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 µm (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 (0.1degree 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.