CircuitStudio® Evaluation Guide
For Eagle™ Users
WHY CHANGE & WHY NOW?

Are you falling short of meeting minimum design specifications, or missing release dates and product cost targets? Do you have the expertise to design the perfect board, but are unable to achieve your “feature elegance” targets due to the limitations of your design environment? Do you prefer to own, rather than rent, your software license? Have you decided that it’s time to change?

With the rapidly increasing complexity of modern products, all with larger circuits in smaller packages, you can no longer accept the inability to meet your goals as normal and permissible. You require a complete and cost effective solution that offers feature-rich products, and an intuitive and modern interface. Now is the time to take a closer look at CircuitStudio.

OVERVIEW OF EAGLE™ PRODUCTS

Eagle™ sells its PCB Design tool in two different stages of expansion:

<table>
<thead>
<tr>
<th>Standard</th>
<th>Premium</th>
</tr>
</thead>
<tbody>
<tr>
<td>PCB Schematic &amp; Layout</td>
<td></td>
</tr>
<tr>
<td>Autorouter</td>
<td></td>
</tr>
<tr>
<td>99 Schematic Sheets</td>
<td>999 Schematic Sheets</td>
</tr>
<tr>
<td>4 Signal Layers</td>
<td>16 Signal Layers</td>
</tr>
<tr>
<td>160 cm² Routing Area</td>
<td>Unlimited Routing Area</td>
</tr>
</tbody>
</table>

Along with these commercial options there is also a Non-Commercial License to choose from.

WHY CIRCUITSTUDIO?

Your electronic designs demand the highest grade of efficiency and performance. When your productivity is measured against immovable deadlines, precise layouts, accurate documentation and exact fittings, you can’t afford to not invest in a complete PCB design platform. CircuitStudio has everything you need to meet your design challenges and is powered by Altium Technology and a proven track record of delivering innovative, differentiating features in predictable and reliable releases.

OVERVIEW OF CIRCUITSTUDIO

A unified environment for your design, data and release management process. CircuitStudio provides a unified PCB design environment that is easy to learn and use, following the Windows standard behaviour. You have the features you need to make design decisions early, perform tasks efficiently and implement checks and balances throughout your design process.

With CircuitStudio, you’re equipped with everything you need to solve even the most complex design challenges, including:
Advanced Supply Chain Management

Always ensure you make intelligent part selections with real-time pricing and availability data from your most trusted and reliable suppliers. Reduce the likelihood of costly and time-consuming design re-spins with complete visibility over supply chain data early in the design process.

Access to the Altium Content Vault

Access to over 200k components from the Altium Content Vault.
Centralized Output Generation

Control the consistency and reliability of your project outputs with the ability to configure and generate accurate design data from a centralized location, all at one time with optional validation checks to halt the process in the event you forgot to do your final ERC/DRC checks before generation.

Seamless ECAD/MCAD View

Within the PCB editor, easily switch between 2D and 3D views of your board. Using components with STEP models included, or adding STEP models or 3D Component Bodies to your custom footprints provides a real world view of your PCB in 3D. Take this further with the ability to export a STEP model of your entire PCB to collaborate with your MCAD counterparts, as well as full 3D part to part clearance checking using PCB Design Rules.
ALL WITHIN ONE, MODERN USER INTERFACE

All CircuitStudio features are presented within one, modern user interface (UI). Whether you’re responsible for every aspect of the design process or delegated specific tasks, a consistent selection and editing paradigm allows you to quickly move between design tasks. The context sensitive UI changes when you switch from one aspect of the process or document to another, providing you with the most relevant and intuitive selections. If you focus solely on one element of the design process, the UI can be configured to match your preferences. The consistent look & feel allows you to quickly become proficient as you take on additional design tasks.

LIBRARY AND COMPONENT MANAGEMENT IN EAGLE™

Eagle™ offers a file-based library which incorporates common file access limitations to share library content with other team members. There’s also a feature that allows you to add design links to components to maintain the synchronization across your schematic and PCB layouts.

LIBRARY AND COMPONENT MANAGEMENT IN CIRCUITSTUDIO

Altium is the leader in providing a complete solution for the PCB design, development and production process. One of the fundamental aspects of this process includes a close connection to supply chain and real-time component management. In CircuitStudio, there are two main library concepts available. Easy file-based structures as well as access to the Altium Content Vault.

EAGLE™ SCHEMATIC CAPTURE

Eagle™ offers a nice schematic entry that provides the most commonly needed features. However, there are some obvious limitations when it comes to bottom-up development of hierarchical designs and snapping outside the standard grid. With the user-interface being quite different from the Windows world, there can be a learning curve that could result in one or two gray hairs.
CIRCUITSTUDIO SCHEMATIC CAPTURE

CircuitStudio schematic capture technology is built using Altium Technology who has long been recognized as a technology differentiator. Engineers and designers will find that CircuitStudio schematic features are easy to learn and quickly improve productivity on all designs ranging between relatively simple single-sheet schematics to complex multi-sheet hierarchical projects. Here's why:

- **Document your intent, not just your netlist**. Schematic documents should be more than just pin to pin connections and components, with great looking schematics and excellent markup tools. Create a schematic which clearly documents your intent, as well as your interconnections.

- **Starting a schematic is fast and easy** with intuitive dialogs, simple placement and drawing tools, and templates... your schematics look great before a component is even placed!

- **Quickly select and place qualified components** from integrated libraries with real-time links to component suppliers.

- **Leverage powerful ECO feature to transfer a captured design to a new PCB**, make changes to an existing design on either the schematic or PCB, synchronize the schematic and board, and compare and resolve differences.

- **Local History and Version Control** make it easy to revert to an earlier copy, and a powerful differences engine can compare your current document with past versions to see what has changed.

![CircuitStudio Schematic](image)
RULES AND CONSTRAINTS IN EAGLE™

In Eagle™, you’ll find a basic, matrix-driven way to configure minimum clearances and width definitions for elements and net classes. More complex rulesets that take different Layers into account cannot be defined.

RULES AND CONSTRAINTS IN CIRCUITSTUDIO

True to its unified and easy-to-use nature, CircuitStudio provides a streamlined PCB Rules and Constraints Editor, with more control over the entire design process:

- Easily browse, create, prioritize, define the scope, edit, duplicate and delete rules all in one editor.
- Evaluate your rules in a table-based summary that provides straightforward review.
- Define multiple rules of the same type, but each targeting different objects.
- Decide exactly what a rule’s precedence will be and how it will be applied to target objects.
- Create new rules in a step-by-step manner with the New Rule Wizard guiding you along the way.

PLACEMENT AND ROUTING IN EAGLE™

Being able to place components solely within the boundaries of a PCB limits the developer when cleaning up the mess of components caused by the automatic, uncontrollable and disorganized schematic-to-PCB synchronization. The idea of sorting and placing components according to their destination task and possibly routing those modules is completely impossible in this way. Even more annoying is making adjustments when most of the routing work has already been done, as routing the last set of tracks often requires pushing other tracks away. An alternative to deleting and redrawing those tracks manually should exist to facilitate the routing process for designers.
PLACEMENT AND ROUTING IN CIRCUITSTUDIO

It’s crucial to have an organized and efficient placement of components on your PCB. CircuitStudio offers enhanced capabilities to ensure proper component placement and create the most efficient board layout possible. When footprints are first loaded into the PCB they are conveniently grouped and placed just outside the board shape ready to be moved into position with a variety of easy placement tools, or simple drag and drop.

- Dynamically place and drag components utilizing various placement tools, easy selection and group movement, and simple drag and drop functionality
- Easily align multiple components to keep your board layout organized and tidy.
- Mask or filter objects in the workspace to gain more visibility over your board.
- Optimize your routing layers with the Layer Stack Manager giving you full control over all layers.
- Automatic Loop Removal makes correcting a route easy, simply route the new connection over the top of the old, and the older route will be removed automatically upon completion of the new path.

CircuitStudio is continually adding more powerful and differentiating placement and routing features. These features will increase your productivity, streamline your core PCB design tasks, and reduce your time to market.

FOR A FULL EVALUATION

Obtain a 30-day full featured evaluation license at [http://www.circuitstudio.com/](http://www.circuitstudio.com/)