# EAGLE Version 6 - Introduction of the Control Panel
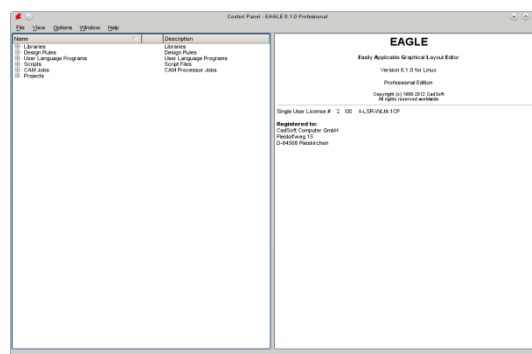
**CadSoft Computer introduced a new major release of their Layout Software EAGLE in December 2011. The new version 6 starts with all the functionality an Electronic Design Engineer needs for drawing schematics and creating a printed circuit board. New is that EAGLE's internal data structure is in XML format, which currently is unique in the professional ECAD software world.**
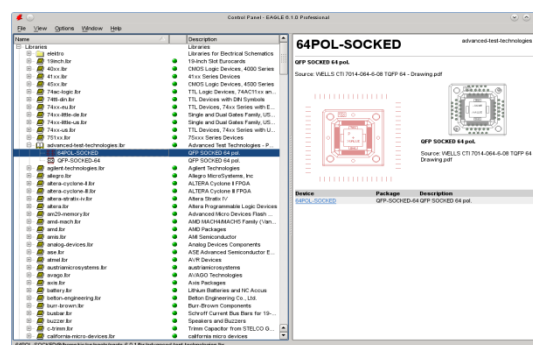


Control Panel -- Tree view

The first article of this series will explain the basics of the EAGLE Control Panel which is the EAGLE starting window. The user can get an overview about the Libraries and the User Language Programs EAGLE comes with and that's the place where you manage existing and new projects. It's also the place where you can decide about a number of program settings.

On the left hand side of the Control Panel you find the so-called tree view. Opening one of the branches, for example "Libraries", shows all the library files that come with EAGLE. Expand one of the entries and you will see the Devices and Packages that are contained in the library file. Click onto one of the entries and the preview of the compo-nent will be shown.



Library preview

The same can be done for "User Language Programs" or, for example, for the "Projects" branch. By the way: User Language Programs (ULPs) are more or less simple, C-like programs that can be used for a variety of tasks. It is possible to export data from your schematic or layout into any format. There is for example "dxf.ulp" which creates DXF data or "bom.ulp" for creating a bill of materials. Importing data is possible as well.
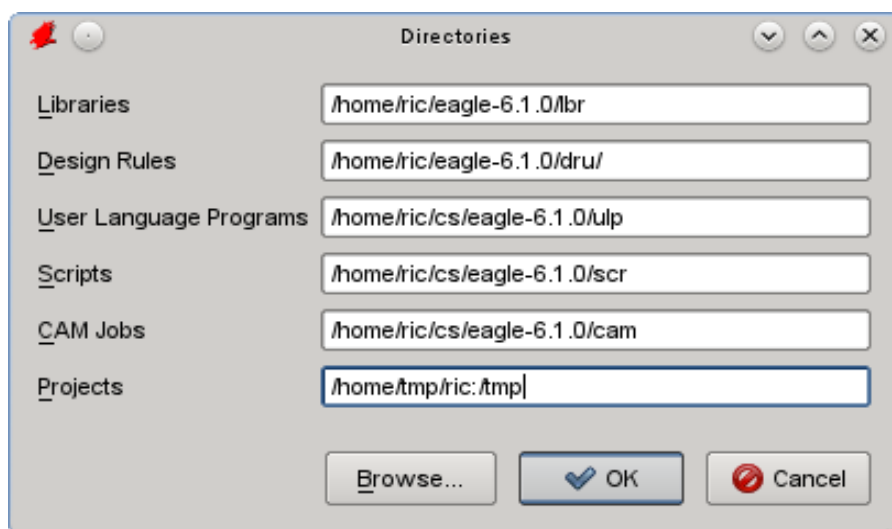
Provided you have a good description of the data format you could write such a program by yourself in order to import data into EAGLE.
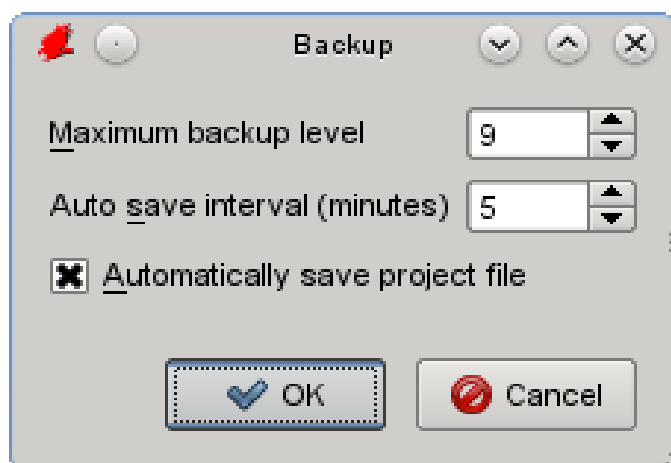
The whole User Language is described in the EAGLE help file. ULPs can be used to modify your project and automize certain tasks. All in all that's a very flexible tool with countless options. Look into the ULPs that come with EAGLE or check for additional files in the Download area of CadSoft web site http://www.cadsoft.de.

On top of the Control Panel there is located the menu bar. The "Options" entry leads you to some interesting configuration possibilities. Click onto the first entry "Directories". Here you can decide about where EAGLE has to look for its files. Libraries, User Language Programs, the Projects, Design Rule files and CAM jobs can be located in any folder, either on your local computer or on a network server. You are allowed to specify several folders, if you like to. Let's say you prefer to have the default libraries available on a server, and your personal ones are located in your private folders.
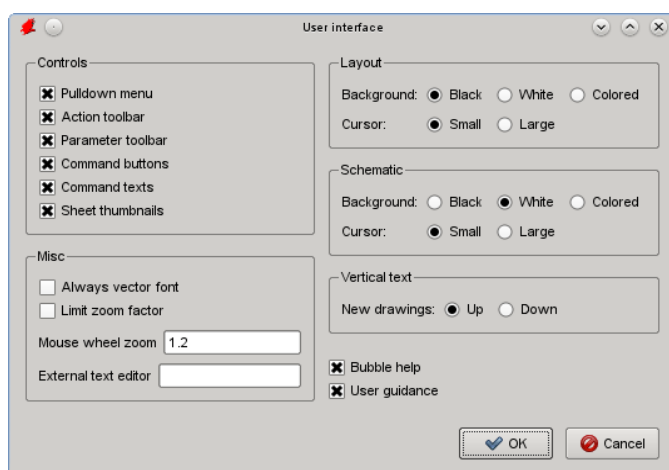


Directories dialog

The second entry in the "Options" menu is about "Backup". EAGLE creates a number of backup files each time you save your project. There is a maximum of 9 possible backups. They are named with the file extension .b#1 to .b#9 for the board file (*.brd). The same for schematics (*.sch) and library files (*.lbr). They get the extensions .s#1 to s#9 and .l#1 to .l#9 accordingly. The automatic backup which is created automtically in the given interval has the extension .b/s/l##. If you have to recover one of the backups you rename it to a valid EAGLE extension.
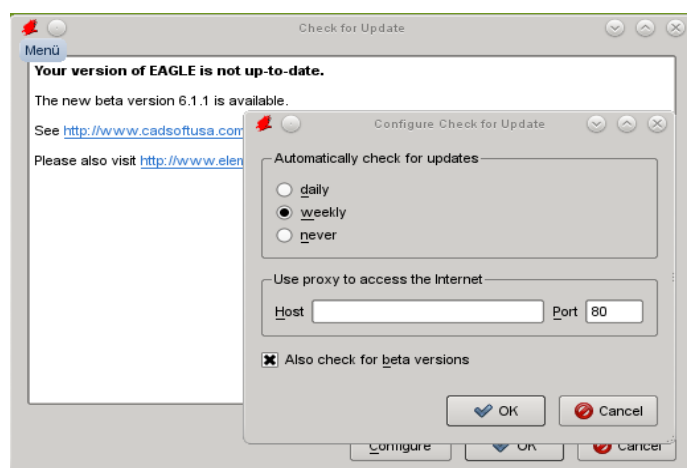
Backup dialog

The "Options/User Interface" menu allows you to change the background colors of the Editor windows or the appearance of the mouse cursor. You can also decide about the toolbars to be displayed or hidden.



User interface

One further interesting setting can be found in the "Help/Check for Update" menu. Here you can control whether EAGLE is allowed to check for newer versions available on the CadSoft webserver. As soon as a new release is available EAGLE will tell you. You can EAGLE let check for beta versions as well. So you can be sure to be always up to date.



Update check

As stated in the beginning, all the basic settings can be made in the EAGLE Control Panel. EAGLE will save all these in your "home" folder in a user-specific configuration file which is named .eaglerc for Linux and Mac OS-X and eaglerc.usr for Windows.

*There are some more options in the menus. Look into them and find out what you can do with the Control Panel. By the way: The EAGLE manual which comes with EAGLE as a PDF file, contains an extra chapter about the Control Panel.*