

EAGLE Version 5 Update Information

=====

This file contains information for users of previous EAGLE versions.
Please read this file entirely if you are updating from an EAGLE version
prior to 5.12.0!

WARNING: Due to some necessary changes in the data structure once you edit
a file with version 5.x you will no longer be able to edit it
with versions prior to 5.0!

PLEASE MAKE BACKUP COPIES OF YOUR CURRENT BOARD-, SCHEMATIC- AND
LIBRARY-FILES BEFORE EDITING THEM WITH VERSION 5.0!

WARNING: AFTER UPDATING ANY FILES PLEASE RUN BOTH AN ELECTRICAL RULE
CHECK

(ERC) AND A DESIGN RULE CHECK (DRC)! YOU MAY NEED TO ADJUST THE
DESIGN RULE PARAMETERS UNDER "Edit/Design rules..." TO YOUR
SPECIFIC NEEDS! SEE ALSO THE REMARKS REGARDING RESTRINGS AND
MINIMUM DISTANCES BETWEEN COPPER AND DIMENSIONS UNDER "Design
Rules"
BELOW!

Release notes for EAGLE 5.12.0

=====

* DISPLAY command:

- The DISPLAY command can now also delete internal layers, as long as they
are empty.

* LAYER command:

- The LAYER command can now also delete internal layers, as long as they
are empty.

* Miscellaneous:

- Dialog CHANGE package/technology: Support of external links and representation
of images.
- ULP function dlgTextView: Proper support for links to local files (open application).

* Bugfixes:

- Group selection by polygon: Avoid selecting too much in special case.
- PRINT command: Transfer options to print dialog if not ended with ';'.
- CAM processor dialog: Avoid settings getting overridden from previously opened
CAM file; Avoid crash after save and opening recent file.
- ULP functions setgroup/ingroup: Bugfix if called for other editor window and no
group defined yet.

- ULP function dlgListView: No sorting for parameter sort=0.
- ULP function dlgComboBox: Avoid jumping combobox size in special case (dlgRedisplay called).
- Fix for wrong REPLACE with package and technology in special constellations.
- Info dialog for arcs/wires: Correct handling of cap style while changing curve to 0/from 0.

Release notes for EAGLE 5.11.0

=====

* User Language, DesignLink:

- The User Language dialog object dlgStringEdit now accepts additional parameters to implement an input history.
- The User Language function system() no longer prompts the user for confirmation before executing the command.
- Modified the User Language network functions so that they use the proxy as set up in "Help/Check for Update/Config".
- DesignLink enhancements: Check price & availability for all parts of a schematic, load a shopping cart (designlink-order.ulp and designlink-inc.ulp, replacing designlink.ulp).
- DesignLink: Removed the "Customer Details" dialog as the user administration is not in service on Element14 web page.
- DesignLink: Added a text export function to order list (designlink-order.ulp) and library tool (designlink-lbr.ulp).
- Added a PCB quote service as ULP (pcb-service.ulp), that can extract manufacturing parameters from the board and send to an Element14 site to get a quote for it.

* CHANGE command:

- The CHANGE command no longer selects airwires when changing parameters that don't apply to airwires (like layer or width).

* EXPORT command:

- Reduced the minimum resolution in the EXPORT IMAGE command to 1.

* MENU command:

- The MENU command can now handle icons.
- The MENU command now automatically turns on the textual command menu.
- By default the textual command menu is now located right next to the action toolbar.
- The default eagle.scr file now contains MENU commands that configure the textual command menu with buttons for DesignLink and the new PCB service.

* SET command:

- The new SET variable CONFIRM can be used to automatically confirm message

dialogs (see "Help/Editor Commands/SET/Automatic Confirmation").

* Miscellaneous:

- If the user has modified the widths of the columns in a `dlgListView`, they no longer fall back to their initial values when a different item is selected.
- The new command line option '-U' can be used to define the location of the 'eaglerc' file in which EAGLE stores user settings. Note that beta versions of EAGLE no longer store their user settings in a separate 'eaglerc' file with the extension '.beta'. They now use the same 'eaglerc' file as officially released versions. If you want to keep things separate for beta versions, use the '-U' option.
- The new SET parameter `Option.AutoLoadMatchingDrawingFile` can be used to suppress the query "Do you also want to load ...?" in the EDIT command.
- The new SET parameters `Cmd.Name.RenameEntireNetByDefault` and `Cmd.Name.RenameEntireSignalByDefault` can be used to control the default behavior of the NAME command.
- Support relative paths of image links in libraries in ADD and REPLACE dialog.

* Bugfixes:

- Fixed the layer sequence in the PRINT dialog's preview in case the output is mirrored.
- When opening an existing project, windows that were minimized when the project was closed, and were previously maximized, will now behave correctly when clicking on their taskbar entry.
- Fixed a crash when executing a script file with several "edit ...; close;" sequences in the same line.
- Fixed wrongfully trying to open the device links in the Control Panel as external hyperlinks when resizing the Control Panel.
- Fixed showing an invoked gate in the INVOKE dialog.
- Fixed canceling a GROUP/MOVE between sheets.
- Fixed handling frames in the EXPORT SCRIPT command.
- Fixed exporting partlists with the EXPORT command in case the part value contains "%1".
- Fixed handling external hyperlinks in the ADD dialog.
- Under Linux the runtime libraries `libssl` and `libcrypto` are now linked statically to the executable file, because on some systems the required version of these files was not available.
- Fixed converting lowercase to uppercase characters when renaming a package variant via the dialog.
- Fixed alphanumeric sorting of strings.
- Fixed setting the sort order in the INVOKE dialog.
- Fixed recognizing a CAM job as "modified" if only the name of a job section has been modified.
- Fixed the values of `UL_VIA.diameter` and `UL_VIA.shape` for supply layers in which the via is connected, in case the Design Rules parameter "Supply/Generate thermals for vias" is turned off.
- Fixed limiting stored dialog sizes to smaller desktops.

- Fixed handling leading or trailing blanks when the user enters a Freemium code.
- Fixed missing spaces after '-' between <nobr> tags.
- The User Language function ingroup() now checks whether there is actually a group defined.
- Export library as script: Handle single quotes correctly when used within attribute values.
- Fixed displaying the large crosshair cursor at the current mouse position when turning it on.
- Fixed retrieving the parameters of the PRINT dialog from the eaglerc settings, so that cfgset() can properly set them.
- Fixed handling UL_HOLE.diameter[], UL_PAD.diameter[], UL_PAD.shape[], UL_VIA.diameter[] and UL_VIA.shape[] in case the board contains supply layers that are not visible when the ULP runs.
- Avoid erroneous change of width of ULP comboboxes.
- Fix performance issue with big schematics with many device attributes.
- Fix asymmetric alignment of texts in xref labels with frame.
- Fixed: New attributes defined in schematic did not get current rotation of element when propagated to board.
- Fixed crash when moving Thunderbird email files across EAGLE windows by drag&drop.
- Use actual wire width for line creation in layer 'unrouted', not zero.
- Suppress hidden airwires in group selection.
- Fixed a missing cell padding in the Technologies/Attributes display of the library editor under Windows.
- Fixed point-in-polygon algorithm for group selection by polygon.
- Enable display of web based images in various description fields and HTML editor.
- Fixed crash when doing a show on a part instance from other than current sheet.
- Improved performance of DRC with selected area.
- Fixed distortion in printing of mirrored circles.
- Bugfix: Make GROUP on signals excluded with RATSNEST work again.

Release notes for EAGLE 5.10.0

=====

* Internationalization:

- The manual and tutorial are now available in Chinese.
- The EAGLE program texts have been translated to Hungarian (note that the texts provided by the Qt GUI library are not available in that language).
- The EAGLE program texts have been translated to Chinese (note that the texts provided by the Qt GUI library are not available in that language).

* User Language:

- The new User Language functions neterror(), netget() and netpost() can

be used to access remote sites on the Internet.

- The User Language function `t2string()` now has an optional format parameter.
- The User Language now provides functions for processing XML code (see "Help/User Language/Builtins/Builtin Functions/XML Functions").
- The User Language function `language()` now by default returns "en" if no valid language code can be determined.
- The new User Language function `country()` can be used to determine the country code of the current system (see "Help/User Language/Builtins/Builtin Functions/Miscellaneous Functions/country()").
- The image URL in the `<img...>` tag of HTML texts may now also be external (``).
- The new User Language dialog function `dlgSelectionChanged()` can be used to determine whether the current selection in a `dlgListView` or `dlgListBox` has changed.
- The new User Language Program 'designlink.ulp' can be used to access Premier Farnell's online product database.
Note that currently the DesignLink server has problems delivering more than 50 search results at a time, and can only deliver the first 500 search results, even if it reports more than 500 hits for any given search.
- The User Language dialog objects `dlgTextView` and `dlgLabel` now open external hyperlinks with the appropriate application program.
- The width of a `dlgComboBox` is now automatically adjusted when the data in its string array is changed.
- The new User Language functions `cfgget()` and `cfgset()` can be used to store parameters in the user's `eaglerc` file.
- The 'Selected' parameter of a `dlgListView` can now be initialized to -2 to make the first item according to the current sort column the selected one.

* Miscellaneous:

- Reverted "Fixed calculating mask data for SMDs with a non-zero roundness" from version 5.7.0, because this caused apertures to be drawn that used to be flashed.
- Hyperlinks contained in the descriptions of files or library objects are now opened with the appropriate application program.
- Removed the remark on `t/bPlace` automatically selecting `t/bNames` etc. from the English version of the online help of the `DISPLAY` command (this was overlooked in version 4.92.3).

* Bugfixes:

- Fixed setting the text orientation when adding a new attribute via the command line.
- Fixed adding the default ".scr" extension in `EXPORT NETSCRIPT`.
- Fixed detecting objects inside hatched polygons that would "fall through" the hatch lines.
- Fixed generating the `eaglecon.exe` file on Windows systems where a realtime virus scanner blocks patching executable files.
- Fixed a crash under Linux if the mouse cursor is inside a combo box in

the parameter toolbar, and a new command is started, so that the content of the parameter toolbar changes.

- Fixed alphanumeric sorting by file type in the tree view of the Control Panel.
- Fixed marking the current sort mode in the Control Panel's "View/Sort".
- Fixed showing an invoked gate when double clicking on it in the INVOKE dialog.
- Fixed displaying the list items of already invoked gates in the INVOKE dialog with a dimmer color.
- Fixed wrongfully assigning generated names to buses when doing GROUP/CUT/PASTE.
- Fixed updating the Control Panel's tree view if an Autorouter control file is saved to disk.
- Fixed drawing artifacts when moving a part in the schematic that has an attribute with a value that is longer than the related placeholder text.

Release notes for EAGLE 5.9.0

=====

* Bugfixes:

- Fixed generating the default board outlines in the BOARD command of the Professional edition.
- Fixed an endianness problem with the Freemium edition on PPC Macs.
- Fixed an unjustified "Load error 295" when loading a schematic with more than one sheet that has been created with the Freemium edition.

Release notes for EAGLE 5.8.0

=====

* Platforms:

- The minimum system requirement for the Mac version of EAGLE is Mac OS X 10.4 (this was forgotten in the version 5.7.0 release notes).

* License:

- CadSoft now provides an EAGLE "Freemium" license, which is valid for a limited time, and is more powerful than the Freeware license. See <http://www.element-14.com/eagle-freemium> for more information.
- The Standard edition of EAGLE can now handle 6 signal layers.
- The "Help/EAGLE License..." option now allows the user to choose between using a customized license file, the Freeware or the Freemium license.

* POLYGON command:

- Added a note to the online help of the POLYGON command, recommending to leave the Isolate parameter at 0 to avoid problems during manufacturing.

* PRINT command:

- The PRINT command now always sets the default output file name to that of the currently loaded file when printing to a PDF or Postscript file.

* SET command:

- SET SNAP_LENGTH now accepts units.

* TEXT command:

- Added a note to the "Attributes" section of the online help for the TEXT command, saying that there should be only one placeholder text per attribute name in any given symbol or package.

* Miscellaneous:

- When a board is loaded, any leftover empty signals are now deleted automatically.
- The total thickness of the layer setup in the Design Rules is now displayed with the unit defined in SET Interface.PreferredUnit.

* Bugfixes:

- Fixed handling UL_VIA.diameter[] for LAYER_TSTOP and LAYER_BSTOP, in case the via's drill is not greater than the value defined in "Design Rules/Masks/Limit" and the via's VIA_FLAG_STOP is set (in which case 0 was returned instead of the actual diameter).
- Fixed making sure that every #include file in a ULP is processed only once.
- Fixed a possible crash with large Autorouter jobs on Mac OS X.
- Fixed handling existing exclamation marks in names when updating files from versions before 5.0.
- Fixed using the selected technology in the REPLACE command.
- Fixed wrongful unsmashing of parts on the source sheet after moving a group to an other sheet.
- Fixed setting the used signal layers after creating a new board from a schematic.
- Fixed reactivating snapping after starting a transparent command while the GROUP command is active.
- Fixed EXPORT IMAGE in monochrome with colored background.
- Fixed redrawing attributes if only their name has been changed.
- Fixed a crash when zooming far in to a mirrored text with non-vector font on Windows.
- Fixed displaying combo boxes on Windows with "extra large fonts".
- Fixed wrongfully adding a wire to an existing signal if the wire is placed at a point where a connected SMD pad exists on a different layer.
- If the ATTRIBUTE command is used with the invalid syntax
ATTRIBUTE <partname> <attributename> (x y);
a "Missing attribute value" error is now given.
- Fixed a missing confirmation in the schematic when changing the value of an attribute that has the 'constant' option set in the library.

- Fixed a possible crash when doing an UnSmash on the last element in a board that contains no signals, and where the element has attributes.
- Fixed artifacts in combo box lists on Mac OS X 10.6.
- Fixed a possible crash when printing to a PDF file on Mac OS X 10.4.
- Fixed displaying local attributes in the schematic in case there is a global attribute with the same name.
- Fixed sluggish performance, especially after UNDO/REDO with schematics that contain many sheets.
- Fixed printing on Mac OS X 10.6 with some printer drivers (for instance the "HP Laserjet 5100 Series").
- Fixed a memory leak in case a toolbar is wider than the editor window.
- Fixed a possible crash when starting a command in the schematic editor, doing a SHOW in the board editor, and finally continuing the command in the schematic editor in such a way that a forward annotation changes the data in the board editor.
- Fixed handling signal polygons in case a net that is present only on a single sheet is completely renamed.
- Fixed quoting technology names in EXPORT SCRIPT.
- Fixed switching the "OK" button to "Show" in the INVOKE dialog when clicking on a gate that is already invoked.

Release notes for EAGLE 5.7.0

=====

* Platforms:

- As of this version, every EAGLE license is valid for all supported platforms, without any more additional fee.

* Control Panel:

- The "View/Refresh" option in the Control Panel's pulldown menu now has the platform specific hotkey.
- The default action when double clicking, pressing Space or Enter on a project directory in the Control Panel's tree view is now to open/close the project.

* User Language:

- The new User Language functions setgroup() and clrgroup() can be used to set and clear the flags that define whether an object is within the current group.
- It is now possible to directly access members in an array of UL_* objects, as in

```
UL_SMD smds[];
...
if (smds[i].x == smds[j].x)
...
```

* ADD command:

- The search string in the ADD dialog is no longer modified if the ADD command is called with a unique device name (from the command line or the right mouse button history function of the ADD button).

* AUTO command:

- The AUTO command now supports selecting signal names by wildcards.

* BOARD command:

- Added a consistency check right after a board is newly created from a schematic, in order to immediately catch any problems that otherwise might have turned up later on.

* CUT command:

- The CUT command now clears the paste buffer in case the group is empty.

* DISPLAY command:

- The DISPLAY dialog now has an 'Apply' button.

* DELETE command:

- Pads/smds can now be deleted from a package in a library, even if the package is in use by a device, as long as the pad/smd is not referenced by any pin in any device of that library.
- Pins can now be deleted from a symbol in a library, even if the symbol is in use by a device set, as long as the pin is not referenced by any pad/smd in any device of that library.

* GROUP command:

- Made GROUP ALL also select objects outside the defined coordinate area.

* LABEL command:

- The coordinates used in xref labels no longer refer to the surrounding rectangle of the net, but rather take each object into account separately.

* MARK command:

- The MARK command now immediately updates the grid display, without the need to first move the mouse.

* NAME command:

- The NAME command now only offers objects that actually have a name in case

there are several objects within the select radius.

* OPTIMIZE command:

- The OPTIMIZE command no longer takes airwires into account.
- The OPTIMIZE command now flashes the selected signal to provide visual feedback to a mouse click.

* PAD command:

- The PAD command can now add a pad to a package, even if that package is in use by a device.

* PIN command:

- The PIN command can now add pins to a symbol, even if that symbol is in use by a device set.

* PRINT command:

- The PRINT dialog now contains a list of all available printers for direct selection. Printing to a PDF or Postscript file can also be selected from this list.
- The PRINT command now remembers if the last print went into a file.
- The PRINT command has a new option named PRINTER, which can be used to print to a specific printer.
- The new option CAPTION of the PRINT command can be used to print a caption from the command line.
- The new option PAPER of the PRINT command can be used to print to a given paper size.
- The file name in the PRINT FILE option may now contain placeholders to insert the actual drawing file name or extension.
- The PRINT FILE command now refuses to write any of the file extensions .brd.sch.lbr.gpi.dri.whl.drl.ulp.epf.cam.scr.exe.com.bat.cmd.sys.dll, to make sure files are not overwritten inadvertently.

* RATSNEST command:

- The RATSNEST command no longer stops when it encounters an unknown signal name, but rather tells the user about the unknown signals and offers the choice of continuing.
- The RATSNEST command, if given a signal name, now only processes the polygons of the given signal, not all polygons of all signals.
- The context menu of wires and vias that belong to a signal now contains the RATSNEST command.

* ROUTE command:

- The ROUTE command no longer activates the current layer when it is started, but waits until an airwire has actually been selected.

- The Follow-me router now uses the via diameter as set in the parameter toolbar.

* SET command:

- When entering values for the Catch_Factor or Select_Factor in the SET command via the command line, numbers in the range [1 ... 100] are now taken as percentages to avoid errors caused by not entering the factor as a decimal number between [0.0 ... 1.0].
- The new parameter Option.LayerSequence can be used to define the sequence in which layers are rendered (see "Help/Editor Commands/SET").

* SHOW command:

- The SHOW command with the '@' option now also draws the pointer rectangle in the other editor window, if f/b annotation is active.
- The highlighted objects from the SHOW command's dialog now stay highlighted, even after closing the dialog (until the end of the SHOW command).

* SMD command:

- The SMD command can now add an smd to a package, even if that package is in use by a device.

* UPDATE command:

- The UPDATE command now allow selecting multiple files from the file dialog.

* USE command:

- The USE command now allow selecting multiple files from the file dialog.

* VALUE command:

- The VALUE command button now also has a history function.

* VIA command:

- The VIA command now issues an error message if there are no vias defined in the layer setup.

* CAM Processor:

- Added CAM Processor driver GERBER_RS274X_25 with 2.5 inch coordinate format.
- The default CAM jobs no longer have the "Mirror" option checked.

* Miscellaneous:

- The sorting sequence of numeric strings that differ only in the number of leading zeros in their numeric parts has been made stable (i.e. sorting them more than once always leads to the same sequence).
- The Properties dialog of a wire now also displays the angle of the wire.
- The progress display of the CAM Processor no longer keeps popping into the foreground.
- Improved the performance of recalculating the ratsnest in the board after an ADD, DELETE or PASTE command in the schematic, in case many pins are processed that are connected to the same net.
- Made the origin crosses for elements in a board smaller, so that they don't overlap that much in case of small SMD parts.
- The Properties dialog of an element now applies modified coordinates and orientation settings even if the "Locked" flag is checked.
- Added a general note to the online help about the special coordinate value '@'.
- Leading and trailing blanks in file and directory names are now ignored.
- When defining a new drill diameter in the "Options/Set/Drill/New" dialog, the default unit is now the same as in the editor window's grid.
- Fixed handling empty keys in bom.ulp.
- For better readability the sheet thumbnails now show only the individual sheet number, without the total number of sheets.
- Speeded up handling parts with lots of attributes in the schematic.
- The Design Rules plausibility check now tests whether the clearance values for 'same signal' are not larger than those for 'different signals'.
- The `_OUTLINES_` polygon (if present) is now calculated with increased precision, to allow the milling contours to better follow the object outlines.
- The detection of connected parts of a calculated polygons (which is used, for instance, in the RATSNEST and ROUTE commands) has been speeded up.
- Removed an unnecessary dependency of libXft.so in the Linux version.
- The new SET parameter `Interface.MouseButtonReleaseTimeout` can be used to set a timeout within which a mouse button release will trigger a button's action, even if the release happened outside the button's area (default is 500ms).
- The color index in the dxf.ulp now starts at 1, since AutoCAD 2007 doesn't like color 0.
- The CAM Processor's warning about polygons that may produce very large plot data is now only given if the layer containing that polygon will actually be plotted.
- The Value entry in the Properties dialog of a part where the device has "Value off" now contains a check box that indicates that this value overwrites the default (i.e. the device name). When this box is unchecked, the value falls back to the default.
- When opening a project, the windows that get opened are no longer scaled to the current desktop size, but rather moved and resized if necessary to make them fit entirely onto the desktop.
- Switching between windows with Alt+0, Alt+1 etc. has been disabled on the Mac OS X platform, because on some localized keyboard layouts these key combinations are already used otherwise.
- Implemented the placeholders >SHEETNR and >SHEETS to make the sheet number

and the total number of sheets in a schematic available separately.

- Added a note to the online help about selecting the layer in the ATTRIBUTE and LABEL commands.
- When a schematic is loaded, any leftover empty nets are now deleted automatically.
- If a board file that has been created with an EAGLE edition that can handle only a subset of the routing layers is loaded with an edition that can handle additional routing layers, and the layer setup is changed so that additional layers are used, these layers are now created automatically.
- Improved performance of panning outside the surrounding rectangle of the entire drawing.

* Bugfixes:

- Fixed missing technology when adding string to the command button history of the REPLACE command.
- Fixed a crash with negative 'number' parameter in the User Language function sort().
- Fixed handling links like in dlgTextView.
- The "Link" variable in dlgTextView can now be used to initially scroll to a given position in the text.
- Fixed sorting in the INVOKE dialog.
- Fixed handling links like in the Control Panel.
- Fixed restoring the width of the sheet thumbnail view if the view is docked.
- Fixed handling the minimum drill distance in the Autorouter.
- Fixed refreshing the draw window if color palette entries are changed via the SET command.
- The modification "Fixed dynamic airwire calculation in the ROUTE command in case layers containing objects that belong to the routed signal are not displayed" that was done in version 5.2.4, has been revoked because it caused segments of a signal that are connected through wires in the undisplayed layer not to be seen as connected, which resulted in an incorrectly calculated dynamic airwire.
- Fixed redrawing polygons that extend outside the board area when the Autorouter is started.
- Fixed artifacts in case a smashed part that contains the placeholders >PLOT_DATE_TIME, >LAST_DATE_TIME or >DRAWING_NAME is moved.
- Fixed the paper size values in the PRINT dialog.
- Fixed skipping airwires when pasting from a board into a package drawing.
- Texts and rectangles in user defined layers of a board or package, that have a non-orthogonal angle, are now skipped when pasting them into a sheet or symbol drawing.
- The parameters of the editor window no longer fall back to defaults when loading a different drawing.
- Fixed handling signal polygons in the Autorouter.
- Fixed an unjustified warning message "The attribute '...' is already defined in the board with value '...'! Overwrite?".
- Fixed UL_PART.attribute[] in case the actual name of the attribute is

- longer than the string given as the index, as in "ABC" vs. "ABCDE".
- Fixed a possible loss of consistency after GROUP/CUT/PASTE in case a part on a schematic sheet is in the group, but the net wire connected to one of its pins isn't. After deleting some other wires from that net the consistency between board and schematic may have been lost.
 - Fixed adding the "Cancel" button to the message box in case a script is run in a library and nothing has been selected for editing.
 - Fixed calling the User Language function strxstr() without any optional parameter.
 - Fixed a possible crash in case a net wire completely disappears when moving one of its end points onto the other.
 - Fixed the User Language function strtol(), so that hex values above 0x7FFFFFFF are converted to negative integers.
 - Fixed a crash in the Follow-me router when changing the grid and the current layer while an airwire is attached to the cursor.
 - Fixed handling lowercase characters in placeholder texts for attributes in the SMASH command.
 - Fixed handling the '.tif' extension in the EXPORT IMAGE command.
 - Fixed handling nested typecasts in User Language programs.
 - Fixed handling wire bend styles in the ROUTE command without autorouter module consistently in case right mouse button clicks and clicking on the bend style buttons in the toolbar is mixed.
 - Fixed an unnecessary extra automatic backup if the automatic backup is triggered in one editor window, while a lengthy command is running in the other editor window.
 - Fixed a possible data corruption if the automatic backup is triggered in one editor window, while a lengthy command is running in the other editor window.
 - Fixed displaying script comments containing UTF-8 characters in the Control Panel.
 - Fixed handling "dead keys" under Windows.
 - Fixed a crash in the Follow-me router when selecting a new airwire with Ctrl-Left mouse button.
 - Fixed a possible crash when using the file dialog after closing a project.
 - Fixed calling UL_ATTRIBUTE.defaultvalue for attributes that are defined in the library from within a schematic in case a project is open.
 - Fixed calculating the pointer rectangle in the SHOW command in case the shown net segment contains only pin references (no wires, labels or junctions).
 - Fixed pasting smashed parts with attributes in the schematic (the attribute value was lost in such cases).
 - Fixed the ERC to detect nets with only one pin connected to them, in case the only other pin types connected to that net are supply pins that have no pad.
 - Fixed handling backslashes and exclamation marks in part names and values when updating from versions before 5.0.
 - Fixed a crash on Mac OS X when leaving the program while the CAM Processor window is open and has keyboard focus.
 - Fixed the library update in case only attributes with empty values have been modified.

- Fixed a hangup when pressing the right mouse button inside an entry field that checks its contents and currently contains invalid data.
- The EXPORT IMAGE command no longer leaves a zero length file in case it runs out of memory.
- The RIPUP command no longer selects objects it can't rip up.
- The CAM Processor message "Package '...' of element '...' contains a polygon that may cause extremely large plot data" had the package and element name reversed.
- Fixed a loss of the sheet thumbnail view if temporarily changing to another virtual desktop under GNOME.
- Fixed a problem with REPLACE in the board if the schematic has been closed immediately before from within a script.
- Fixed handling newlines when saving *.cam files.
- Fixed handling supply layers in case a board is edited that contains supply layers, and then another board is loaded that doesn't contain one or more of these layers.
- Fixed disappearing group polygon lines in case the WINDOW command is used while a group polygon is being drawn.
- Fixed handling the .dev, .pac and .sym extensions if they are entered in the EDIT dialog of the library editor.
- Workaround for a problem with constructing a maximized window initially minimized on Windows.
- Fixed calculating the bounding rectangle of xref labels.
- Fixed the DISPLAY LAST command in case the previous DISPLAY command didn't actually change anything.
- Fixed drawing artifacts with SMDs in case the Cream frame mask parameter is not zero.
- Fixed overlapping texts in the LayerSetup dialog in case only the Top layer is active.
- Fixed handling negative array indexes in User Language programs.
- Fixed calculating mask data for SMDs with a non-zero roundness.
- Fixed a possible data corruption in case a script contains a CONNECT command followed by a REMOVE command (without an intermediate EDIT command).
- Fixed optimizing duplicate overlapping wires in case there are pads or vias at both ends.
- Fixed optimizing a short wire that is overlapped by a longer wire at a point where a third wire ends.
- Fixed the online help for the User Language statement 'while'.
- Fixed wire coordinates in UL_PIN.texts().
- Fixed relocating wires and vias in the CHANGE PACKAGE command.
- Fixed detecting clearance errors between round pads, vias or smds if the minimum clearance value is larger than 4.634mm.
- Fixed wrong z-order after re-docking a floating toolbar under Linux.
- Fixed resizing floating parameter toolbars if their contents changes.
- Fixed setting the width of the columns in the attribute list of the properties dialog and in the change package dialog.
- Now removing duplicate PinRefs and Connects when loading a schematic or a board, respectively.
- Fixed a superfluous UNDO step after loading another schematic from

within the schematic editor, or creating a new sheet in an existing schematic.

- Fixed handling error conditions in case the PRINT command tries to write to a file that is currently open by another application.
- Fixed updating the surrounding rectangle of the source sheet when moving a part from one sheet to another.
- Fixed displaying layer colors and fill styles in dialogs on Ubuntu-Linux (problem observed on Ubuntu version 9.10).
- Fixed drawing the large cursor on Ubuntu-Linux with desktop transparency effect (problem observed on Ubuntu version 9.10).

Release notes for EAGLE 5.6.0

=====

* Bugfixes:

- Fixed RIPUP to make it stop at pads when clicking on an airwire (was broken since version 5.2.4).
- Fixed a possible "Abort: Unknown objectType..." in the SIGNAL command.
- Fixed a bug in the Autorouter that sometimes caused DRC errors at pads, vias and SMDs.
- Fixed the status message after a DRC or ERC, which sometimes wrongfully contained the word "approved".
- Fixed a possible inconsistency of net classes after renaming all segments of a net on the current sheet.
- Fixed a possible inconsistency of net classes after pasting parts with power pins that automatically create new signals in the board.
- Fixed handling cfPad/SmdImpact in the Autorouter.
- Fixed terminating a track in the Follow-me router by clicking twice at the same point.
- Fixed an unexpected offset of the airwire when clicking at an unroutable point in the Follow-me router.

Release notes for EAGLE 5.5.0

=====

* User Language:

- The new function strxstr() can be used to search strings using regular expressions in User Language Programs.
- Added a note to the online help for UL_DEVICE about using the 'package' data member to check whether this device has a package.

* Design Rule Check:

- The DRC no longer issues a "Drill Distance" error for vias that have only one layer in common, e.g. 1-2 and 2-3 ("stacked vias").
- The Design Rules dialog now issues a warning if the layer setup uses any layers that are not available in the licensed program edition.

* Text editor:

- It is now possible to use an external text editor instead of EAGLE's built in text editor (see "Help/Editor Windows/Text Editor").

* AUTO command:

- The new option FOLLOWME in the AUTO command opens the Autorouter dialog in a mode where only the parameters controlling the Follow-me router can be modified.
- Routing from and to pads, smds and vias in the Autorouter has been improved.

* CHANGE command:

- The CHANGE PACKAGE/TECHNOLOGY command no longer overwrites the value of parts where the device has "value off", but the user has overwritten the value (which is detected by comparing the old value to the name of the old device).

* CONNECT command:

- Improved handling the width of the connection list in the CONNECT dialog.

* DRC command:

- The DRC command no longer automatically brings up the ERRORS dialog if there are only approved errors.

* ERC command:

- The ERC command no longer automatically brings up the ERRORS dialog if there are only approved errors/warnings.
- The ERC message "OUTPUT and ... pins mixed on net ..." has been enhanced to show the actual direction of the referenced pin, so that the offending pin can be easily located.

* EXPORT command:

- The new option WINDOW of the EXPORT IMAGE command can be used to export an image that contains only the currently visible window selection of the drawing.

* PRINT command:

- The new option SHEETS of the PRINT command can be used to print a given range of sheets from the command line.
- The new option WINDOW of the PRINT command can be used to print only the currently visible window selection of the drawing.
- The new options PORTRAIT and LANDSCAPE of the PRINT command can be used to

define the orientation of the resulting output.

* RENAME command:

- The RENAME dialog now contains the current name of the object, so that it is easier to make small changes.

* REPLACE command:

- The REPLACE command no longer overwrites the value of parts where the device has "value off", but the user has overwritten the value (which is detected by comparing the old value to the name of the old device).

* ROUTE command:

- The ROUTE command can now be used in "Follow-me" mode, where the Autorouter is used to automatically route the selected airwire according to the current Design Rules and Autorouter parameters. Note that you need to have the Autorouter module enabled in your EAGLE license in order to use the Follow-me router.
- The ROUTE command now automatically sets the wire width and via drill according to the values defined by the Design Rules and net classes if the new flag "Options/Set/Misc/Auto set route width and drill" is set.

* SET command:

- The new wire bend styles '8' and '9' are used to control the behavior of the Follow-me router. '8' means route only the short end of the selected airwire, while '9' routes both ends.

* SHOW command:

- If the '@' character is given in the command line of the SHOW command, a pointer rectangle is now drawn around the shown object. This is helpful in locating small objects that wouldn't show up too well just through highlighting.
- If an object given by name in the SHOW command is not found on the current schematic sheet, a dialog is now presented containing a list of sheets on which the object is found.

* UPDATE command:

- The library update no longer overwrites the value of parts where the device has "value off", but the user has overwritten the value (which is detected by comparing the old value to the name of the old device).

* Miscellaneous:

- Improved performance of approving ERC/DRC errors.

- Speeded up clearing the ERRORS list when clicking on the "Clear all" button of the dialog.
- Modified the way EAGLE writes its files in order to avoid data corruption on broken NAS (Network Attached Storage) systems.
- Speeded up calculating signal polygons.
- The packages of parts that are added to the schematic are now placed on the board using the current grid.
- Speeded up filling the ERRORS list after running the DRC/ERC.
- The "File/Save all" function now also saves the eaglerc file.
- Some users don't want the warning message about a supply pin overwriting a generated net name. The command

SET Warning.SupplyPinAutoOverwriteGeneratedNetName 1;

can now be used to disable that warning.

- Added a note to the "Stop" checkbox in the properties dialog of a via, telling the user that this is not applicable if the drill diameter of the via exceeds the value defined in "Design Rules/Masks/Limit").
- When a supply pin is placed on a net segment, the user is now asked whether the net segment shall be renamed to the name of the supply pin.

* Bugfixes:

- Fixed checking for balanced pairs of '{' and '}' in the MENU command.
- Fixed updating the via length combo box if the layer setup is modified while the VIA command is active.
- Fixed displaying technology names in the device editor's Description pane in case they differ only in leading zeroes of their numeric parts.
- Fixed marking a drawing as modified if a global attribute is changed.
- Added missing User Language constants FRAME_BORDER_*.
- Fixed changing the layer of a polygon in a package or symbol (it wouldn't change between signal and non-signal layers).
- Fixed a crash when doing a Drag&Drop from the WinRAR program under Windows (dropping still doesn't work from WinRAR, but at least EAGLE won't crash any more).
- Fixed updating the path names of items in the Control Panel's tree view if the directory in which they are contained is renamed from within the tree view.
- Fixed handling the 'catch factor' in case it is smaller than the 'select factor'.
- Fixed handling the indicator lines if a smashed text of a locked part is contained in a GROUPE MOVE.
- Fixed updating the via diameter display in the Properties dialog when changing the drill diameter and clicking on 'Apply'.
- Fixed updating the tree list in the ADD, REPLACE etc. dialog when reopening such dialogs after modifying a library from outside of EAGLE.
- Fixed printing pad shapes in inner layers.
- Fixed drawing drill holes of pads and vias (they were drawn one pixel too small in radius).
- Fixed unjustified ERC warning "Segment of net ... has no visual connection

- (like Label, Bus or Supply pin) to other segments of the same net" in case of empty segments (which sometimes were left over from older versions).
- Fixed sticky status messages like "DRC:".
 - The layer setup tab in the Design Rules dialog now displays the total thickness of the board by adding up all copper and isolate values.
 - Fixed a possible loss of consistency when deleting a junction from a pin.
 - Fixed displaying status messages in the text editor.
 - Added missing constants LAYER_INFO and LAYER_GUIDE to the online help page for UL_LAYER.
 - Fixed using the MIRROR and ROTATE command after defining a new group while one of these commands is already active.
 - Fixed handling the tag in the description editor and in the ADD dialog.
 - Fixed an occasional problem with renaming temporary files on Windows in case the new file name already exists.
 - Fixed keeping approved DRC errors when clicking on the "Clear all" button in the ERRORS dialog.
 - Fixed initial display of a pasted group at the mouse position in case the PASTE command was entered from the command line with an orientation.
 - Fixed showing the current angle in the Angle combo box of the parameter toolbar if the value has been entered in the command line and is not one of the default orthogonal values.
 - Fixed a crash when clicking with the MITER command on a wire of a polygon that consists of only three edges.
 - Fixed generating technology and attribute lines in EXPORT SCRIPT for devices that have no package (like frames etc.).
 - Fixed detaching a pin from a net in case one net wire ends at the pin, while another net wire of the same net segment passes over the pin's connection point, and the wire ending at the pin is deleted. In such a situation, the pin was left connected to the net, and when it was later connected to a different net, the corresponding pad in the board was connected to two different signals.
 - Fixed the online help of the <a> HTML tag (it allows only links within the same document).
 - Fixed approving DRC errors in the ERRORS dialog in case some errors are currently not displayed, because their layer is hidden.
 - Fixed a crash in the Mac version in case a device was loaded in the library editor, the "EAGLE" pulldown menu was opened and the mouse cursor was moved over the (wrongfully disabled) "Quit EAGLE" item.
 - Fixed toggling the group membership of objects when clicking on them with the Ctrl key pressed.

Release notes for EAGLE 5.4.0

=====

* Miscellaneous:

- Speeded up the window refresh in the schematic editor in case there are many parts with lots of attributes that don't display their value.

* Bugfixes:

- Fixed detecting faulty nameless attributes for parts in case a part is copied and there have been attributes deleted before that.
- Fixed a crash in the DRC with zero-width arcs in the t/bRestrict layers.
- Fixed checking zero-width straight wires in the t/bRestrict layers.
- Fixed changing the net name of every segment on the current sheet in case the same net is also present on other sheets.
- Fixed handling UNC paths in Options/Directories.
- Fixed selecting multi-gate parts by name in the NAME command (the old name had to be given as a full gate name, e.g. IC1A, instead of the basic part name, e.g. IC1).
- Fixed drawing artefacts when moving a part in the board that has pads or smds with long names, and displaying the pad names is activated.
- Fixed a crash in the User Language function fileglob() in case the given pattern string is empty.
- Fixed a duplicate confirmation when using the User Language function dlgFileSave().
- Fixed clearing the status bar in the SHOW command when clicking on an object that has no name after highlighting a named object.
- Fixed handling zero length airwires when moving a group.
- Making sure the actual via length used in the VIA command is one of those defined in the layer setup.
- Fixed superfluous airwires when checking whether a wire is connected to a polygon.
- Fixed calculating polygons with a large Width that are connected to pads with thermals.
- Fixed faulty backannotation when changing an attribute of an element in the board where the corresponding part in the schematic doesn't have that attribute.
- Fixed handling pin names with @nn when CUT/PASTEing.
- Fixed a crash when entering 'CLASS -8'.
- Fixed clearing the "Updating libraries from paste buffer" status message in the PASTE command.
- Fixed invoking instances after CUT/PASTE in case they were smashed and were not part of the original group.
- When loading a schematic or library file from an earlier version, the new layers 97 and 98 are now automatically added.
- Fixed handling the drill holes of blind/buried vias in the CAM Processor in case none of the layers affected by a particular via is active (if the "Fill pads" option was unchecked, the hole was always drawn and may have punched through wires in other layers).
- Fixed handling the offset of wires and texts of frames contained in a device's symbol in UL_FRAME.
- Fixed handling the orientation and positioning of texts of frames contained in a device's symbol in the CAM Processor.
- Fixed handling extensions in the RENAME command.
- Fixed calculating the number of edges for arcs and circles in the CAM Processor's output devices.
- Fixed a possible data corruption if the RIPUP command with the '!'

option was used without terminating the command line with ';'.

Release notes for EAGLE 5.3.0

=====

* Platforms:

- The Linux version of EAGLE no longer uses libXinerama and libXfixes, because these libraries are not available on all systems.

* User interface:

- Normally EAGLE doesn't automatically position the mouse cursor. However, some users want the cursor to be repositioned to the point where it has been before a context menu in the drawing editor was opened. The command

`SET Option.RepositionMouseCursorAfterContextMenu 1;`

can now be used to get this functionality.

- Shift+Right mouse button now reverses the direction of rotation in commands that rotate objects with the right mouse button.

* AUTO command:

- Added the options LOAD and SAVE to the AUTO command, which allow loading and saving the Autorouter parameters.

* BOARD command:

- The default board outlines generated by the BOARD command now start at the origin of the drawing area, and are no longer offset so that they are centered within the 50mil grid.

* INVOKE command:

- The INVOKE command now displays in the status line the part and device name of the object attached to the cursor.

* PASTE command:

- The PASTE command now accepts an 'orientation' parameter in the command line.
- The PASTE command is no longer a transparent command, because this prevented its parameter toolbar from being shown.

* ROUTE command:

- When routing an airwire that starts at an already routed wire, the new wire's width is now automatically adjusted to that of the existing wire if the Shift key is pressed when selecting the airwire.

- The end point of the dynamically calculated airwire is now always used as an additional snap point, even if it is off grid. If the remaining airwire has a length that is shorter than SNAP_LENGTH, the routed wire automatically snaps to the airwire's end point, and stays there until the mouse pointer is moved at least SNAP_LENGTH away from that point. The SET parameter SNAP_BENDED is now obsolete, but is still tolerated for compatibility.

* SET command:

- When routing an airwire that starts at an already routed wire, the new wire's width is now automatically adjusted to that of the existing wire.
- The SET parameter SNAP_BENDED is now obsolete, but is still tolerated for compatibility.

* Miscellaneous:

- Improved selecting wires of the same signal that are close together.
- Pin names with '@' are now displayed in the previews the same way as in a schematic (with everything after and including the '@' stripped).

* Bugfixes:

- Now clearing the status bar after a Properties dialog.
- Fixed handling polygons in the t/bStop layer of mirrored elements in the DRC.
- Fixed getting the path to the AppData directory from the Windows registry in case it contains non-ASCII characters.
- Fixed a crash in the NET command when clicking on a bus that has a name that consists only of a "SYNONYM:" part.
- Fixed handling via lengths in the Properties dialog that are not allowed according to the layer setup.
- Fixed a crash in the CHANGE LAYER command with signal wires.
- Fixed displaying and printing objects in layer 50.
- Fixed renaming a net in the Properties dialog, in case the same net is present on other sheets, too.
- Fixed a possible loss of consistency when making changes in the schematic, switching to another sheet, making more changes, and then doing UNDO in the board, in which case the number of undo steps in the two editor windows drifted apart. Every time the sheet is changed in the schematic editor, a separate UNDO step is now stored, even if there is no actual change made on that sheet.
- Fixed unSMASHing parts in the schematic (this deleted attributes from the part in the board).
- Fixed extremely long delay when saving a file.
- Fixed the example regarding the use of SMD_FLAG_STOP in the online help page for UL_SMD.
- Fixed displaying the status bar header in case a command runs for a longer time and the mouse is moved inside the editor window.
- Fixed the User Language function UL_SIGNAL.airwireshidden.

- Fixed marking the text in the first string entry field in a User Language dialog when the dialog is opened (didn't work under Windows).
- Fixed handling the \$HOME environment variable in the directories set in the Options/Directories dialog under Windows.
- Fixed selecting pins with the GROUP command (they were included in the group even if the Symbols layer was not active).
- Fixed faulty nameless attributes for parts in case a part is copied and there have been attributes deleted before that.
- Fixed ensuring that the text cursor is visible when opening a text editor window while loading a project.
- Fixed displaying the placeholder texts ">DRAWING_NAME", ">LAST_DATE_TIME", ">PLOT_DATE_TIME" and ">SHEET" in a new, yet unsaved drawing.
- Fixed keeping the layer list focused when clicking on a button in the DISPLAY dialog.
- Fixed always displaying text in the HTML editor's upper pane as HTML.
- Fixed centering on the current error in the ERRORS dialog by pressing the 'Enter' or 'Return' key.
- Fixed setting the focus in the Design Rules dialog.
- Fixed refreshing the Control Panel's tree view after copying a directory.
- Fixed erasing the original airwire after displaying a dialog while the ROUTE command is active with an airwire attached to the mouse pointer.
- Fixed reporting calls to non-existing object members in the User Language (like, for instance, UL_WIRE.arc.curve).
- Fixed setting the 'Curve' parameter to 0 in the Properties dialog of an arc shaped wire.
- Fixed a possible crash when using CLOSE in a script file and a message box appears after that.
- Fixed automatically placing elements in the BOARD, ADD and PASTE command (issued from the schematic), so that they are not placed outside the valid range of coordinates.
- Fixed sorting newly created items in the Control Panel's tree view.
- Fixed drawing artefacts when doing UNDO after RIPUP.
- Fixed duplicate data for rotated rectangular apertures in Gerber RS-274-X.
- Fixed drawing artefacts when doing UNDO after deleting a polygon edge.
- Fixed dynamic airwire calculation in the ROUTE command in case layers containing objects that belong to the routed signal are not displayed.
- Fixed entering values into the Angle combo box of the parameter toolbar.
- Fixed setting HKEY_CLASSES_ROOT\Applications\eagle.exe\shell\open\command in the Windows registry ("command" was a string instead of a key).
- Fixed printing lines with zero width on the Mac (they were printed too thick).
- Fixed a possible hangup when pressing a character or digit key in the table of the attribute dialog of the library editor.

Release notes for EAGLE 5.2.0

=====

* ATTRIBUTE command:

- The "Technologies" combo box in the "New/Change Attribute" dialog now lists

the name of the current technology.

- Changed the behavior of the ATTRIBUTE command in case no coordinates are given in the command line, so that the value of an existing attribute can be changed via the command line.

* DRC command:

- The DRC now reports supply layers, that are not contained in the layer setup of the Design Rules, as "Layer Setup" errors.

* SET command:

- The SET command can now be used to set parameters of the eaglerc file.

* Miscellaneous:

- If you don't like the way unprocessed polygons display their edges (as dotted lines), you can change this with the command

SET Option.DrawUnprocessedPolygonEdgesContinuous 1

The edges of unprocessed polygons will then be displayed as continuous lines, as was the case before version 5 (however, they will not be highlighted).

- The commands CUT, DRC, ERC, ERRORS, EXPORT, GROUP, OPTIMIZE, PASTE, PRINT, RATSNEST, UPDATE, USE and WRITE are now "transparent", which means they no longer terminate a previously active command like MOVE etc.
- The layers are now drawn in a sequence that properly reflects the actual resulting board. For instance, the bottom placeplan is drawn first, then the signal layers from bottom to top, then the top placeplan. This sequence is reversed when printing mirrored.
- Added a note to the online help of the PAD and VIA commands, saying that the shape only applies to the outer layers (in inner layers the shape is always "round").
- Parts on the board are now shifted into the allowed area of the Light or Standard edition, respectively, if they extend outside that area after a REPLACE, CHANGE PACKAGE or UPDATE operation.

* Bugfixes:

- CHANGE SHAPE now silently ignores the options LONG and OFFSET when entered in the command line of a board editor window.
- Fixed displaying very narrow rectangles under Linux, in case the resulting rectangle on the screen is only a single pixel wide.
- Fixed visibility of sheet numbers in sheet thumbnails in case of black background.
- Fixed calculating the number of edges when drawing an arc with round endings in the CAM Processor.
- Fixed changing the wire bend style with the parameter toolbar buttons after switching into arc mode with Ctrl+Left.

- Fixed a User Language performance issue with string expressions like 's += t' in loops with many executions.
- Fixed a crash when changing the grid color while a library is open without having edited a particular package, symbol or device set, yet, and the grid is actually displayed.
- Fixed a performance issue in displaying status messages and progress bars.
- Fixed handling uppercase characters in directory names in the Control Panel's tree view under Windows and Mac OS X.
- Fixed setting the state of a dlgCheckBox from within its statement.
- Fixed a crash when entering a curve parameter (like "@+20") in a command that works with wires, without an actual wire attached to the cursor.
- Fixed a loss of the current text marking when clicking into a numerical entry field with the right mouse button.
- Fixed a possible data corruption after changing the coordinates of an object or the layer of a wire in the properties dialog, in case this resulted in a wire optimization.
- Fixed handling the 'A' modifier in coordinates entered in the command line.
- Fixed redisplaying a ULP dialog if the dlgRedisplay() call is followed by a loop statement.
- Fixed an unexpected change of the offset in a dlgTextView in case an other dialog item is modified.
- Fixed displaying status messages while a script is running.
- Fixed calculating the width of the last column in a dlgListView.
- Fixed updating the layer colors in the parameter toolbar's Layer combo box in case they are modified with the DISPLAY or SET command while a command is active.
- Fixed forward annotating a change of a net name in the properties dialog of a net wire or junction.
- Fixed drawing rotated rectangles with the CAM Processor's GERBER_RS274X driver in case the drawing consists exclusively of such rectangles.
- Fixed setting the paper size in the printer dialog on Linux systems that use the CUPS printing system.
- Fixed setting a modified attribute value back to the library default in case the default value is empty.
- Fixed overwriting a non-empty attribute default value with an empty value in the schematic.
- Fixed RIPUP of polygons of selected signals in case the command line is terminated with ';'.
- Fixed missing spaces when rendering '-' between <nobr> tags.
- Fixed displaying the placeholder texts ">DRAWING_NAME", ">LAST_DATE_TIME", ">PLOT_DATE_TIME" and ">SHEET" in case a part is smashed.
- Fixed handling file names that start with a '.' in the PRINT command.
- Fixed unexpected wire fragments in CAM Processor and PRINT output.
- Fixed a possible crash in the DRC when handling extremely small arcs.
- The values for "Catch factor" and "Select factor" can now be entered as decimal numbers in the "Options/Set/Misc" menu to allow values less than 1.
- Fixed the description of the palette() function in the online help to properly mention the alpha channel of ARGB values.
- Fixed the description of REAL_EPSILON in the online help.

- Fixed selecting a group with the alternate grid in the COPY command.
- Fixed subtracting very small and thin arcs from polygons.
- Fixed handling UTF-8 characters in the readfile() function when reading into a string array.
- Fixed the DRC to have it check unconnected pads/smds against objects in the Dimension layer, as it was the case in version 4.x.
- Fixed mirroring polygons in packages when dragging the packages in the ADD command.
- Fixed storing relative path names in the project file in case, e.g., a project named /abc/def is open and a file named /abc/def-1/some/name is edited.
- Fixed marking the text in the first string entry field in a User Language dialog when the dialog is opened.
- Fixed unwanted flashing of signal when placing a wire in the ROUTE command.
- Fixed a crash if a CHANGE command is applied to a group containing a polygon that is in the calculated state, and that CHANGE causes a recalculation of the polygon.
- Fixed calculating the bounding rectangle of placeholders for global attributes.
- Fixed handling escaped '&' characters in dlgLabel in case there is no actual hotkey in the label.

Release notes for EAGLE 5.1.0

=====

* Platforms:

- Now setting HKEY_CLASSES_ROOT\Applications\eagle.exe\shell\open\command in the Windows registry to allow associating file extensions with the EAGLE executable.
- The installer now uses the User Access Control dialog to request Administrator rights when running under a limited user account on Windows Vista.

* UPDATE command:

- Rephrased the online help for 'UPDATE old_library_name = new_library_name', to make it clear that the library inside the board/schematic file will be renamed, not the external library file.

* CAM Processor:

- The CAM Processor's GERBER_RS274X driver now generates polygon area fill codes when drawing rotated rectangles.

* Miscellaneous:

- Decimal numbers beginning with a comma are now accepted in the command line, provided the locale settings define the comma as decimal delimiter.
- If a text like >ABC is placed directly in a board or schematic, and there

is a global attribute with the same name and an empty value, the origin cross of that text is now displayed to allow the user to manipulate the text.

* Bugfixes:

- Fixed a crash when using the CAM Processor in command line mode (option -X) and having "Always vector font" turned off.
- Fixed drawing very short lines under Windows and Mac OS X, in case their coordinates in pixels result in a zero length.
- Fixed setting word wrapping in dlgLabel in case the label text contains HTML tags.
- Fixed rendering the '-' character between <nobr>...</nobr> tags.
- Fixed handling airwires in the ROUTE command after routing part of a signal (the remaining airwire reacted on CHANGE WIDTH).
- Fixed copying and CUT/PASTEing parts with attributes defined at part level.
- Fixed the User Language function printf() to correctly handle the value 0x00 and those above 0x7F in binary mode.
- Fixed defining a GROUP if the last coordinate of the group polygon is equal to the first one.
- Now ignoring faulty window coordinates in eagle.epf and eaglerc files.
- Fixed running User Language Programs that start with a Byte Order Mark.
- Fixed a crash in the ADD/REPLACE dialog when adding/replacing from a library, closing the editor window, opening the editor window again and adding/replacing again using the same library.
- Fixed positioning windows when opening a project under Linux.
- Fixed handling the XREF keyword in the LABEL command.
- Fixed storing color palette values in case they are defined in the command line or a script using the SET command.
- Fixed drawing circles with a solid fill style on Windows and Mac OS X in case the width is about as big as the radius.
- Fixed drawing lines with zero width when zooming far into the drawing.
- Fixed positioning a dlgMessageBox that contains HTML text.
- Fixed handling the '@radius' parameter in the WIRE command (and all others that accept such a parameter).
- Fixed updating the library name in the board when replacing a part in the schematic and only the library name has changed.
- Fixed handling minimum clearance values for circles in the Autorouter and when processing polygons.
- Fixed handling the "Use alpha blending" flag when displaying the color dialog for white or colored background.
- Fixed removing the error indicator from the board editor when closing or clearing the ERC error list.
- Fixed handling UTF-8 character constants in ULP files.
- Fixed displaying tool tips ("bubble help") in the parameter toolbars.
- Fixed processing keyboard shortcuts in the library editor in case no pulldown menu has been opened, yet.
- Fixed handling 'char' variables in the User Language as 8 bit values.
- Fixed displaying the root directory in the Control Panel's tree view, in case it is part of one of the directory paths.

- Fixed handling approved errors that have exactly the same signature.
- Fixed autorouting selected signals in case the net classes define particular clearance values between signals that have been selected for routing and those not selected.
- Fixed scrolling the drawing area with the mouse wheel if the "Mouse wheel zoom" factor is set to 0.
- Fixed the default wildcard in the User Language dialog functions `dlgFileOpen()` and `dlgFileSave()` under Windows. With "*" the Windows file dialog doesn't handle *.lnk files correctly, therefore the default wildcard is now "*.*)" as in previous versions.
- Fixed graphics distortions when running the Linux version of EAGLE on an X11 server that runs on a "big endian" system.
- Fixed drawing artefacts from the origin cross when moving a gate in a device set.
- Fixed a crash after highlighting a DRC error, switching to another board, and then using the DISPLAY command to change the visible layers.
- Fixed forward annotation in case a new attribute with an empty value is added to a part in the schematic, or the value of an existing attribute is changed to " (empty) in the schematic.
- Fixed false positive DRC errors for round objects in case the actual clearance is exactly the same as the minimum clearance.
- Fixed reference line for net labels when pointing to an arc.
- Fixed invalid coordinates in data generated by the CAM Processor for draw devices (like HPGL plotters) in case of very short lines that result in a zero length in machine units.
- Fixed handling part attributes that overwrite global attributes (the name field was disabled) and attributes with an empty value (the value field was disabled).
- Fixed handling bounding rectangles of smashed parts, in case the package or symbol contains nothing but placeholder texts.
- Fixed handling the last value in the Colors parameter of CAM job files.
- Fixed "File/Save all" so that it saves the project file even if "Automatically save project file" is not set.
- Fixed handling `UL_SMD.dx()/dy()` and `UL_PAD.diameter()` when called with a "stop" or "cream" layer in case the smd or pad has the NOSTOP or NOCREAM flag set.
- Fixed deleting local attributes that overwrite global attributes.
- Fixed changing the value of local attributes that overwrite global attributes in case the current value is empty.
- Fixed splitting wires when moving a wire endpoint onto an other wire.
- Fixed resolving *.lnk files under Windows in the Control Panel's tree view.
- Fixed displaying status texts in case an object's information contains one of the characters '<', '>' or '&'.
- Fixed using an attribute named VALUE as default value for devices that have 'Value On'.
- Fixed a performance issue in the User Language function `status()`.
- Fixed displaying the name of the loaded file in the status bar of the CAM Processor (it disappeared as soon as the mouse cursor was moved over the pull down menu).

- Fixed handling supply layers in the Autorouter in case a signal that has a supply layer is already (partially) routed and the Autorouter is run (the wires and vias of such a signal were not "seen" by the Autorouter and thus routed again; other signals routed in the same Autorouter run may even have overlapped or gotten too close to them).
- Fixed checking the font and width of texts of smashed elements in the DRC.
- Fixed an unexpected jump of the group when selecting the group with Ctrl+Right in a command after clicking into the drawing area with Ctrl+Right without an active command.
- Fixed the SPLIT command for polygon edges in case the split point is on the original wire.

Release notes for EAGLE 5.0.0

=====

* Platforms:

- The minimum system requirement on Windows platforms is now Windows 2000, XP or Vista.
- The Mac OS X version of EAGLE no longer requires an X11 server, and comes as a "Universal Binary" that runs on PPC and Intel Macs.
- The Linux version now comes as a self extracting shell script with a setup dialog.
- The buttons in dialogs are now placed in the sequence suggested by the interface guidelines for the particular platform.
- The Windows version of EAGLE now stores the 'eaglerc.usr' file under the directory that is defined by the registry key "HKEY_CURRENT_USER\Software\Microsoft\Windows\CurrentVersion\Explorer\Shell Folders\AppData" if no environment variable named HOME is defined. If no such file exists at the new location, it tries to read it from the old location.

* User interface:

- When clicking with the right mouse button on an object in an editor window, a context specific popup menu is now displayed from which commands that apply to this object can be selected.
- Since the context menu function on the right mouse button interferes with the selection of groups, a group is now selected with Ctrl plus right mouse button. If you want to have the old method of selecting groups back, you can add the line

```
Option.ToggleCtrlForGroupSelectionAndContextMenu = "1"
```

to the ~/.eaglerc file. This will allow selecting groups with the right mouse button only and require Ctrl plus right mouse button for context menus.

- The context specific object menu contains an entry named "Properties", which allows to display (and modify some of) the object's properties.
- The schematic editor now contains a thumbnail view of all sheets. Clicking on a thumbnail switches to that sheet. Drag&drop in the

thumbnail view can be used to reorder the sheets, and the context menu allows you to add and delete sheets.

- The attributes of parts in the board and schematic can now be selected by the commands that allow selecting objects with names by entering the concatenation of part name and attribute name, as in

MOVE R5>VALUE

- The context menu of package variants in the library editor now contains an option 'Edit Package' to quickly access the package of the selected variant.
- The context menu of a gate in the device editor now contains an option 'Edit Symbol' to quickly access the symbol of the selected gate.
- Renaming an item in the Control Panel is no longer done by clicking into the text of an already selected item (this has caused too many unintended activations of the editing mode). Use the context menu instead.
- The positions of all open Windows can now be stored using "Options/Window positions" in the Control Panel. Newly opened windows of the same type will then be positioned at the same places.
- Status messages and User Guidance are now displayed simultaneously in the status bar of the editor window.
- If the pulldown menu in an editor window is hidden, the Alt+X key no longer leaves the program. To have this functionality even with the pulldown menu hidden, use "ASSIGN Alt+X Quit;".
- When switching between the sheets of a schematic, the current zoom level is now saved.
- Panning the editor window with the center mouse button no longer requires to press the Shift key to exceed the area defined by the scrollbars.

* User Language:

- Entries in a `dlgListView` are now displayed in multiple lines if they contain '\n' characters.
- The User Language function `UL_POLYGON.contours()` now supports a second parameter that allows looping through 'positive' and 'negative' polygons separately.
- The User Language function `UL_CLASS.clearance` now accepts a parameter that allows retrieving the clearance between two net classes.
- The User Language objects `UL_ELEMENT` and `UL_INSTANCE` now have a new member function 'smashed'.
- Include statements in ULPs that contain relative paths, as in

```
#include "dir/file.ulp"
```

are now searched for within the directories entered under "Options/Directories/User Language Programs".

- The new User Language function `timems()` delivers the time in milliseconds since the start of the ULP.
- The new User Language function `ingroup()` can be used to check whether an object is within the current group.

- The new User Language function `system()` can be used to execute external programs.
- The User Language objects `UL_ELEMENT`, `UL_INSTANCE` and `UL_NET` have new data members 'column' and 'row' which return the location within a drawing frame.
- The User Language member functions `UL_ELEMENT.smashed` and `UL_INSTANCE.smashed` now accept an optional parameter text name that allows you to query whether there is a smashed parameter text by that name.
- In the User Language the labels of a segment (both bus as well as net) can now be accessed through the new object type `UL_LABEL`, which is generated by the new loop member `UL_SEGMENT.labels()`. The old way of accessing labels through `UL_SEGMENT.texts()` is now deprecated and won't handle cross-reference labels correctly. The actual text of a label is now returned by `UL_LABEL.text`.
- The User Language object `UL_SCHEMATIC` has a new member 'xreflabel', which returns the format string used to display cross-reference labels.
- The User Language object `UL_SCHEMATIC` has a new member 'xrefpart', which returns the format string used to display part cross-references.
- The User Language object `UL_INSTANCE` has a new loop member named 'xrefs', which loops through the gates that represent the contact cross-reference.

* Screen drawing:

- Drawing on screen no longer uses "raster OPs". The individual layers are now drawn using "alpha blending". Each color (except for the background color, which is always opaque) can have its own alpha value, which defines how transparent it is. A value of 0 means the color is fully transparent (i.e. invisible), while 255 means the color is completely opaque. When reading an `eaglerc` file from an older version, the alpha values of all colors are initialized to a default value if all palette entries have an alpha value of 0. You can use the script 'defaultcolors.scr' to set the colors to the new defaults. When printing, the alpha values are always set to 255.
- Since the layer colors no longer use additive mixing, but rather use alpha blending, the default background color in the layout editor window has been changed to white.
- If you want to have the old raster OP behavior on black background, you can uncheck the "Use alpha blending" box in the "Options/Set/Colors" dialog. In that case the alpha value defined for the colors is ignored when using a black background, and colors are mixed using an OR function.
- The colors above index 15 in the color palettes are now by default initialized with a medium gray value.

* User defined Attributes:

- In a library, devices can now have "attributes", which are arbitrary user definable "name/value" pairs. Attributes are related to the individual "technology" variants of a device.
- The new command `ATTRIBUTE` can be used to define the attributes of a given

technology variant (see "Help/Editor commands/ATTRIBUTE" for details).

- The new User Language object UL_ATTRIBUTE can be used to access attributes (see "Help/User Language/Object Types/UL_ATTRIBUTE" for details).
- The User Language objects UL_PART, UL_INSTANCE, UL_ELEMENT and UL_DEVICE have a new loop member named 'attributes()'
- The User Language objects UL_PART and UL_ELEMENT have a new member function named 'attribute()', which can be used to query a part for the value of a given attribute.
- The User Language object UL_ELEMENT has a new member function named 'attribute()', which can be used to query an element for the value of a given attribute.
- In a 'symbol' or 'package' drawing, any text that starts with a '>' character and matches an attribute name of the actual part or element will be replaced by the attribute value in the schematic or board, respectively (see "Help/Editor commands/TEXT" for details).
- The SMASH command now smashes all texts in the symbol or package that start with '>' and match an actual attribute name, and assigns them as attributes to the part (except for the traditional placeholder texts like ">NAME", ">VALUE" etc., which are treated like before).
- Boards and schematics can now have global attributes.
- The User Language objects UL_BOARD and UL_SCHEMATIC have a new loop member named 'attributes()', which can be used to loop through the global attributes.

* Locking the position of a part:

- The new command LOCK can be used to lock the position of a part in the board.
- The origin of a locked part is displayed as an 'x' to have a visual indication that the part is locked.
- The User Language object UL_ELEMENT has a new data member 'locked', which return the setting of the lock flag.

* Popup menus for buttons:

- Various buttons in the editor window now have a popup menu that contains a list of recently used items or user defined aliases (depending on the button type). These buttons are marked with a small black arrow at the bottom right corner of their icon. To access this list, click on the button and hold the mouse button pressed until the list pops up, or click on the button with the right mouse button.
- The button popup menus for DISPLAY, GRID and WINDOW contain two special entries: "Last" restores the previous settings, and "New..." queries the user for a new alias name and stores the current settings under that name.

* Aliases for command parameters:

- The DISPLAY, GRID and WINDOW commands now have an extended syntax that allows the user to define "aliases" for certain parameter settings. The syntax to handle these aliases is:

CMD = <name> <parameters>

Defines the alias with the given <name> to expand to the given <parameters>. The <name> may consist of any characters, except blank or semicolon, and is treated case insensitive.

CMD = <name> @

Defines the alias with the given <name> to expand to the current parameter settings of the command.

CMD = ?

Asks the user to enter a name for defining an alias for the current parameter settings of the command.

CMD = <name>

Opens the dialog of the command and allows the user to adjust the set of parameters that will be defined as an alias under the given <name>. In case of the WINDOW command a rectangle can be defined that represents the desired window area.

CMD = <name>;

Deletes the alias with the given <name>.

CMD <name>

Expands the alias with the given <name> and executes the command with the resulting set of parameters. The <name> may be abbreviated and there may be other parameters before and after the alias (even other aliases). Note that aliases have precedence over other parameter names of the command.

Example:

```
DISPLAY = MyLayers None Top Bottom Pads Vias Unrouted
```

Defines the alias "MyLayers" which, when used as in

```
DISPLAY myl
```

will display just the layers Top, Bottom, Pads, Vias and Unrouted.
Note the abbreviated use of the alias and the case insensitivity.

* Inverted names:

- The names of inverted ("active low") signals can now be displayed with

a bar over the text ("underline"). To do so, the name needs to be preceded with an exclamation mark (!), as in

!RESET

which would result in

RESET

This is not limited to signal names, but can be used in any text. It is also possible to overline only part of a text, as in

!RST!/NMI

R/!W

which would result in

RST/NMI

R/W

Note that the second exclamation mark indicates the end of the overline. There can be any number of overlines in a text. If a text shall contain an exclamation mark that doesn't generate an overline, it needs to be escaped by a backslash. In order to keep the need for escaping exclamation marks at a minimum, an exclamation mark doesn't start an overline if it is the last character of a text, or if it is immediately followed by a blank, another exclamation mark, a double or single quote, or by a right parenthesis, bracket or brace. Any non-escaped exclamation mark or comma that appears after an exclamation mark that started an overline will end the overline (the comma as an overline terminator is necessary for busses).

- When updating files from older versions, a backslash in any pin, net, bus or signal name will be replaced with the appropriate exclamation mark. Any backslash or exclamation mark in a normal text will be escaped by preceding it with a backslash as necessary, since the backslash is now a real escape symbol in texts.

* Drawing frame:

- The new command FRAME can be used to draw a frame with numbered columns and rows.
- The new User Language object UL_FRAME can be used to access the data of a drawing frame.
- The User Language objects UL_ELEMENT, UL_INSTANCE and UL_NET have new data members 'column' and 'row' which return the location within a drawing frame.
- The drawing frames in the "frames" library now use this new frame object.

* Cross-reference labels:

- A "label" on a net segment now has a new property named "xref", which puts it into "cross-reference" mode. In this mode it will be displayed according to the "Xref label format" string defined under "Options/Set/Misc", and will show its text at a different offset from its origin, so that it can be placed nicely at the end of a net wire.
- A cross-reference label that is placed on the end of a net wire will connect to the wire so that the wire is moved with the label, and vice versa.
- The format of cross-reference labels can be defined in the "Options/Set/Misc" dialog under "Xref label format". See "Help/Editor Commands/LABEL" for a list of placeholders that can be used here.
- The User Language object UL_SCHEMATIC has a new member 'xreflabel', which returns the format string used to display cross-reference labels.
- The SET command has the new parameter XREF_LABEL_FORMAT, which can be used to define the cross-reference label format string.
- The CHANGE command has a new option XREF that can take the values OFF and ON, and can be used to change whether a label is "plain" or "cross-reference".
- The LABEL command has the new option XREF to define a cross-reference label. There are also two new icons in the parameter toolbar to set this option.
- In the User Language the labels of a segment (both bus as well as net) can now be accessed through the new object type UL_LABEL, which is generated by the new loop member UL_SEGMENT.labels(). The old way of accessing labels through UL_SEGMENT.texts() is now deprecated and won't handle cross-reference labels correctly. The actual text of a label is now returned by UL_LABEL.text.

* Part cross-reference:

- The new text variable '>XREF' can be used in a symbol drawing to display a cross-reference to the MUST gate of the device this symbol is used in. A typical application for this are the contacts of a relay, where the '>XREF' text variable would display the frame coordinates of the relay's coil.
- The format of part cross-references can be defined in the "Options/Set/Misc" dialog under "Xref part format". See "Help/Editor Commands/TEXT" for a list of placeholders that can be used here.
- The SET command has the new parameter XREF_PART_FORMAT, which can be used to define the part cross-reference format string.
- The User Language object UL_SCHEMATIC has a new member 'xrefpart', which returns the format string used to display part cross-references.

* Contact cross-reference:

- EAGLE can now automatically generate a contact cross-reference, which is mainly used for relay coils and contacts in electrical schematics.
- The contact cross-reference is generated for the first MUST gate in a

part, and will display all other gates that have an '>XREF' text variable in their symbol drawing. The MUST gate is typically the coil of a relay, while the other gates are the contacts.

- The contact cross-reference is displayed at the same X coordinate as the MUST gate, and at the Y coordinate defined by a text variable with a value of '>CONTACT_XREF'. This text can be placed either in a frame symbol, or directly on the schematic sheet drawing. The first one encountered will be used. If no such text is found, no contact cross-reference will be generated.
- The User Language object UL_INSTANCE has a new loop member named 'xrefs', which loops through the gates that represent the contact cross-reference.

* ADD command:

- The syntax of the ADD command has been changed to allow using libraries with blanks in their file name. Note that now the device, package or symbol name always has to come first.

* ASSIGN command:

- On the Mac the ASSIGN command now knows the "Cmd" modifier key.

* BOARD command:

- The BOARD command now accepts a parameter that defines the raster in which to place the parts when generating the board, as in

```
BOARD 5mm
```

which would place the parts in a 5 millimeter raster (default is 50mil). The number must be given with a unit, and the maximum allowed value is 10mm.

* CHANGE command:

- The CHANGE command now selects only objects that actually possess the property that shall be changed.
- When selecting an object with the CHANGE command, that object is now flashed to indicate the change to the user.
- CHANGE LAYER now also works with a group.
- The new CHANGE option DISPLAY can be used to change the display mode of an attribute.
- The options in the CHANGE popup menu are now sorted alphabetically.
- CHANGE TEXT now accepts the new text on the command line and allows it to be applied to any number of text objects or the current group.
- The CHANGE command has a new option XREF that can take the values OFF and ON, and can be used to change whether a label is "plain" or "cross-reference".

* CLASS command:

- The minimum clearance between signals of different net classes can now be defined in a matrix, allowing you to define separate individual values for any combination of two net classes, as well as within the same net class (see "Help/Editor Commands/CLASS").

* COPY command:

- The COPY command can now copy a group by clicking with the right mouse button.

* DELETE command:

- The DELETE command can now select parts, pads, smds, pins and gates by name. The option SIGNALS to delete all signals in a board still exists, so if a part with the name SIGNALS shall be deleted, its name must be written in single quotes.

* DISPLAY command:

- The DISPLAY command no longer automatically turns related layers on or off when used with the t/bPlace or Symbols layer.

The parameter

```
Option.DisplayRelatedLayers = "0"
```

to the ~/.eaglerc file is now obsolete.

- The DISPLAY command now supports "aliases" for parameter settings (see "Aliases for command parameters").
- The DISPLAY command has a new option "Last", which restores the settings as they were before the previous DISPLAY command.

* DRC command:

- The DRC now reports wires in supply layers as errors if they are part of a signal that is connected to any pad or smd.
- The DRC now always checks all signal layers, no matter whether they are currently visible or not.
- The DRC now reports an error if an object in the t/bPlace, t/bNames or t/bValues layer overlaps with an object in the t/bStop layer (provided these layers are active when the DRC is run).
- The DRC no longer reports objects in the Top or Bottom layer that intersect with objects in the t/bRestrict layer in the same package.
- The DRC now distinguishes between clearance violations and actual overlaps of copper between different signals.
- The Design Rules dialog now marks the name of the Design Rules with an asterisk if they have been modified.

* EDIT command:

- The EDIT command can now insert and reorder schematic sheets.
- Switching between sheets in a schematic no longer clears the undo buffer. Adding, removing or reordering a sheet, however, still clears the undo buffer.

* ERC command:

- The results of the Electrical Rule Check (ERC) are now listed in a dialog, where clicking on a list item graphically marks the result in the editor window (if applicable).
- The parameter `Erc.SuppressAdditionalWarnings` in the `~/.eaglerc` file is obsolete. Errors and warnings are now presented separately in the ERRORS dialog.
- The ERC now checks for parts with user definable values that have no actual value.
- The ERC now warns about unconnected input pins of uninvoked gates.
- The ERC now warns if a net has more than one segment, and any of these doesn't indicate that it is part of a larger net (like, for instance, though a label, bus or supply pin).
- The ERC now checks whether the name of a net segment that is connected to a bus is actually contained in that bus.
- The ERC now warns if a pin is connected to a net, but there is no visible indication of the connection (like a net wire, junction or another pin).

* ERRORS command:

- If the ERRORS command is entered without having run an ERC or DRC before, the appropriate check is now started first automatically.
- The ERRORS dialog now allows the user to mark messages as "approved", which suppresses the error indicator in the editor window (see "Help/Editor Commands/ERRORS").

* EXPORT command:

- The default output format for EXPORT IMAGE is now PNG on all platforms (on Windows it used to be BMP).

* GRID command:

- The GRID command now supports "aliases" for parameter settings (see "Aliases for command parameters").
- The GRID dialog no longer has a "Last" button, because this functionality is now implemented through the command button popup menu. Click on the GRID button with the right mouse button to access the "Last" function.

* GROUP command:

- The GROUP command now has a new option ALL, which can be used to define a group that includes the entire drawing area.
- The GROUP command can now be used with the Shift and Ctrl key to extend

the group or toggle the group membership of individual objects, respectively.

* HELP command:

- Since Windows Vista doesn't support the Windows Help file format any more, EAGLE now uses the same HTML formatted help on all platforms.
- The Help window now has a "Find" bar where you can enter a text that will be used to filter all help pages, so that only those that contain the text will be shown.
- The help texts are now stored in one single HTML file for each language.

* INFO command:

- The INFO command can now select parts, pads, smds, pins and gates by name.
- The INFO command now brings up the same dialog as the Properties option in the context menu of drawing objects, and also allows changing properties.

* INVOKE command:

- If an already invoked gate is selected in the INVOKE dialog, the default button changes to "Show", and a click on it zooms the editor window in on the selected gate, switching to a different sheet if necessary.

* LABEL command:

- The LABEL command has the new option XREF to define a cross-reference label.
- The LABEL command now accepts an 'orientation' parameter to define the orientation of the label textually.

* MIRROR command:

- The MIRROR command now also works with rectangles.
- The MIRROR command can now select parts, pads, smds and pins by name.

* MOVE command:

- The MOVE command can now select parts, pads, smds, pins and gates by name.
- The MOVE command can now move a group of objects from one schematic sheet to an other, without modifying the board.

* NAME command:

- The NAME command can now rename an individual polygon, which moves the polygon from one signal to another.
- The NAME command can now select parts, pads, smds, pins and gates by name.

* PACKAGE command:

- The PACKAGE command, when used in the board or schematic editor, now behave exactly like CHANGE PACKAGE.

* PRINT command:

- The PRINT command has a new option named FILE, which can be used to print into a file.
- The PRINT command can now create PDF (Portable Document Format) files. These files are fully searchable for any (non-vector-font) texts they contain.
- The PRINT dialog now has a preview of the printed object.
- The scale factor in the PRINT command is now limited to the range 0.001...1000.
- The calibration values for printing are now limited to the range 0.1...2.
- The border values as delivered by the printer driver are now rounded up to the next higher multiple of 0.1mm.

* RATSNEST command:

- The RATSNEST command now ignores wires in supply layers.
- The RATSNEST command can now be called with signal names to calculate only the airwires and polygons of selected signals.
- The RATSNEST command can now hide the airwires of selected signals.
- The RATSNEST command now displays the name of the currently processed signal in the status line.
- The RATSNEST command now generates airwires for objects inside hatched polygons that would "fall through" the hatch lines. Thermal and annulus rings inside a hatched polygon that do not have solid contact to any of the polygon lines are no longer generated.

* REPLACE command:

- The REPLACE command now works in the schematic, too.

* RIPUP command:

- The RIPUP command now has a new option '@' to allow ripping up all or selected polygons.
- The RIPUP command can now handle wildcards in signal names.

* ROTATE command:

- The ROTATE command can now select parts, pads, smds and pins by name.

* ROUTE command:

- The "Via-Layers" combo box has been removed from the parameter toolbar of the ROUTE command, since the ROUTE command always automatically determines the minimum necessary via to make a connection.

- The ROUTE command can now select airwires by signal name.
- The ROUTE command no longer allows routing in supply layers.
- The ROUTE command with the Ctrl key pressed can now also start routing at a via.

* SET command:

- The new SET variable CATCH_FACTOR defines the distance from the cursor up to which objects are taken into account when clicking with the mouse (see "Help/Editor Commands/SET").
- The SET variable GRID_REDRAW is now obsolete, but is still tolerated for compatibility.
- The SET command can now configure the popup menus for values of Isolate, Spacing and Miter by setting the Isolate_Menu, Spacing_Menu and Miter_Menu arrays.
- The SET command has the new parameter XREF_LABEL_FORMAT, which can be used to define the cross-reference label format string.
- The SET command has the new parameter XREF_PART_FORMAT, which can be used to define the part cross-reference format string.

* SHOW command:

- The SHOW command now works with wildcards.
- The SHOW command now highlights the individual nets belonging to a bus if a bus is selected.
- The SHOW command now accepts a list of arguments and highlights all the matching objects.
- The SHOW command with the name of an individual instance (like IC1A, which is the gate A of part IC1) now shows exactly that instance.
- The SHOW command now uses the Ctrl key to toggle the highlight status of the selected object, which also allows more than one object to be highlighted at the same time.

* SMASH command:

- The SMASH command can now select parts by name.

* SPLIT command:

- The SPLIT command now automatically picks up the next wire segment when placing a splitted wire. That way an already routed wire can be re-routed more easily.
- The SPLIT command now also works with the "freehand" wire bend style.

* TECHNOLOGY command:

- The TECHNOLOGY command, when used in the board or schematic editor, now behaves exactly like CHANGE TECHNOLOGY.

* WINDOW command:

- The WINDOW command now supports "aliases" for parameter settings (see "Aliases for command parameters").
- The WINDOW command has a new option "Last", which restores the settings as they were before the previous WINDOW command.

* WIRE command:

- Starting a WIRE with the Ctrl key pressed now snaps the starting point of the new wire to the coordinates of the closest existing wire. This is especially useful if the existing wire is off grid. It also adjusts the current width, layer and style to those of the existing wire. If the current bend style is 7 ("Freehand"), the new wire will form a smooth continuation of the existing wire.

* CAM Processor:

- The output file name in the CAM Processor can now be defined using several "placeholders" (see "Help/Generating Output/CAM Processor/Output File"). The old variant of using ".ext" or ".*#" still works, but is obsolete.
- The photoplotter and drill station info files now start with the fixed string "Generated by EAGLE CAM Processor", followed by the EAGLE version number.

* Autorouter:

- The autorouter now displays in the status line the name of the currently processed signal and the time (in seconds) it has spent on a particular connection in case it takes longer than 5 seconds.

* Text editor:

- The "Find&Replace" dialog in the text editor now has a "Prompt before Replace" option.
- The keyboard shortcuts in the text editor now follow the platform specific standards.

* Polygons:

- When processing signal polygons, round objects are subtracted in such a way that the resulting error does not exceed 0.05mm (50 micron), which means that the distance between an object and a generated polygon edge may be up to 0.05mm larger than the value defined for the clearance or isolation, respectively. This is done to keep the number of polygon edges reasonably low.
- Signal polygons in "outline" mode are now displayed with dotted wires, so that they can be distinguished from other wires.
- Fixed calculating polygons in signals that also contain other wires, vias, pads or smds. If none of these other objects is on the same layer as the

polygon, the polygon was actually calculated instead of being shown as outlines.

IMPORTANT NOTE: This fix may cause polygons that have previously been calculated to be not calculated any more, and thus be missing from the printout or CAM data! These polygons were electrically "floating", because they were not connected to any other part of the same signal.

* Miscellaneous:

- The endings of wires and arcs with cap=round, as well as 'long' and 'offset' pads, are now displayed round on all devices (no more octagonal approximation).
- GROUP/MOVE now preserves the connectivity between wires, airwires and vias, even if one of the related layers is not currently displayed.
- Increased the maximum nesting level in config files to avoid problems with autorouter control files that define a large number of optimizing passes.
- The semicolon (;) is now no longer accepted in object names, to avoid problems with parsing command lines.
- The Printer.InternalRendering parameter in the ~/.eaglerc file has no meaning any more. Printing under Windows should now always work.
- Trailing whitespace following the continuation character '\' in script files is now ignored.
- Blanks in layer names are no longer accepted.
- Selecting objects with only a single selection point (like pads, vias etc.) has been improved.
- The different language versions are now all installed at once. Language specific files are distinguished by adding the two letter language code to their name, as in README_en. The DESCRIPTION files in directories hold the language specific texts between HTML tags <language en>...</language>. CAM Processor and Design Rules files separate different language versions of their parameters by adding, e.g., [en] to the parameter names, as in Name[en]="Component side".
- The positions of the window splitters in the library editor are now stored separately for each drawing type.
- If a layer name or a parameter alias entered by the user fully matches, it is now preferred over a partially matching one. For instance, if there are two layers named "Abcdef" and "Abc" (in this sequence) and the user enters "Ab", then the first matching layer "Abcdef" will be selected. If the user enters "Abc", the second, fully matching layer is selected.
- Coordinates and sizes (like width, diameter etc.) can now be given with units, as in 50mil or 0.8mm. If no unit is given, the current grid unit is used.
- The info string for parts (as displayed by various command, like SHOW and MOVE) now also lists the part's value in the status line.
- Entries in the "Open recent" lists are now only added when a file is actually loaded from or saved to disk.
- When moving a net or bus label, a line is now drawn to the closest point of the segment this label belongs to.
- If a command displays a progress bar, the window title now displays the current percentage.

- Changed the term "Rich Text" to HTML, to avoid confusing it with a text format from Microsoft.
- In order to allow easier selection of circles that have a large width (as compared to their radius), circles can now be selected not only at their radius, but also at their inner and outer circumference.
- The Professional Edition can now handle up to 999 schematic sheets.
- EAGLE can now automatically check if there is a new version available on the CadSoft server. You can explicitly run this check through "Help/Check for Update" from the Control Panel.
- The Projects path under Windows now also contains "\$HOME\eagle" to offer the user a default location for saving own projects.
- The new command line option -C can be used to start EAGLE with a command string that will be executed in the editor window (see "Help/Command Line Options").
- Centering on errors now also works if the error is close to the edge of the drawing.
- Decimal numbers can now be entered with a comma as the decimal separator (if the locale settings allow this). It is strongly recommended, though, to use the 'dot' as the decimal separator when writing scripts or ULPs that return EAGLE commands through the exit() function, otherwise they might not work on other systems.
- The device editor now displays origin crosses at the individual gates' origins.
- The new schematic layers "Info" (97) and "Guide" (98) can be used for general information and guide lines, respectively. The latter is mainly for electrical schematics, to help properly align relay coils etc.

* Bugfixes:

- Fixed a possible loss of connectivity if a wire disappears (because both ends coincide) after moving a group, and that same group is moved again.
- Fixed a possible crash when moving a group that contains supply pins that get connected to a net with a different name which is also contained in the group.
- Fixed a possible unjustified "close but unconnected wires" error in the ERC.
- Fixed checking for '\$' at the beginning of the layer name in the "Change layer properties" dialog.
- Fixed handling empty texts in DRC and polygon subtraction.
- The LAYER command can no longer switch to a layer that is not contained in the layer menu.
- Fixed removing the currently edited device set from within a script.
- Fixed handling layer names case insensitively in the DISPLAY dialog.
- Fixed a problem with temporary files in case the same user runs more than one instance of EAGLE on the same computer.
- Fixed handling HTML tags in error messages in the status line of the text editor.
- Fixed assigning to UL_* objects for "virtual" objects like the wires of a calculated polygon of a text with vector font.
- Fixed updating files from versions before 4.0 (sometimes duplicate

library names were not detected properly in old files).

- The #require directive in the User Language didn't handle release and revision numbers above and including 50 correctly.
- Fixed handling coordinate parameters in the DRC command.
- Fixed handling circles with $\text{width}/2 > \text{radius}$ in the Autorouter.
- Preventing the autorouter from setting vias at places where this is not possible when routing boards with a single signal layer and a supply layer.
- Fixed the library update in case the library no longer contains package variants that are present, but unused in the schematic.