, CircuitMaker, Eagle with the ULP script "netlist_protel.ulp" **from Autodesk** and many other CAD programs.
- **Tango :** Produced by Orcad Capture, DipTrace, Proteus-ISIS (not Proteus Lite) and many others.
- **Orcad PCB2 :** Produced by KiCAD, Pulsonix and others.
- **EasyPC-Generic :** Produced by Easy-PC and DesignSpark PCB. In EasyPC V11 or DesignSpark PCB V1 select "Output / Reports / Generic Netlist / Run", ensure "Include Components" is checked, then click "Generate".
- **UltiCap :** For UltiCap DOS and Ulticap Windows. Requires export of the Component Placement File also.
- **Seetrax :** Requires export of the Parts file also.
- **ZenitPCB :** Produced by ZenitPCB.

Contact the author if you require another format added to VeeCAD.

Free Editors

VeeCAD installs ready-to-run with the editors marked ** recommended **. While VeeCAD works with the other editors, you will have to make your own adapted VeeCAD outline libraries. Here are some **editor reviews.**

- TinyCAD Schematic Editor ** recommended **
  - Windows Open Source
  - Runs well under Wine.
  - VeeCAD installs ready to run TinyCAD libraries so you can start work immediately.
  - Decent multi-sheet schematic editor for Windows.
  - Simple interface, good looking schematics.
  - See the **TinyCAD Guide** for information.
  - **http://tinycad.sourceforge.net**
- DesignSparkPCB Schematic-PCB ** recommended **
  - Windows.
  - Non-crippled design tool from RS Components engineering community site.
  - Exports the Easy-PC Generic format which VeeCAD supports.
  - VeeCAD installs ready to run outline (footprint) libraries so you can start work immediately.
  - **http://www.designspark.com/**
- KiCad Schematic-PCB ** recommended **
  - Windows, Linux Open Source.
  - Hierarchical page schematic editor.

    - VeeCAD installs ready to run KiCad libraries so you can start work immediately.
    - Includes large library of well-drawn components.
    - Exports the Orcad PCB2 net format which VeeCAD supports.
    - **https://kicad-pcb.org/**
- BSch3V Schematic Editor
    - Windows Open Source.
    - Beautifully rendered and printed schematic symbols and text.
    - Netlist generator program produces Protel net format which VeeCAD supports.
    - Japanese version also.
    - Basic multi sheet facility.
    - **http://www.suigyodo.com/online/e/index.htm**
- DipTrace Schematic-PCB
    - Windows.
    - Freeware version for non-commercial use with full schematic functionality and PCB limited to 300 pins, plus commercial versions.
    - Exports the Tango net format which VeeCAD supports.
    - **http://www.diptrace.com/**
- Target 3001 Schematic-PCB
    - Windows.
    - Freeware version for non-commercial use with PCB limited to 250 pins, plus commercial versions.
    - Exports the Protel net format which VeeCAD supports.
    - **http://www.ibfriedrich.com/**

Choosing a Schematic Editor

- A schematic editor is a useful tool for drawing circuit diagrams, as well as for use with PCB or VeeCAD editors.
- Free or Lite editions of commercial editors are often licenced only for personal use or may not export a usable netlist.
- TinyCAD, DesignSparkPCB, KiCAD and BSch3V can be used commercially for free.
- Schematic editors in simulators like LTSpice, Livewire, Crocodile Physics etc cannot act as your general editor since they omit important component information and hit limitations such as single page, inflexible or fixed component libraries, limited or no netlist.
- KiCAD and Eagle have Linux versions. Users have run DipTrace under Linux/Wine. TinyCAD runs well under Wine.