

# Make Your PCB Design Ideas Fly High With **EAGLE**

**PRIYA RAVINDRAN**

**Y**ou set out to design your own printed circuit board (PCB) but are confused about which tool to use. How do you come to a decision? What are the factors you need to consider? In this article, we introduce you to EAGLE and tell you what you need to know to work with this tool.

EAGLE, a prototype of CadSoft Computers, is an easily applicable graphical layout editor. It is a flexible, expandable and scriptable electronic design automation application with

three modules, namely, schematic editor, layout editor and library editor, with identical user interfaces (UIs).

Designing your circuit has been made easy by access to design components in Farnell database, from within EAGLE environment via DesignLink, from within EAGLE environment, owing to the acquisition of CadSoft by Premier Farnell in 2009. Thus, you have everything you might need, in one place.

## A new angle in simulation

EAGLE supports the newly-developed interface of PCB-Sim that lets you do the following:

3. Read S-parameter based data and convert to time-domain, with automatic correction of inaccuracies in the input data

4. Work between EAGLE based and SPICE based models with the addition of an integrated SPICE simulation engine. Also, see the circuit's functioning as waveforms

5. Easily import input-output buffer information specification models that describe the behaviour of an integrated circuit, thanks to SPICE engine

## Route your design as EAGLE flies

Autorouting is fully integrated into the basic program. You can choose between manual or automatic routing option, and switch between the two easily. If you choose to go with the manual routing method, simply follow the follow-me-router tool that guides you at every stage. The smallest routing grid is 0.02mm in size. You can place components as you please, within the layout structure and use up to 16 signal layers. If you

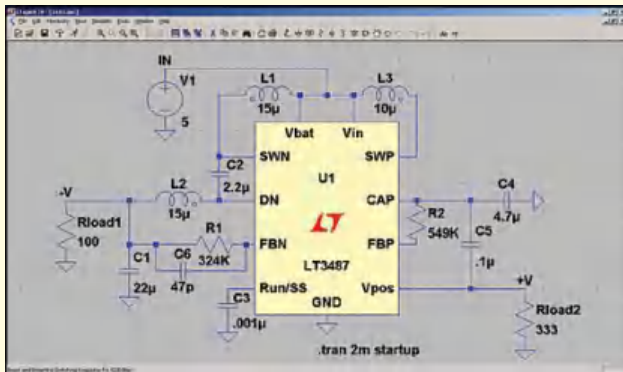


Fig. 1: Simulating with SPICE (Image courtesy: [chrisgammell.com/best-free-spice-program](http://chrisgammell.com/best-free-spice-program))

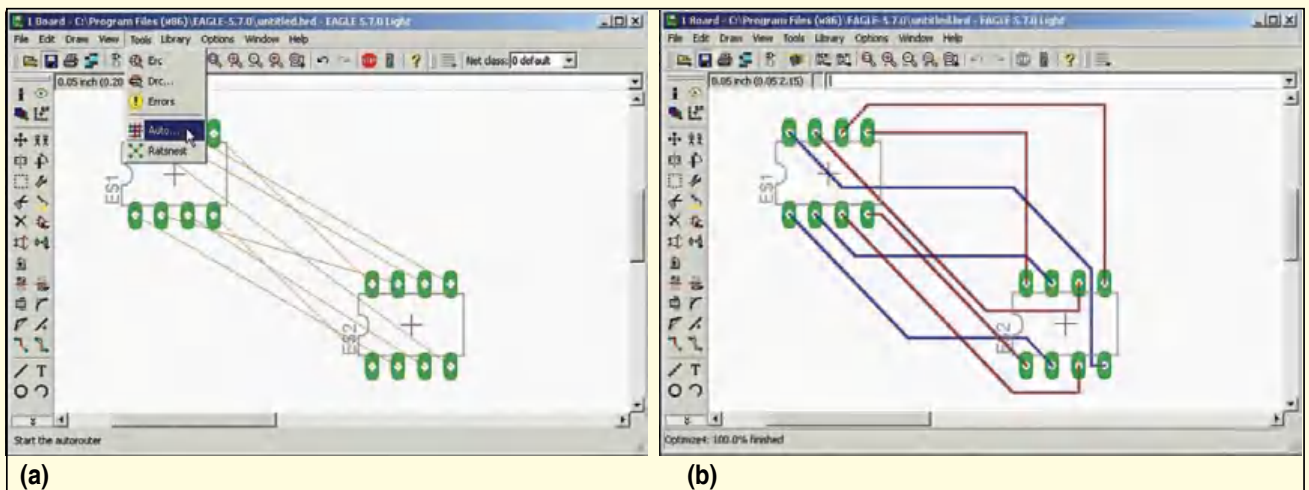


Fig. 2: Make your design route by itself: (a) before autorouting, and (b) after autorouting (Image courtesy: [www.youtube.com](http://www.youtube.com))

spot an error after you finish your design, there is no need to panic. You can rework your design as many times as you want, with Ripup&Retry algorithm.

You are done with circuit construction but your design is so huge that you have multiple ways of routing the signals. Which path do you choose? How do you decide which is better? Version 7 of the tool comes to the rescue with multi-thread routing. With this, you can run multiple configurations simultaneously and choose the outcome that best accommodates your design. You can also choose to go with TopRouter option that results in boards with significantly lesser points of transition and effectively reduces cost and effort.

## It has never been easier to print a PCB

Print your PCB without any hassle with the integrated PCBQuote within EAGLE. You can choose from options like quick-turn PCB prototypes, full-specification prototypes, low-cost production and custom PCB quotes, thanks to a tie-up with PCB manufacturers.

The software is compatible with Windows, Linux and Mac systems and is easy to use. SparkFun, the popular do-it-yourself site, works with EAGLE files. Adafruit, Arduino

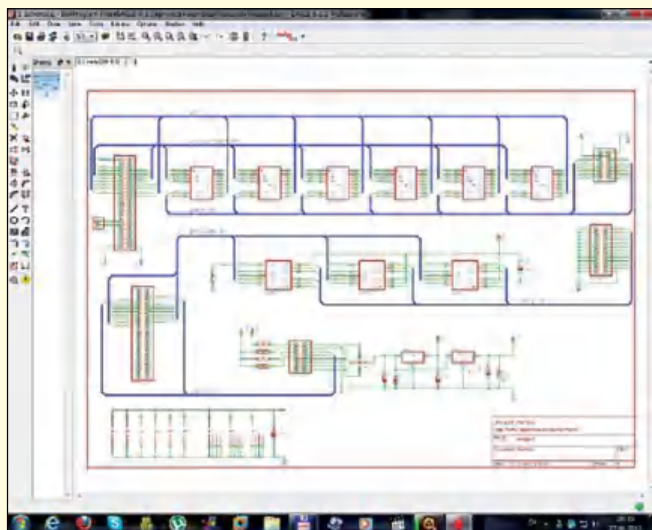
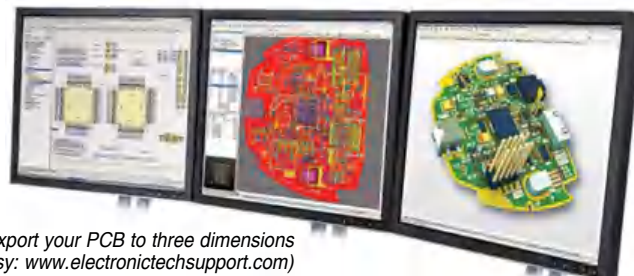


Fig. 3: A scheme of the schematic  
(Image courtesy: thegioitinhoc.vn/do-hoa)

## The new and exciting

- View and work with library components much easily with an enhanced library editor
- Access components better with a revamped control panel
- Improve performance and simplify installation with the 64-bit version
- Get a clear 3D picture of your design and make 3D PCBs with the extended intermediate data format option
- Sub-divide your design and reuse it with the hierarchical design feature that was introduced in version 7.2
- Enjoy the visual experience with a modernised graphical UI



Export your PCB to three dimensions  
(Image courtesy: www.electronictechsupport.com)

## Awards and accolades

EAGLE has been placed first in the category Development Software of the annual readers' awards by German magazine *Elektronik*, consecutively, in 2014 and 2015.

*Elektronik* has awarded CadSoft's EAGLE software Engineering Software Product of the Year nine times (in the years 2000, 2003, 2004, 2006, 2009, 2010, 2012, 2013 and 2014). The award is decided by votes from the design engineering readership of the German based magazine.

and Dangerous Prototypes also put EAGLE to good use.

## Eager to get on it but wondering where to start

**Design the schematic.** First, create a new project for your design.

Make sure you include all required libraries, and whatever you create next, under this folder.

Next, open one of the 999 sheets provided and make your circuit. Add the parts; you can either drag-and-drop these or simply copy from another circuit, connect these together and use the automatical-

ly-generated supplies to power your circuit. You can even define your own net classes.

Once the circuit is done, run an electrical rule check to check for design errors and schematic-layout compatibility issues.

To make your work easy, you can also use pre-written programs, the user language programs (ULPs). You just need to add the ULP to your code and run it, of course, with the modifications you need. You can find ULPs at [www.cadsoftusa.com/downloads/ulps](http://www.cadsoftusa.com/downloads/ulps)

**Create the layout.** The layout is a 4m × 4m structure offering full support to surface-mount devices and you work with it using the layout editor. You can play around with your circuit to make it look exactly the way you want. Lock in place the components in your schematic, rotate these at arbitrary angles and add text to name components and orient these as you wish.

Tracks are flexible and the layout also supports differential pair routing. As you route your circuit, signal lines are dynamically quantified. Run a design rule check, and check if your design is correct. Generate data in whatever format or package variant you require by defining the programming language.

### Offering the support you might need

Registered EAGLE customers are always free to write to the technical team at their respective locations, the email ids of which can be found on their website. A frequently asked questions (FAQ) page ([www.cadsoftusa.com/training-service/faq](http://www.cadsoftusa.com/training-service/faq)) and a forum ([www.cadsoftusa.com/community](http://www.cadsoftusa.com/community)) aids in solving user queries. A download area ([www.cadsoftusa.com/downloads/libraries](http://www.cadsoftusa.com/downloads/libraries)) provides links to all their latest releases.

CadSoft also provides online trainings, webinars, videos and tutorials to help you work with the software, the details of which can also be found on their website.

You can purchase the software online by logging into CadSoft website ([www.cadsoftusa.com](http://www.cadsoftusa.com)). Once bought, upgrades, licence renewal, additions like the library, projects and so on require minimal effort.

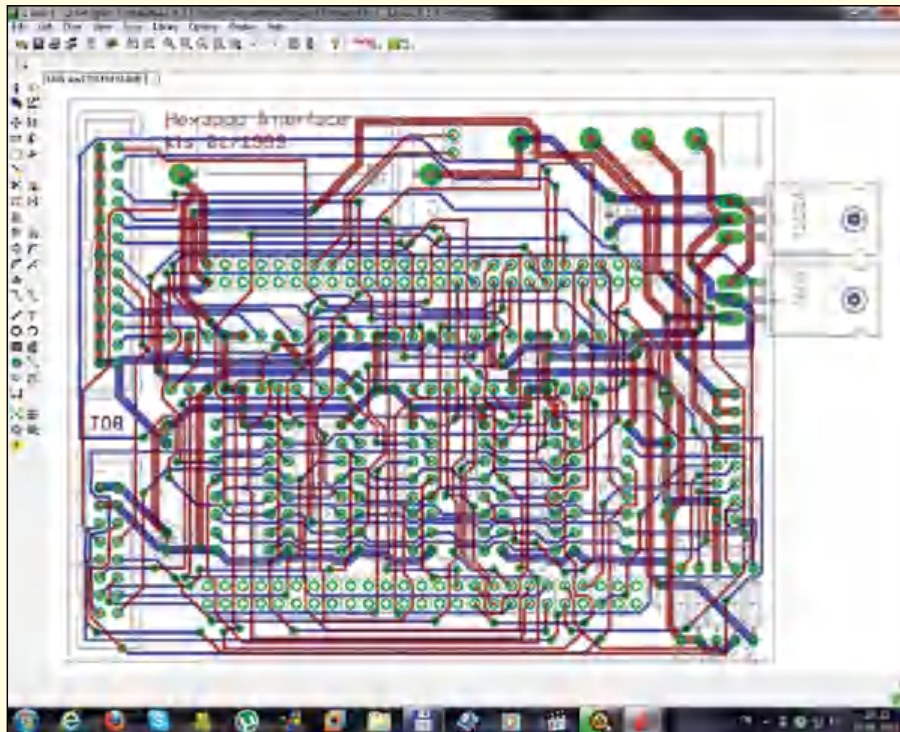


Fig. 4: Be your layout's architect (Image courtesy: [thegioitinhoc.vn/do-hoa](http://thegioitinhoc.vn/do-hoa))

Decide if this is the right tool for you by becoming familiar with the limited version. All you need to do is install the setup from the DVD and explore. The same is also available on the official website as EAGLE Light Edition.

### Users and their views

EAGLE seems to be getting mixed reviews. While it has its own set of loyal customers who have been using it for years, some users seem to pre-

fer other software like KiCad, Circuit Wizard and Design Spark.

What makes EAGLE kick for the loyalists is the free version (limited only by the size of the board), support and response from CadSoft to their queries, the fact that there is information aplenty available about EAGLE and strong support from PCB manufacturers.

The problem areas have been the language in the UI; terms used seem to convey meanings other than intended, like usage of 'drop' for 'hide' and such. There are also some complaints about the software being a little tedious to learn initially, but once you spend that initial time, you seem to have learnt the trick.

Well, a software application that is a winner of awards and one that has so many regular users, including commercial PCB designers and professionals, must have something right in it. With the tool being upgraded with every new release, it is left to you to try this for yourself and arrive at a verdict. Write to us about your take on EAGLE, after using it. ●

### Quotes courtesy: [www.newark.com/cadsoft](http://www.newark.com/cadsoft)

"Each module you can imagine, starting with a simple power adaptor via the motor-control module, up to the battery management system developed in-house, is developed by using EAGLE. In this case, EAGLE is a one-stop solution. First, it can be used to create circuit plans and layouts, ranging from simple to complex. EAGLE makes it child's play to optimise the size of the board or the general arrangement of components. Since elements such as the required training period and efficiency (one program for all) are our top priority and the setup has allowed all students, even those without any previous knowledge, to get to grips with the operation in just a few hours, EAGLE software package from CadSoft has become a standard tool, which we cannot do without."

—Sarah Schwöbel, *munichMotorsport*

"What I like most about EAGLE is the flexibility it offers. With EAGLE ULPs, there is no limit to what I can create [in terms of] designing PCB boards!"

—Thomas Benoit, *SERELEC*

"I design my boards using EAGLE, buy components via the integrated DesignLink and get these produced with the offered PCB fabrication service. This is just great!"

—*Electronics engineer*