

Tak sme sa dočkali novej verzie tohoto návrhového systému.
Sťahovať môžete z [tejto adresy](#).

Popis vylepšení a opráv, [viď nižšie....](#)
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

1. `UL_SMD smds[];`
2. `...`
3. `if (smds[i].x == smds[j].x)`
4. `...`

- 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

1.
``

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

1.
``

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 an other 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 an other 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).