

Managing EAGLE Directories

by James Morrison, Stratford Digital Incorporated

Introduction

This application note describes a good practice in configuring the directory paths in EAGLE. New users can avoid some common pitfalls by following these suggestions.

EAGLE Directories

The EAGLE directory settings are similar to the DOS PATH environment variable. There is a setting for each type of design file associated with EAGLE:

CAM	CAM processor settings
DRU	Design Rules
LBR	Libraries
SCR	Scripts
ULP	User Language Programs
EPF	EAGLE Project Files

By default, these all point to directories within the installation directory of EAGLE.

By adding a user folder to these entries two things are accomplished:

- 1) EAGLE will automatically search through the folder when trying to find a file.
- 2) The folder will be added to the list of folders in Control Panel for easy access.

Don't Edit in Install Directory

A common mistake is to simply use the default settings and work within the installation directory.

There are a few reasons to **NOT** do this:

- 1) In Vista, some users don't have permission to modify these files.
- 2) Uninstalling/upgrading the software will overwrite or delete your design files.
- 3) Version control systems should be used with your design data but not with installed binaries.

Item 2) has caused many users to lose their data.

Configuring the Directories

These settings are configured in EAGLE Control Panel. From the menu choose Options->Directories. This will bring up the dialog box below:

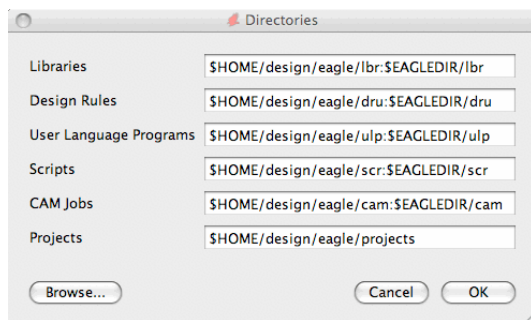


Figure 1 – EAGLE Directories

To change the settings the user simply edits the appropriate text field.

The user can also place the cursor in the appropriate text box and use the "Browse" button to a directory. Note: This will put the new directory at the end of the list and hence the lowest priority. Cut and paste the new entry to the appropriate location in the list.

Delimiter Peril

When using multiple directories a delimiter is used to separate the entries. In Mac and Linux the delimiter is a colon i.e. ":". But in Windows the delimiter is a semi-colon i.e. ";".

Special Variables

EAGLE has two special variables that can simplify the configuration.

`$HOME` – expands to the home directory in Linux and Mac. In Windows it will expand to the HOME

environment variable if set or else it expands to "My Documents".

`$EAGLEDIR` – expands to the base install directory of EAGLE

Recommended Configuration

Stratford Digital recommends adding a user folder as the base for all EAGLE related design files and making it the highest priority.

In Linux and Mac, we recommend using `$HOME/design/eagle`.

In Windows we recommend `c:\design\eagle` (or perhaps another drive).

When `$HOME` is used in Windows (pointing to "My Documents") the path to the files can get quite long. This happens because "My Documents" is actually an alias to a much longer path. Some hardware tools have limitations on the length of the overall path or have issues with spaces in the names of files or directories. To help mitigate these limitations it is often useful to start with a shorter base path name.

As an aside, we highly recommend using complete revision control on this user folder.

It is also useful to keep the EAGLE install directory as the lowest priority in the path to allow for easier access to the many files that come with EAGLE.

See Figure 1 for the recommended settings for Linux and Mac.

Bonus for EET Customers

For customers of the EAGLE Enterprise Toolkit (EET) the newly released installer will configure these settings automatically (if enabled) and copy the latest version of the utilities down from the Stratford Digital server.

Conclusion

By following these recommendations data loss is avoided during product upgrades. Installation files and user data are kept separate while allowing quick access to the user's custom EAGLE configuration and data files.



CadSoft EAGLE 5.0 is a highly integrated and cost-effective CAD tool to design printed circuit boards (PCB's).

Licenses and upgrades can be purchased online in the US and Canada through Stratford Digital www.eaglecentral.ca/purchase/.

For more information about the program or to download the latest version of EAGLE 5.0 go to www.cadsoftusa.com.

Stratford Digital specializes in embedded electronic design. Turnkey Product Design and Engineering Design Services are offered that cover the entire range of embedded technologies: hardware, uController and uProcessor firmware, FPGA and PLD programmable logic using VHDL and Verilog, and all supporting circuitry.

Stratford Digital has been using CadSoft EAGLE for almost 10 years to help design and implement our customers' ideas and products.

After recognizing the great value in the toolset, Stratford Digital became an official CadSoft EAGLE dealer to the US and Canada. Online purchases can be made at www.eaglecentral.ca/purchase/.