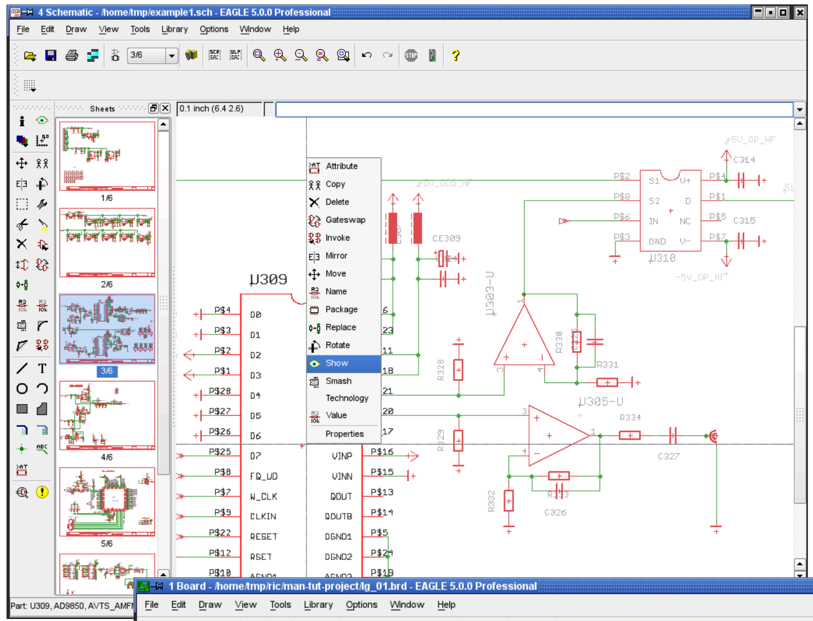
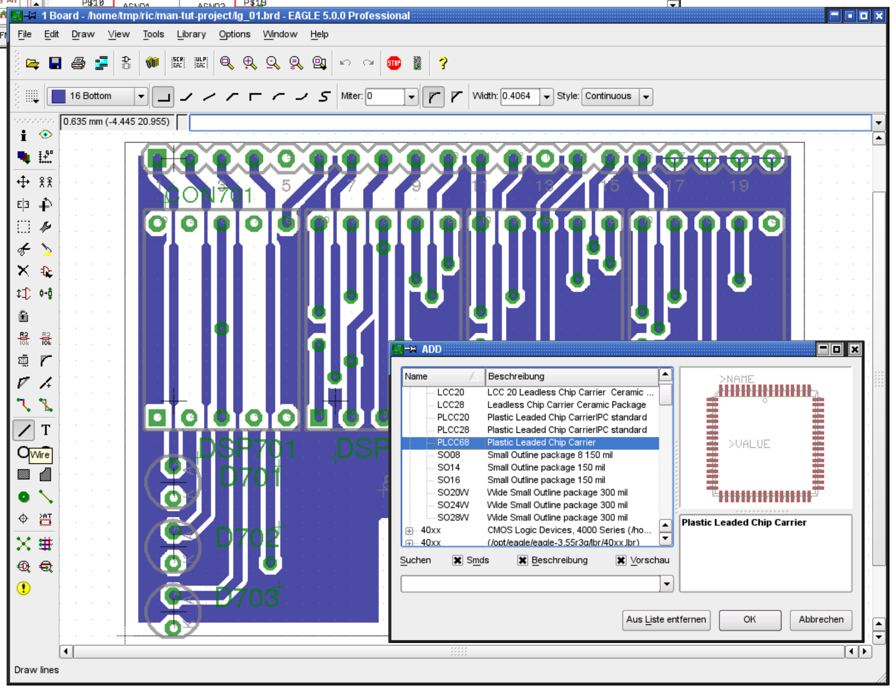
Three Modules - One Program





Schematic Editor

The Schematic Editor gives you more than just a circuit diagram. When the circuit is drawn, a large part of the layout work is already done!

The Schematic Editor is also availbale without the Layout Editor, for example for drawing electrical schematics.

Layout Editor

The Layout Editor allows you to design whole circuit boards manually. Define the board's dimensions, arrange the components on the board, and lay-out the traces. The Design Rule Check helps you to check your given rules. Manufacturing data are generated with the integrated CAM Processor.

Layout and Schematic Editor come with the Library Editor for component definition.

Autorouter

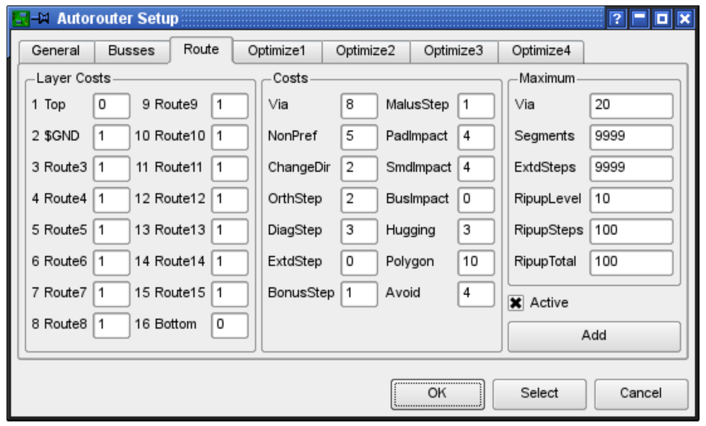
The integrated Autorouter takes a lot of routine work away from you. It handles boards of up to 16 layers. Smallest routing grid: 0.02 mm. You can apply the automatic routing to individual signals, to groups of signals, or to all signals that are still unrouted. Design rules and strategy parameters can be set by the user.

Technical Data

EAGLE Professional: max. drawing area 1.6 m x 1.6 m (64 in x 64 in) • 1/10000 mm resolution • 255 drawing layers • 16 signal layers • up to 999 sheets per schematic • sorting of schematic sheets • extensive libraries supplied • undo and redo function for any number of commands • replace command in schematic • graphical ERC • components on both sides • freely definable attributes for components • support of blind, buried, and micro vias • copper pouring • on-line Forward&Back annotation • on-line calculation of airwires when laying-out wires • wires can be drawn as arcs • selective hiding of airwires • extended net classes matrix for autorouter and DRC • round or rounded SMD pads • different pad shapes for top, bottom, or inner layers • output drivers for all standard photoplotters and drill formats • print preview • PDF export • execution of command files • integrated C-like User Language permits manipulation of EAGLE data for any other software and hardware (e. g. simulators, component insertion machines, ATE).

EAGLE Standard: As Professional, but routing is only possible on an area of 160 mm x 100 mm (6.3 x 4 inches), and in up to four signal layers. Larger boards and drawings outside this range can be loaded. Schematics with a maximum of 99 sheets.

EAGLE Light: As Standard, but the maximums are a routing area of 100 mm x 80 mm (4 x 3.2 inches), two signal layers and one schematic sheet.



CadSoft Computer, Inc.
19620 Pines Blvd., Pembroke Pines, FL 33029
Support (954) 237-09325, Fax (954) 237-0968
E-Mail : info@cadsoftusa.com
Web: http://www.cadsoftusa.com

Rely on 20 years of experience and success

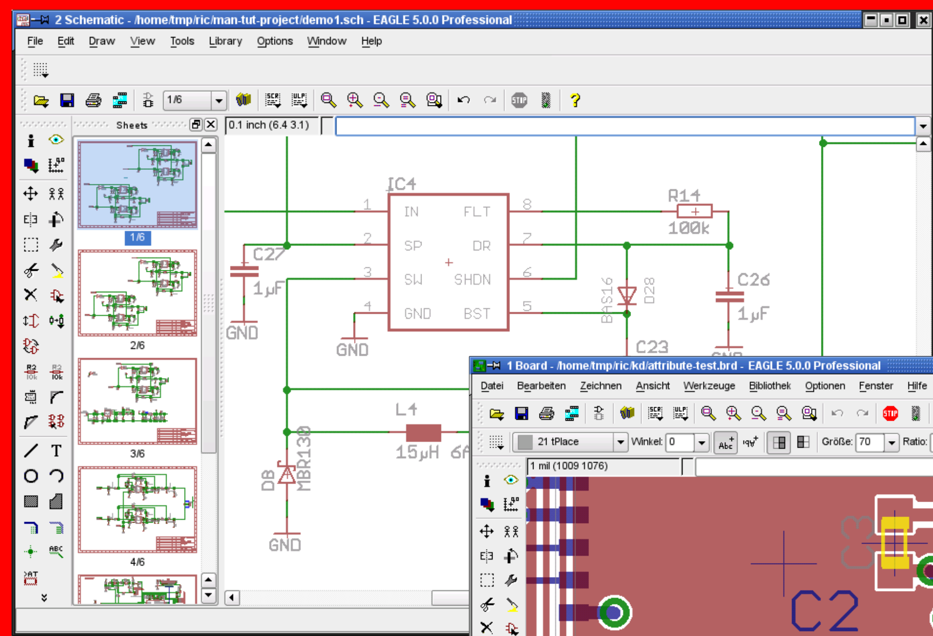
Download our Freeware for testing from www.cadsoftusa.com

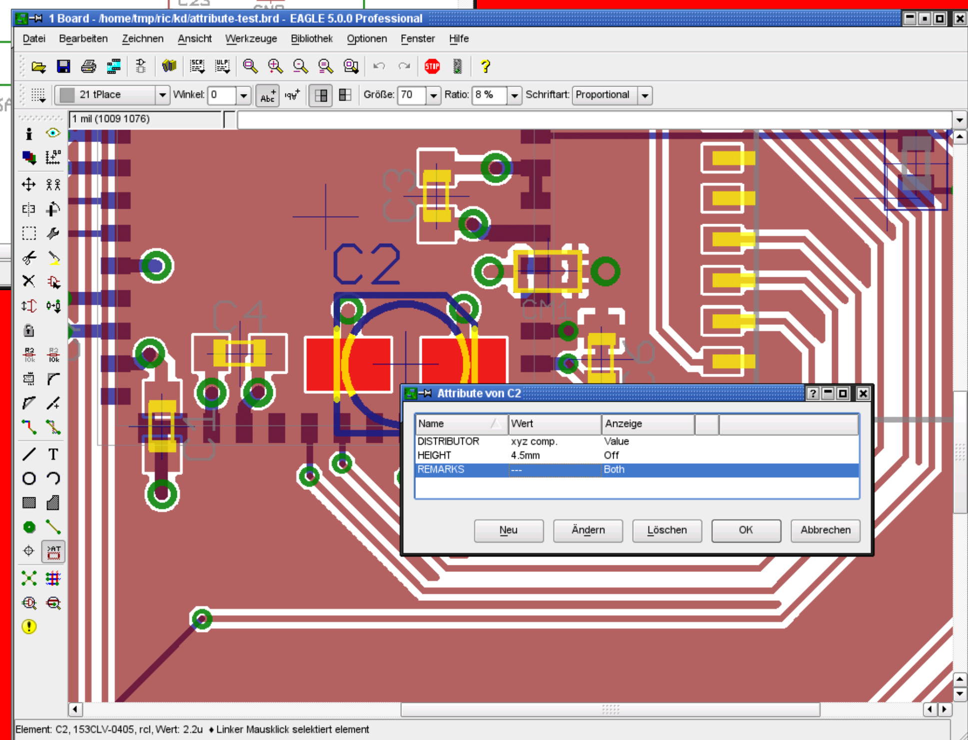
EAGLE

Version 5

Schematic • Layout • Autorouter

for
Windows®
Linux®
Mac®





Why have so many top companies chosen EAGLE as their PC-board design tool?

Quality without Compromise

With EAGLE you can create pcb films and documentation which satisfy the highest demands on quality. There is no need to compromise either quality or size.

Easy to Learn and Use

EAGLE is very easy to use, in spite of all its facilities. Do not underestimate the effect this can have on costs! Schematic, layout and component editors have identical user interfaces. A unique UNDO/REDO function, with which any previous state of the design can be reconstructed with a few keystrokes, is just one example of EAGLE's friendliness.

Continuity in Development

EAGLE is developed by CadSoft itself. Always in touch with our customer's needs. EAGLE 5, for example, now supports development of Electrical Schematics with basic functions like frame coordinates, or cross-references for nets and components. Therefore the Schematic Editor is available as a stand-alone module now.

Effective Customer Support

We are always ready to listen to our customers. This means you can talk to somebody about what you would like to see, and about suggestions for improvement.
And at CadSoft there is no hotline fee!

Try Before Buy

Check out EAGLE with our Light Edition which is freeware for test purposes and non-commercial projects. EAGLE Light allows you to design boards up to 4 x 3.2 inches. All other features are the same as in the Professional and Standard Edition. All of the component libraries are contained in EAGLE Light, as well.

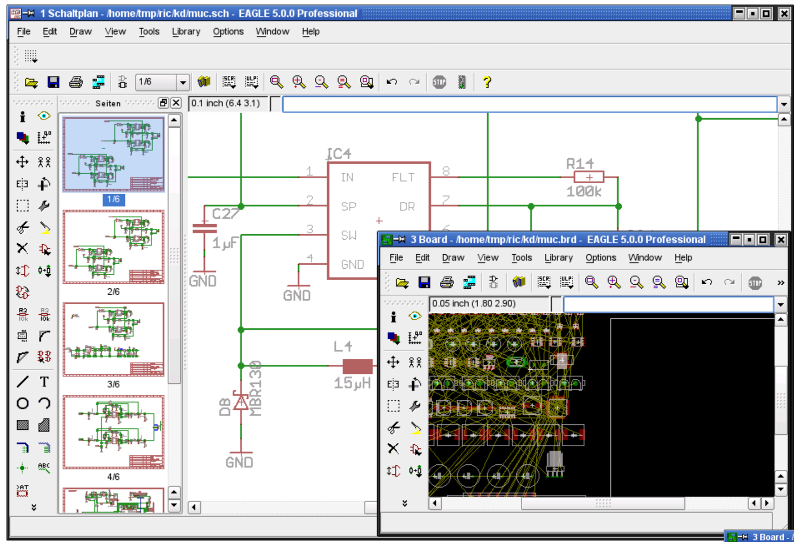
Talk to the Rest of the World

Closed solutions are out! Cooperation with other programs and the output of data to processing engines is becoming ever more important. The C-like User Language is able to arrange the input and output of data flexibly. But don't worry, you don't have to be a programmer to benefit from this.

Just a few of our customers

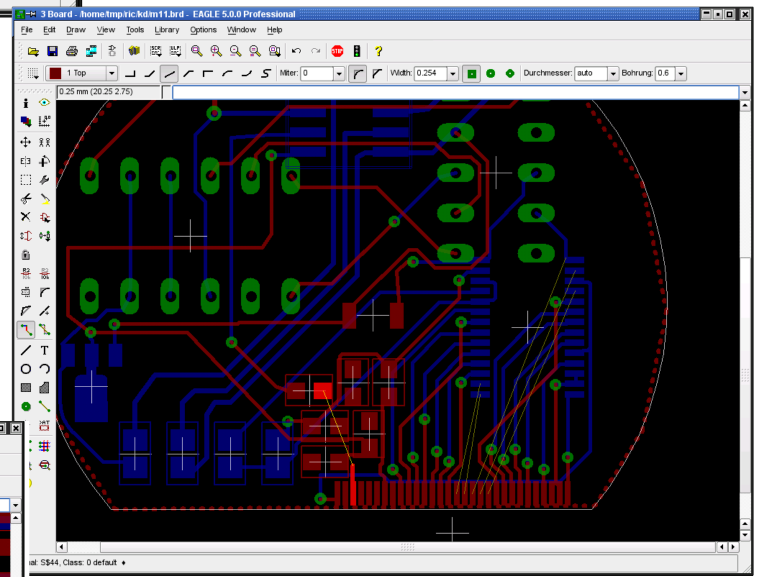
3M, AEG, Airbus, AMD, AMP, Amphenol-Tuchel, AT&T, Audi, BASF, Bayer, Black&Decker, BMW, Caltech, Daimler Chrysler, DFW International Airport, Du Pont, Eastman Kodak, Goodyear, Grundig, Hewlett-Packard, Hitachi, Honeywell, Hughes, IBM, Intel, ITT, Lockheed, Lufthansa, Mercedes, Motorola, Minox, NASA, National Semiconductor, NCR, NEC, Nokia, Philips.

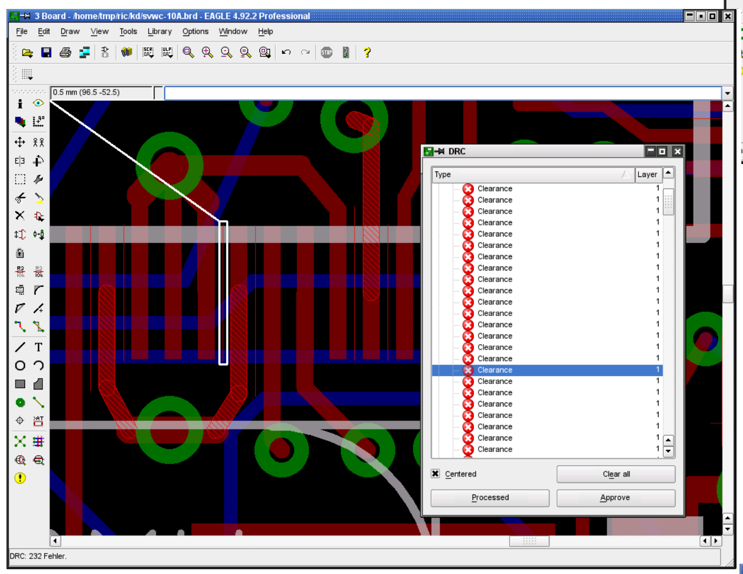
Design with EAGLE: It's that simple!



You draw a schematic using the symbols which you fetch from the supplied libraries. Then click the Switch-to-Board icon, and the component's packages appear next to an empty board, with the connections shown as rubber bands. You can then arrange the components on the board. Thanks to on-line Forward&Back annotation, the schematic and the board always remain consistent.

Now the desired connections (shown as rubber bands) are converted into non-crossing tracks. In EAGLE this can be done by hand, or the Autorouter can do it automatically. Critical connections are usually made by hand, while the remainder are left to the Autorouter.





To rule out the possibility of errors caused by the manual work, EAGLE carries out a Design Rule Check (DRC). This will check whether, for example, there are shortcircuits on the board, or whether different potentials are too close to one another.

You can design your own circuit symbols and packages as easily as you can schematics and layouts. You can alter or extend existing libraries to suit your needs or comfortably compose your own libraries. You decide what the symbols will look like, where the labeling should appear and what size it should have, what shape and size of solder land you want to use, what the rotation of a package should be. Even complicated components are defined within a few minutes.