PCB DESIGN TOOLS
from element14
WHY EAGLE?
- Low cost, easy-to-use printed circuit board design solution with maintenance and support included
- The #1 design tool for open source hardware platforms
- Real-time access to element14 parts library for research and selection (over 500,000 components)
- Links to popular prototype vendors for low cost, low volume fabrication and assembly
- Data exporting for 3D visualization, simulation and board production
- Multi-user discounts and educational pricing available
- Fully supported on Windows, Linux and Mac
- Customizable modules and multiple editions available

SCHEMATIC EDITOR
- Allows up to 999 sheets to support complex designs
- Control schematic designs with Electrical Rule check (ERC)
- Hierarchical design promotes design reuse of electrical blocks
- Arrange schematic sheets with Drag & Drop capability

LAYOUT DESIGN
- Performs design rule checks to ensure all connections are correct
- Support for differential pair routing and meanders
- Easy-to-use dimensioning tool

AUTOROUTER
- Automatically route single connections, selected connections or an entire board
- Interactive ‘follow me router’
- Supports blind and buried vias for multi-layer designs up to 16 layers
- Specify custom routing features, including layer preferences and routing grids

CadSoft Eagle

Accelerate your PCB design with Eagle
Easily Applicable Graphical Layout Editor (EAGLE) is an EDA CAD tool for designing printed circuit boards, featuring a low initial cost and many licensing options to fit the needs of the maker, hobbyist or designer.

LICENSING & PURCHASING OPTIONS
Eagle is available with all or only the specific modules you need for schematic, layout and auto routing in the following editions.
- **Light**: 1 schematic sheet, 2 signal layers, 100×80mm routing area. Light edition includes all three modules.
- **Standard**: 99 schematic sheets, 6 signal layers, 160×100mm routing area.
- **Professional**: 999 schematic sheets, 16 signal layers, 4×4m routing area.
- **Hobbyist**: all the Standard version capabilities for non-commercial use.

Want to learn more? element14.com/Eagle
WHY CIRCUITSTUDIO?
When it’s time to design a PCB, you need to get to work instantly on the task at hand. CircuitStudio gives you an intuitive, streamlined interface to help get your designs done fast.

POWERFUL NATIVE 3D PCB
Native 3D PCB in CircuitStudio lets you visualize and navigate your board as it appears in the real world.
- Powerful PCB design engine gives you the freedom to quickly layout and route your board
- View and navigate your board in Native 3D to verify mechanical form
- Directly integrate 3D STEP models for component bodies and mechanical enclosures
- A comprehensive PCB rule system with integrated DRC protects the integrity of your design intentions
- Interactive pin swapping synchronized with your schematic allows you to reassign the nets on a selected component in seconds

STRAIGHT-FORWARD TOOLS
CircuitStudio gives you the flexibility to create detailed and comprehensive schematic designs and quickly synchronize them for board layout in the same easy-to-use interface.
- Streamlined schematic editing engine gives you powerful tools to capture your design before board layout
- Create complete multi-sheet and hierarchical schematic designs
- Mixed-mode simulation capabilities are included in the same streamlined interface
- Add revision control to your projects and documents with version control support
- Easily import existing designs using the Eagle importer for projects and libraries

POWERFUL LAYOUT & AUTOROUTING
• Interactive routing features include push, stop-at-first and look ahead
• One-click auto routing
• Differential pair routing for high-speed trace design
• Customizable design constraints including object clearance and copper-to-copper spacing

EXPANSIVE LIBRARY
CircuitStudio offers an expansive library of components with accurate parts data and models so you can spend less time building models and more time using them.
- ISO 9000 compliant library features over 350,000 parts with full symbols, footprints and 3D Models
- Life-cycle managed parts list
- Live link to part distributor costs and availability

Want to learn more? element14.com/AltiumCS

PROFESSIONAL PCB design tool, ready to pick up and go
CircuitStudio is built on 25 years of Altium R&D experience and features powerful capabilities for schematic capture, PCB layout and routing, as well as 3D viewing to get your designs done faster and easier.

LICENSING & PURCHASING OPTIONS
CircuitStudio is offered as a single seat perpetual license with an annual subscription fee for access to updates and maintenance fixes. If three or more seats are needed please contact our Altium-certified sales and support teams.
- North America Support Altiumsupport@newark.com
- Europe Support Altiumsupport@farnell.com
- Asia Pacific Support Altiumsupport@element14.com
# TOOL FEATURES COMPARISON

## SCHEMATIC EDITOR

<table>
<thead>
<tr>
<th>Feature</th>
<th>CadSoft Eagle</th>
<th>CircuitStudio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Schematic sheets</td>
<td>Up to 999 sheets per schematic</td>
<td>Virtually unlimited number of schematics</td>
</tr>
<tr>
<td>Design collaboration</td>
<td>Hierarchical schematics with drag &amp; drop sheet sorting</td>
<td>Advanced multi-sheet and hierarchical schematic management</td>
</tr>
<tr>
<td>Component update</td>
<td>Replace function for parts without loss of consistency between schematic and layout</td>
<td>All project documents are unified and linked, allowing easy update from library to schematic to PCB without loss of information</td>
</tr>
<tr>
<td>Design consistency</td>
<td>Automatic online forward and back annotation between schematic and board</td>
<td>Fully configurable forward and backward annotation and automatic checks. Simple and comprehensive Engineering Change Order (ECO) process for board updates from the schematic</td>
</tr>
<tr>
<td>Supply pin auto connections</td>
<td>Automatic generation of supply connections</td>
<td>Electrical connectivity is handled automatically for wires, buses, signal harnesses, net labels and power ports</td>
</tr>
<tr>
<td>ERC support</td>
<td>Electrical Rule Check in the schematic and consistency check between schematic and layout</td>
<td>Comprehensive project compilation and electrical rules checking</td>
</tr>
</tbody>
</table>

## LAYOUT EDITOR

<table>
<thead>
<tr>
<th>Feature</th>
<th>CadSoft Eagle</th>
<th>CircuitStudio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum board size</td>
<td>Up to: 4×4m (about 150×150 inch) board support</td>
<td>Virtually unlimited board area</td>
</tr>
<tr>
<td>Supported component technology</td>
<td>Full SMD + BGA and package variants support</td>
<td>Full support for even the most complex component and package types, able to attach multiple footprints to a device</td>
</tr>
<tr>
<td>Component placement</td>
<td>Parts placement locking, rotation in 0.1 degree steps with text in any orientation</td>
<td>Advanced snap grid settings for parts placement rotated to any angle, tools for silkcreen component positioning in any orientation</td>
</tr>
<tr>
<td>Pair routing and meander support</td>
<td>Routing and length compensation tools</td>
<td>Tools for routing complex high speed topologies</td>
</tr>
<tr>
<td>Trace chamfers</td>
<td>Mitering and rounded corner wire joints</td>
<td>Quickly add mitered and rounded corners to completed routing. Freely drag components while maintaining connectivity to tracks</td>
</tr>
<tr>
<td>DRC reports</td>
<td>Graphically displayed DRC results</td>
<td>Fully-customizable DRC in real-time or batches</td>
</tr>
<tr>
<td>User defined functionality</td>
<td>User definable programming language and large ULP Program libraries freely available</td>
<td>Add third-party extensions or develop your own with the CircuitStudio SDK</td>
</tr>
<tr>
<td>File export</td>
<td>Output of manufacturing data for pen plotters, photo plotters and drilling machines with the CAM Processor</td>
<td>Customizable file generators output full range of industry-standard file formats</td>
</tr>
<tr>
<td>Autorouter</td>
<td>Grid-less Algorithm, multi-Core support</td>
<td>One-click autorouter with including a push and shove mode</td>
</tr>
<tr>
<td>3D capabilities</td>
<td>3D visualization and export support</td>
<td>Native 3D visualization and MCAD integration</td>
</tr>
<tr>
<td>Signal analysis</td>
<td>Dynamic signal line calculation while routing</td>
<td>Dynamic connection lines and interactive masking options</td>
</tr>
<tr>
<td>Copper pouring</td>
<td>Copper pouring (ground planes) supported</td>
<td>Dedicated copper plane layers, polygons editing, and click and drag</td>
</tr>
</tbody>
</table>

For more information on Design Tools