

# Buy now and get your upgrade for FREE!

**EAGLE**  
**V6**

THE NEXT GENERATION

## Coming Soon: CadSoft EAGLE Version 6

Features for unique flexibility and efficiency:



### **XML database structure redesign**

**Read, edit and parse the data outside of the EAGLE tool**

The new XML database structure provides the ability to write scripts that manipulate designs in the EAGLE format which will give users huge productivity benefits. You can make design changes, import information from other designs, and even translate from other formats. This is the first CAD database format to use ASCII text as a format and could become a standard in the industry for communicating designs – making EAGLE the most flexible, user friendly and productive PCB tool on the market.

### **Design reuse**

**Merge board/schematic pairs using  
the PASTE function with full consistency**

You can copy an already finished design consisting of a consistent pair of Schematic and Layout into another project. Enumeration of components and signals will be exactly the same in board and schematic, so that consistency between schematic and board will be maintained. This supports easy design, for example, of multi-channel devices. You only have to design schematic and layout for one channel and can copy it easily.

### **Routing capability enhancement**

**Optimize your work-flow with Undo/Redo log**

The UNDO/REDO log shows the history of each step you have made. You have full control of the steps going back or forward.

**Work with flawless conversions between mm  
and inches through increased internal resolution  
(allows grids of 1/4, 1/8, 1/16, 1/32 and 1/64 mil)**

If you have to work with both systems, metric and imperial, you know about problems that can arise in converting the usual imperial grid values (always divided by 2) and the resulting mm values. Increasing the internal resolution up to 1/64 mil will allow a proper conversion in all situations.

### **Library editor optimization**

**Connect a pin to multiple pads**

You can now connect one pin in the schematic symbol to any number of pads of the package. With this function EAGLE can handle components' internal connected pins. Compared to previous versions schematic symbols look nicer and you save time in defining them.

### **Draw arbitrary pad shapes easily**

Take a pad or an SMD as basis and draw the final contour with the help of the WIRE or POLYGON command. These objects will be considered as part of the pad/SMD. No more problems with Design Rules Check for such objects.

**Benefit from unlimited number of technologies  
and package variants**

Until now the maximum number of technologies and package variants was limited. In some cases one could reach the limits and had to split a library into two parts. No more limitation in EAGLE 6.

\*The upgrade to version 6 is limited to the purchased/  
updated editions and modules. For more information  
see [www.cadsoft.de/conditions\\_buy\\_now](http://www.cadsoft.de/conditions_buy_now)

Purchase your EAGLE license:

**element14**  
BE IN YOUR ELEMENT  
[www.element14.com](http://www.element14.com)



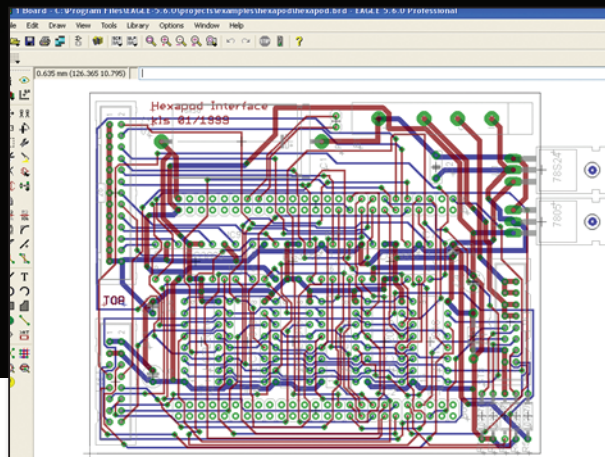
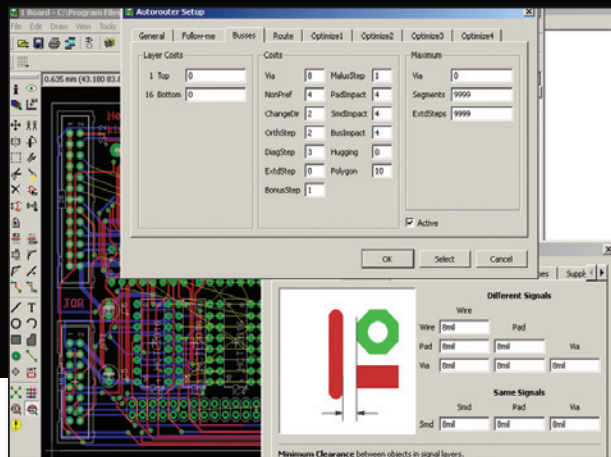
[www.farnell.com](http://www.farnell.com)

# CadSoft EAGLE Version 6

Features for unique flexibility and efficiency:

EAGLE

V6



## Additional features planned for V6

### Benefit from differential pair routing and automatic meanders

Special signals can be routed with exactly the same length. Differential pair routing is used for pairs of differential (equal and opposite) signals across the board to create a balanced transmission system. Creating meanders helps users automatically give a signal a certain length in the pcb. Meanders in traces are used to increase delay times in high speed digital circuits.

### Improve your efficiency through BGA escape routing (ULP)

BGA devices can have a 1000 connections or more. Packages are getting smaller and smaller. It is a hard and tedious work to connect all the pins, set vias, decide about the layers to use, and come out from the BGA area. This ULP helps you in escaping from the BGA in a proper way.

### GUI improvement: define your own context menu

Speed up your work-flow with self-defined content in the context menu. Frequently used commands or combinations of commands can be integrated into the context menu. This minimizes mouse-clicks and increases working speed.

### Be flexible with assembly variants

One schematic but a number of different assembly variants can be handled now.

### Justify your text to left, top, right, bottom, middle or center with one mouse click

This allows you to adjust texts as you know it from text editors. Gives a neat look and makes writing, for example, comments more comfortable.

## Layout editor add-ons

### Dimension your layout automatically

A dimensioning tool where you decide by a few mouse clicks the objects to be dimensioned.

### Create restricted areas with Cutout polygons

Outer and inner layers can now have individual restricted areas that will be avoided by signal polygons. Simply draw a polygon and define it as cutout.

## File description functionality

### Individualize your schematics, sheets, boards and symbols with descriptions

The file description can inform you about the content of a board or schematic. It is shown in the Control Panel without loading the file in the editor windows. This makes your project management clearer. See what this file contains without loading it in an editor window. This also works with sheets of multi-sheet schematics.

element14

BE IN YOUR ELEMENT

[www.element14.com](http://www.element14.com)

Purchase your EAGLE license:



[www.farnell.com](http://www.farnell.com)