EAGLE PCB DESIGN

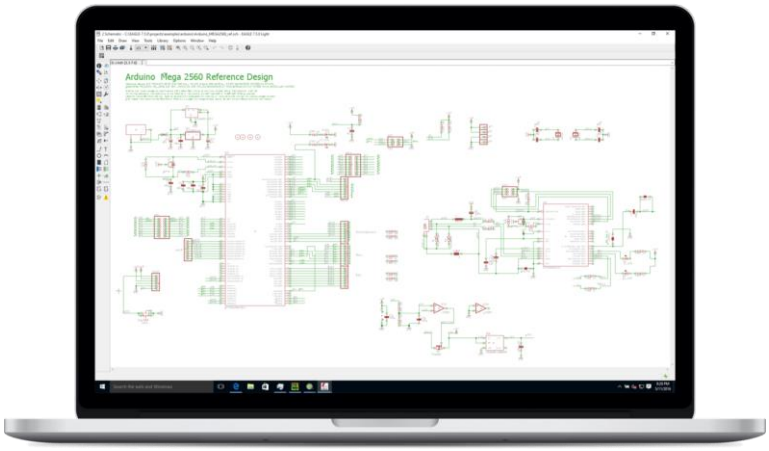
Powerful PCB Design

Tailored to meet the needs of professional engineers, makers and those at school!

Schematic Editor

The schematic editor allows you to create a symbolic easy-to-read representation of your design.
The goal of the schematic is to provide documentation about the design, allowing others to easily understand your design intent.
EAGLE comes with thousands of pre-made parts that simplify your schematic capture tasks.

- 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
- Advanced replace functions
- Online Forward&Back Annotation between Schematic and Board
- Automatic generation of supply connections
- Automatic board generation
- Electrical Rule Check
- User Defined Net Classes for Via Size, Wire Width and Clearance
- Schematic Hierarchy



Layout Editor

The layout editor brings your design to life. The board represents the physical reality of your design.
You can locate your components accurately and route copper traces between them. EAGLE's Layout editor includes sophisticated features such as assembly variants, differential pair routing and much more.

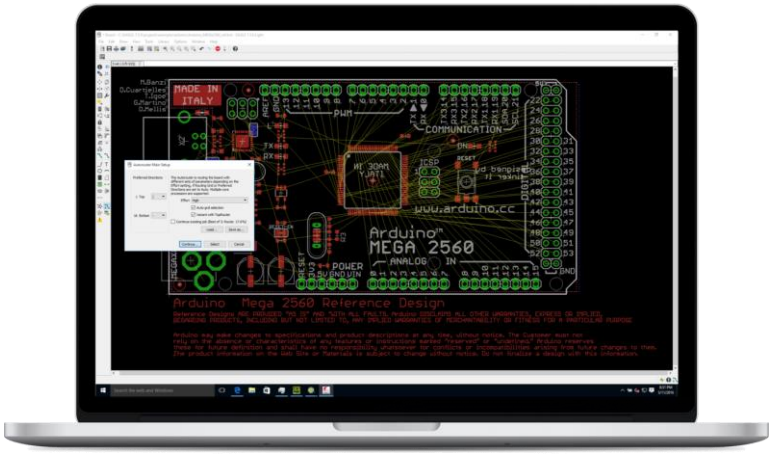
- 4 x 4m (about 150 x 150 inch)
- Full SMD support
- Support of Blind and Buried vias
- Rotation of objects in arbitrary angles (0.1degree steps)
- Components can be locked against moving
- Texts can be placed in any orientation
- Dynamic calculation of signal lines while routing the layout
- Magnetic 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

EAGLE PCB DESIGN

Autorouter

The autorouter is a useful tool for placing copper traces between components.
The autorouter can be guided by adjusting its cost factors to control its routing behavior producing useful results in no time. The Autorouter protects prerouted traces giving you full control over what nets need to be manually routed and which can be routed automatically.

- Routing grid down to 0.02 mm (about 0.8 mil)
- Follow-me-router, a tool that supports you in manual routing; the trace of a selected signal is calculated automatically and adjusts dynamically based on your cursor position
- Ripup&Retry algorithm
- Cost factors allow users to tailor the autorouter's behaviour to their needs
- 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



New since Version 7

Multi-Threaded

The autorouter can run multiple configurations simultaneously. The user is able to select the best outcome that accommodates his/her design. On computers with multiple core processors each thread can run on its own processor core. This optimizes EAGLE's use of the available hardware.

TopRouter

The new autorouter includes the option to use our TopRouter. This intricate process will result in boards with significantly less points of transition compared to our previous algorithm. This means very cost effective boards that will need less manual interaction by the end user.

Additional and New Seats

Edition	Modules/Functions	Schematic Sheets	Signal Layers	Routing Area (mm)	Use	1 USER	3 USER	5 USER	10 USER
Ultimate	Schematic, Layout, Autorouter	999	16	4000x4000	Commercial	LSR-1CP	LSR-3CP	LSR-5CP	LSR-10CP
Premium	Schematic, Layout, Autorouter	99	6	160x100	Commercial	LSR-1CS	LSR-3CS	LSR-5CS	LSR-10CS
Standard	Schematic, Layout, Autorouter	1	2	100x80	Commercial	LSR-1CL			
Ultimate LS	Schematic, Layout	999	16	4000x4000	Commercial	LS-1CP	LS-3CP	LS-5CP	LS-10CP
Premium LS	Schematic, Layout	99	6	160x100	Commercial	LS-1CS	LS-3CS	LS-5CS	LS-10CS
Maker	Schematic, Layout, Autorouter	99	6	160x100	Non-Commercial	LSR-1SS			

Upgrades for existing Eagle Customers (Valid Serial Number REQUIRED)

Edition	Modules/Functions	Schematic Sheets	Signal Layers	Routing Area (mm)	Use	1 USER	3 USER	5 USER	10 USER
Ultimate	Schematic, Layout, Autorouter	999	16	4000x 4000	Commercial	UPGRADE-LSR-1CP	UPGRADE-LSR-3CP	UPGRADE-LSR-5CP	UPGRADE-LSR-10CP
Premium	Schematic, Layout, Autorouter	99	6	160x100	Commercial	UPGRADE-LSR-1CS	UPGRADE-LSR-3CS	UPGRADE-LSR-5CS	UPGRADE-LSR-10CS
Ultimate LS	Schematic, Layout	999	16	4000x 4000	Commercial	UPGRADE-LS-1CP	UPGRADE-LS-3CP	UPGRADE-LS-5CP	UPGRADE-LS-10CP
Premium LS	Schematic, Layout	99	6	160x100	Commercial	UPGRADE-LS-1CS	UPGRADE-LS-3CS	UPGRADE-LS-5CS	UPGRADE-LS-10CS
Maker	Schematic, Layout, Autorouter	99	6	160x100	Non-Commercial	UPGRADE-LSR-1SS			

*** 30 and 50 Seat licenses are also available for purchase and upgrades. Please call for quote.

**** Licenses for education customers are free of charge. Please visit cadsoft.io to download.

X-ON Electronics

Largest Supplier of Electrical and Electronic Components

Click to view similar products for Cadsoft ***manufacturer:***

Other Similar products are found below :

Cadsoft