

# Library Management and Part Creation Standard v1.0

by Cadsoft Computer Support  
11/14/2011

**Objective:** The intent of this document is to provide a BASIC guideline as to the best practices for developing user libraries. We have noticed differing levels of quality among user contributed libraries and our hope is that by developing this standard we will help raise the overall quality of user contributed libraries.

**Usage:** This document is split up into four main sections, Library, Symbols, Packages, and Devices. Under each main section the best practices for that section are outlined followed by some supplemental recommendations which are generally applicable, however they are not as strictly recommended as the best practices.

## Libraries

1. Do not modify EAGLE's default libraries. They represent a known state which support staff can rely on when assisting users. The preferred approach is to copy elements into a user made library and then modify them to suit your needs.
  - Copy procedures are discussed in the EAGLE manual section 8.12.
2. User libraries should not be stored in EAGLE's internal lib folder. The reason being that if anything should happen to your EAGLE installation(deletion, uninstall, viruses, hardware failure, etc.) your libraries will be gone with it. It therefore recommended to store your libraries in a separate location that can be easily backed up.

## *Recommendations for Libraries*

1. In light of practice number 2 listed above, below are some suggestions for best use of user developed libraries.
  - Some users like to make a separate library for each project they work on and store it along with the board and schematic file that way everything is together and easily accessible. A simple USE command in the schematic/board editor will make the library active and available for the ADD command.
  - Another possible scheme, is to store all of the libraries in a single folder located outside of EAGLE's internal directory, and then include it's path in the Library search directory. The advantage of this method is that all of the user libraries are in one place for easy back ups. Section 4.1 of the EAGLE manual discusses the Directories dialog.
  - Set a second path to your own library in Control-Panel | Options | Directories | Libraries

## Symbols

1. Symbol pins must always align to a 0.1"(2.54mm) grid.
2. Symbols must always have >NAME and >VALUE place holders on layers 95 Names and 96 Values respectively.
3. Center the symbol around the origin, however make sure that the pins still lie on a 0.1"(2.54mm) grid.

## *Recommendations for Symbols*

1. Power pins should generally have their pin direction parameter set to Pwr.
2. Input pins should generally have their pin direction parameter set to IN.
3. Output pins should generally have their pin direction parameter set to OUT.

4. Passive pins should generally have their pin direction parameter set to PAS.
5. Also IO, HIZ, NC.
6. Do not use pin direction SUP for a normal Symbols. SUP is only for supply-pins!
7. Any symbol notes or tips should be on layer 97 info.
8. It is better to give general names(OP AMP, DIODE, etc.) to symbols instead of specific part numbers (LM741, 1N4007, etc.). This makes it easy to reuse the symbol if your library is going to contain many variations of the same type of component for example a library full of op amps can probably make use of one op amp symbol.
9. Never name symbol pins with respect to pad numbers/names. Pin names should always represent the function of the pin.

### **Packages**

1. Design a package assuming that it will be placed on top of the board (Use t layers). The MIRROR command can then be used on the board layout to put a component on the bottom of the board.
  - Use tPlace for the component silkscreen which is printed on the PCB and should not contact solder.
  - Use tDocu to draw leads of components leading to pads.
  - Use tNames for the >NAME place holder for the reference designator that will be on the silkscreen.
  - Use tValues for the >VALUE place holder, which will not be part of the silkscreen.
  - Use layer 48 Document or layer 49 Reference for any artistic features or notes that will not appear on the silkscreen.
2. Center the package around the origin.

### *Recommendations for Packages*

1. Draw the outline of the package on the tPlace layer with wire width 8 mil.
2. Don't name pads according to function, they should be named by numbers. One possible exception are BGA's whose pads are normally referenced using letters and numbers.
3. The ADD command in the board editor searches through the description field, so a well setup description field can yield more fruitful searches. Therefore a good description field for a package should include:
  - Standard package name(preferably in bold)
  - Any possible aliases for the package. For example a DIL08 package might also be known as a DIP08.
  - If the package is based off of a standard document such as one of the IPC standards, then it should be referenced in the description field and if possible an HTML link should be provided to the document. Listing this information will help in checking the package for correctness.
  - If applicable length x width dimension.

### **Devices**

1. Always set a prefix, otherwise EAGLE will use U\$ by default.
2. When creating multi-gated components, EAGLE will by default reference them as G\$1, G\$2, etc. These names will not show up on the schematic, therefore use the NAME command to rename the gates A, B, C, etc. A nice way to do this is to type 'A' followed by enter when you have the gate on your mouse cursor before placing it, EAGLE will then continue the pattern for

the other gates.

3. Use P for power symbols.
4. When naming devices use specific part numbers.
5. Always double check add levels and swap levels, otherwise your components might not enter your schematic as you'd expect them to.

#### *Recommendations for Devices*

1. Just like packages, having a good description field can improve the chances of the ADD command finding the component you want. A good description field for a device should include:
  - Device name(preferably in bold).
  - Any other aliases the device might be recognized by.
  - A reference and an HTML link(if possible) to the device's data sheet.
  - Manufacturer(s).
  - Short explanation of the device's function.
  - If applicable length x width dimension.