

# Features of EAGLE

---

## System Requirements

EAGLE is a powerful graphics editor for designing PC-board layouts and schematics. In order to run EAGLE the following is required:

- ◆ Windows 2000, Windows XP, Windows Vista, or Windows 7,
- ◆ Linux based on kernel 2.x for Intel computers, libc6 and X11 with a minimum color depth of 8 bpp,
- ◆ Mac OS X version 10.4 or above (for Power PC or Intel computer),
- ◆ a harddisk with a minimum of 100 MByte free disc space,
- ◆ a minimum graphics resolution of 1024 x 768 pixels,
- ◆ preferably a 3-button mouse.

## Professional Edition

### General

- ◆ maximum drawing area 64 x 64 inches (about 1600 x 1600 mm)
- ◆ resolution 1/10.000 mm (0.1 microns)
- ◆ grid in mm or inch
- ◆ up to 255 drawing layers
- ◆ command (Script) files
- ◆ C-like User Language for data import and export
- ◆ easy library editing
- ◆ composition of self-defined libraries with already existing elements by Drag&Drop
- ◆ easy generation of new package variants from any library by Drag&Drop
- ◆ free rotation of package variants (0.1 degree steps)
- ◆ library browser with powerful search function
- ◆ support of technology feature (e.g. 74L00, 74LS00..)
- ◆ Easy definition of labelled drawing frames
- ◆ free definable attributes, applicable for Devices in the Libraries and in Schematic or Layout
- ◆ integrated PDF data export function
- ◆ export function for graphic files (BMP, TIF, PNG...)

- ◆ printouts via the OS's printer drivers with print preview
- ◆ partlist generation with database support (*bom.ulp*)
- ◆ Drag&Drop in the Control Panel
- ◆ context menu with object-specific commands for all objects, available through a right mouse click
- ◆ properties of objects can be accessed and edited via context menu
- ◆ automatic backup function

### **Layout Editor**

- ◆ full SMD support
- ◆ support of Blind and Buried vias
- ◆ rotation of objects in arbitrary angles (0.1-degree steps)
- ◆ components can be locked against moving
- ◆ texts can be placed in any orientation
- ◆ dynamic calculation of signal lines while routing the layout
- ◆ magnetic-pads function
- ◆ tracks can be drawn with rounded corners in any radius
- ◆ mitering to smooth wire joints
- ◆ Design Rule Check for board layouts (checks e.g. overlaps, measures of pads or tracks)
- ◆ copper pouring (ground plains)
- ◆ package variants support
- ◆ user-definable, free programmable User Language to generate data for mounting machines, test equipments, milling machines or any other data format
- ◆ output of manufacturing data for pen plotters, photo plotters and drilling machines with the CAM Processor

### **Schematic Editor**

- ◆ up to 999 sheets per schematic
- ◆ icon preview for sheets
- ◆ sorting sheets with Drag&Drop
- ◆ cross references for nets
- ◆ automatic generation of contact cross references
- ◆ simple copying of parts
- ◆ replace function for parts without loss of consistency between schematic and layout

- ◆ Online-Forward&Back Annotation between schematic and board
- ◆ automatic board generation
- ◆ automatic generation of supply signals
- ◆ Electrical Rule Check (error check in the Schematic and consistency check between Schematic and Layout)

### **Autrouter Module**

- ◆ fully integrated into basic program
- ◆ uses the layout's Design Rules
- ◆ change between manual and automatic routing at any time
- ◆ basic engine for the Follow-me router, a tool that supports you in manual routing; the trace of a selected signal will be calculated automatically
- ◆ ripup&retry algorithm
- ◆ user-definable strategy by cost factors
- ◆ routing grid down to 0.02 mm (about 0.8 mil)
- ◆ no placement restrictions
- ◆ up to 16 signal layers (with user definable preferred directions)
- ◆ up to 14 supply layers
- ◆ full support of Blind and Buried vias
- ◆ takes into consideration various net classes

### **Standard Edition**

The following restrictions apply to the Standard Edition:

- ◆ The layout area is restricted to a maximum of 160 x 100 mm (about 6.3 x 3.9 inches). Outside this area it is not possible to place packages and draw signals.
- ◆ A maximum number of 4 signal layers are allowed (Top, Route2, Route3, Route14, Route15, Bottom).
- ◆ The Schematic can have a maximum of 99 sheets.

### **Freemium Edition**

The Freemium Edition is a *Free Premium*, which is available only after registration on <http://www.element-14.com/eagle-freemium> and has the following limitations:

- ◆ The board area is restricted to 100 x 80 mm (about 3.9 x 3.2 inches), which corresponds to half of a Eurocard.
- ◆ Only 4 signal layers can be used (Top, Route2, Route15, Bottom).

- ◆ A schematic can consist of a maximum number of 4 sheets.
- ◆ The Freemium license is limited to one single user and computer, and requires an active connection to the Internet in order to work.
- ◆ The license expires 60 days after installation.

## Light Edition

The following restrictions apply to the EAGLE Light Edition:

- ◆ The board area is restricted to 100 x 80 mm (about 3.9 x 3.2 inches). Outside this area it is not possible to place packages and draw signals.
- ◆ Only two signal layers can be used (no inner layers).
- ◆ A schematic can consist of only one single sheet.

Larger Layout and Schematic files can be printed with the *smaller* editions.  
The CAM processor can generate manufacturing data as well.

*It is not possible to combine modules of different editions!  
The Light Edition is available as Freeware for testing, evaluation, and non-commercial use.*