

# EAGLE New User Cheat Sheet

by Jorge Garcia, Cadsoft Support Specialist

## Key things every new user should know:

1. Keep the schematic editor and board editor open together at all times when your working on your design. Closing one editor and continuing to work on the other will break consistency and changes made will no longer track between editors.
2. Don't deviate from a 0.1”(2.54mm) grid in the schematic editor. All of the default EAGLE libraries are made to a 0.1” grid in the schematic editor. If you deviate from this you will find that you have a very difficult time getting your components to connect.
3. Don't use WIRE for anything other than artistic features. Connections in the schematic are defined using the NET command and copper tracks are laid down using the ROUTE command in the layout. If you use wire for either of the above key operations you will find that sometimes components won't connect as expected.
4. EAGLE's search functionality is an exact string search which means if you're off by a single letter in a part number EAGLE won't be able to find it. A more prudent approach is to make liberal use of the wild card character(\*). For example don't search for LM555(you'll get nothing) search instead for \*555\* this tells EAGLE that if 555 shows up any where in the device name this is a valid result.
5. EAGLE uses a verb-noun work flow. What this means is that you first select what action you want to perform and then what objects you want to perform that action on. It may seem odd at first but once you're used to it you'll find that it's a faster way to work.
6. Do not modify EAGLE's default libraries, these are a known state. The best approach is to make your own library and then copy whatever you need from the default libraries to it.
7. Make sure you know where you are saving your work. Do not save anything to EAGLE's installation directory, it is recommended that you save your work in the eagle folder EAGLE creates in Documents(on windows, on Linux or Mac this folder is created in your home directory)
8. The EAGLE manual is included with your installation, you'll find it in the EAGLE installation directory inside the doc folder.
9. In order for EAGLE to establish a connection it must end EXACTLY at a pin. If a net ends just short or just past a pin it is not connected to the pin. A good way to double check a connection is to try moving the part and seeing if the nets follow.
10. Never move nets in an attempt to make a connection, it won't work. Use the NET command to draw the connection instead.
11. In the ADD dialog, the drop button removes a library from the active search list. It does not delete the library and it definitely does not place the component on your design.

## Useful Libraries:

- rcl – This library contains all of the passive linear components you might need resistors, capacitors, and inductors.
- Linear – This library contains many of the common IC's you'll come across when working with electronics such as the LM741 op amp and the 555 timer.
- Supply1 and supply2- These libraries contain the various power symbols you'll need such as GND and VCC symbols. These automatically connect to all of the instances within a schematic. In other words if you have 20 GND symbols on your schematic they are all automatically

connected.

- Ref-packages – This is one of the most useful libraries when making your own parts, most of the common IC footprints are in this library so you can usually copy a footprint to your personal library and then you only have to worry about making the symbol and mapping it to the package.
- Frames- contains drawing frames and title blocks for documenting your schematic, covers most of the common paper sizes.
- Pinhead – this is the library that contains 0.1” pitch headers which are common in many electronic projects.

### **Useful ULPs:**

Some of the ULPs listed below are not included with EAGLE, but are up on our website those are indicated with an \*. The ULPs on our website can be found at [www.cadsoftusa.com](http://www.cadsoftusa.com)-> Downloads-> User Language Programs.

- bom.ulp – This generates a very basic Bill of Materials. There are many variations on the theme which can be found up on our website. Of notable interest are BOM-EX\* and bom with attributes\*.
- EAGLE' UP – Not really a single ULP but a very handy interface between EAGLE and Google Sketchup. It allows you to create 3D models of your design. See <http://eagleup.wordpress.com/> for more info.
- Exp-project-lbr.ulp – This is a very handy ULP which extracts the libraries from a board/schematic pair. Very useful if you don't have access to the libraries that were used to create the original design.
- Explode.ulp\* - Handy ULP which breaks a symbol or package into its constituent lines and arcs.
- Cam2image.ulp- Handy for those that make their own boards at home using the toner-transfer method. Generates high resolution images from the gerber files.
- Import-bmp.ulp – Allows you to import a bitmap image into a board or schematic. Useful for adding company logos or scope captures to your designs.