

---

## Features of EAGLE Version 6.0

### System Requirements

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

- Windows XP, Windows Vista, or Windows 7
- Linux based on kernel 2.6 for Intel PC, X11 in at least 8 bpp mode, 32-bit runtime environment. Libraries: libpng14.so.14, libssl.so.1.0.0, libcrypto.so.1.0.0, libjpeg.so.8
- Mac OS X version 10.5 on Intel Computer
- A minimum graphics resolution of 1024 x 768 pixels
- Preferably a 3 button wheel mouse.

### Professional Edition

#### General

- 16 Layers, 4 x 4m (about 150 x 150 inch)
- Resolution 0.003215  $\mu\text{m}$  (micron)
- 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
- User defined attributes, applicable for Devices in the Libraries and in Schematic or Layout
- Dimensioning tool
- 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
- Designlink, a interface to a Global component supplier

---

## **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
- Differential pair routing
- Meander command for length compensation of signals
- Support of assembly variants
- 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 generation of supply connections
- Automatic board generation
- Electrical Rule Check (error check in the Schematic and consistency check between Schematic and Layout)
- User Defined Net Classes for Via Size, Wire Width and Clearance

## **Autorouter Module**

- Fully integrated into basic program
- Routing grid down to 0.02 mm (about 0.8 mil)
- 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
- No placement restrictions
- Uses the layout's Design Rules
- Change between manual and automatic routing at any time
- Up to 16 signal layers (with user definable preferred directions)
- 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 6 signal layers are allowed (Top, Route2, Route3, Route14, Route15, Bottom).
- The Schematic can have a maximum of 99 sheets.

## 30 days Free Trial Edition (Freemium Edition)

The Free Trial Edition is a *Free Premium*, which is available only after registration on <http://www.element14.com/eaglefreemium> 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 30 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 and does not support pasting from other schematic files.

**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 noncommercial use.*