

EAGLE Version 4.1 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 4.16r2!

WARNING: Due to some necessary changes in the data structure once you edit
a file with version 4.1 you will no longer be able to edit it
with versions prior to 4.1!

PLEASE MAKE BACKUP COPIES OF YOUR CURRENT BOARD-, SCHEMATIC- AND
LIBRARY-FILES BEFORE EDITING THEM WITH VERSION 4.1!

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 4.16r2

=====

* Bugfixes:

- Fixed generating pixmaps that are much longer in one dimension with the EXPORT IMAGE command.
- Fixed handling the net class when renaming a net segment to a new name.
- Fixed a crash when clicking into the empty area of the Technologies dialog.
- Fixed an 'Internal polygon error 73' when zooming far in on a board that contains smds with a roundness other than 0.

Release notes for EAGLE 4.16r1

=====

* Bugfixes:

- Fixed handling '\' in UNC program file name under Windows XP in case the program is installed on a server.
- Fixed handling the ESCape key in ULPs with nested dialogs.
- Fixed handling the program file name under Windows if it starts with ".\".
- Fixed handling German umlauts in the User Language functionsstrupr(), strlwr(), toupper() and tolower() under Windows and Mac OS X.
- Fixed setting the current item in the Control Panel's tree view when drag&dropping a package into a library in case there is also a device set with the same name.
- Fixed unexpected airwires in case an open Pwr pin is connected to a net with the same name, or a Supply pin with the same name is placed on an open Pwr pin.
- Fixed printing rotated or mirrored texts under Linux and Mac OS X.
- Fixed handling upper/lowercase when licensing under Windows on a server.
- Fixed handling the highest byte of the Palette values in the eaglerc file.
- Fixed handling polygons with only two edges, in case one of them is

an arc.

Release notes for EAGLE 4.16

=====

* Bugfixes:

- Fixed CUT/PASTE of net classes (only the first two were actually pasted).
- Fixed handling library name after "Save as" (was wrong in Description editor).
- Fixed CHANGE PACKAGE/TECHNOLOGY in case a device contains more than 254 technologies (which was possible due to a missing check in the technology dialog of the device editor).
- Fixed the technology dialog in the device editor, so that it doesn't accept more than 254 technologies per package variant.
- Fixed a possible data corruption when a supply pin overwrites a net name. Leftover pin references that may have been caused by such a data corruption are automatically deleted during the next library update.
- Fixed an unexpected "Cancel" button in some message boxes.
- Fixed generating annulus symbols for pads that have the NOTHERMALS flag set.
- Fixed faulty splitting of arcs near their end points.
- Fixed a rounding error in handling rectangle coordinates and wire curves.
- Fixed moving mirrored packages with polygons in a board (polygons were displayed in the wrong layer).
- Fixed faulty "Change Class..." lines in the EXPORT NETLIST output from a schematic.
- Fixed a problem with getting the program directory name under Windows XP if the console version of EAGLE was started without a full path name.
- Fixed a possible crash when canceling the console version of EAGLE with Ctrl+C under Windows XP.
- Fixed loading a text file on Windows XP from a non-Windows server (the file was not editable even though it was writable in the file system).
- Fixed storing Undo data when doing a library update where the sequence of gates had changed in a device. If doing UNDO followed by REDO after such an update, some of the part's gates may have been swapped.

Release notes for EAGLE 4.15

=====

* Bugfixes:

- The CHANGE PACKAGE command now updates the package in the board with the version from the schematic, in order to avoid problems in case a REPLACE has been done in the board while the schematic was closed.
- Fixed handling access to the individual characters of a string in ULPs on Mac OS X.
- The COPY command now updates the package in the board with the version from the schematic before actually adding the copied part, in order to avoid problems in case a REPLACE has been done in the board while the schematic was closed.
- The library update now reports a modification to the board even if it was just the renaming of some packages due to a previous REPLACE with the schematic closed.
- Fixed handling empty strings in dlgListView.
- Fixed clearing the selection of a dlgListView.

- Fixed setting user defined default Design Rules when loading an existing library from within a project.
- Fixed an extra line that appeared when closing a group with the right mouse button in case the group was empty.
- Fixed a possible crash in UL_WIRE.pieces().
- Fixed cursor positioning after an error message regarding a loop member in a ULP.

Release notes for EAGLE 4.14

=====

* CAM Processor:

- Increased resolution of EXCELLON driver to 1/10000 inch.
- Automatically generated drill codes are now sorted by drill size.

* User Language:

- The User Language object UL_PIN has a new data member 'net' which returns the name of the net the pin is connected to.

* Bugfixes:

- Fixed an unjustified ripup of wire segments when connecting a pin or net segment to a supply net that consists of more than one segment.
- Fixed the progress display in case a script is called from a ULP through its exit() function.
- Fixed setting a via in the ROUTE command if the placed wire coincides with an existing wire after a layer change.
- Fixed a possible crash when copying a device set from one library to another.
- Fixed handling '>SHEET' in ULPs (the fix in version 4.13r1 was broken).
- Fixed defining a group that contains vias in case the color of the Vias layer has been set to 0.
- Fixed refreshing all signal layers if the Pads or Vias layer is activated and has the color 0.
- Fixed UL_CONTACT.signal in case it is used in a UL_PART context.
- Fixed drawing arcs with a width of one pixel when zooming in.
- Fixed handling unsorted dlgListView objects in User Language Programs (the items were in reverse order).
- Fixed an unjustified "missing return value" error message in case a User Language Function is terminated through the exit() function.
- Fixed handling arrays of UL_* objects in User Language Programs.
- Fixed handling eagle.key in case the program was started using an UNC path name under Windows (you may be asked for your license.key file when you first run this version under Windows; users of the Freeware edition should delete the file 'eagle.key' from their EAGLE binary directory before starting EAGLE, to have it automatically offer the Freeware license again).
- Fixed the ASSIGN command (if the very first ASSIGN was given in the command line, all default key assignments were deleted).
- Fixed handling assigned keys that are mapped to completely written commands that appear in one of the pulldown menus.
- Fixed the CLASS command in case the name of the net class is given before the number.
- Fixed displaying mirrored texts in schematics after pasting from a board.
- Fixed handling 'char' variables in ULPs on Mac OS X.
- Fixed a spurious '%n' in the *.cam files that was introduced in version

4.13r1.

Release notes for EAGLE 4.13r1

=====

* DRC command:

- The "Stop after ... errors" parameter has been removed from the "Misc" tab of the Design Rules dialog. This parameter has sometimes lead to boards not being tested completely, without people noticing the "DRC: Stopped after xx errors" status message, which indicated that the DRC was not completely finished.
The 'maxErrors' parameter in *.dru files is still accepted, but has no more meaning. A board file that is created with this version of EAGLE will have the 'maxErrors' parameter set to 999999, so that an older version that loads this file will perform a full DRC.

* Miscellaneous:

- Now the default key macros are assigned only if no key assignments exist.
- If a library causes an error message when loading it in the ADD command, it will now automatically be removed from the list of used libraries.
- Improved the click&drag detection in case of an object selection in a densely populated area.
- The libstdc++ library is now linked statically under Linux to avoid problems with incompatible versions.

* Bugfixes:

- Fixed deleting a blocked wire if Cmd.Delete.WireJointsWithoutCtrl is set.
- Fixed handling a drill rack file name in an existing Excellon CAM job when switching to a different device.
- Fixed a possible "internal polygon error 73" when drawing a very wide or very high rectangle at a large zoom level.
- Fixed drawing rectangles and circles (they were drawn 1 editor unit too small).
- Fixed handling CUT/PASTE of partially selected polygons in symbols and packages.
- Fixed handling variable names in ULPs that are the same as an object member name within a member function that requires an index.
- Fixed reporting missing apertures in the GERBER device for rotated, non round pads and smds in case of aperture emulation being turned off.
- Fixed calculating the surrounding rectangle after editing a symbol (bug was introduced in 4.12r06).
- Fixed resetting the signal name after drawing a polygon.
- Fixed a possible crash in the ROUTE command when placing a wire and an error message about not being able to set a via pops up, and while that message is on screen, the editor window had to be refreshed (either because the error message has been moved or some other window has obstructed the editor window).
- Fixed handling '>SHEET' in ULPs.
- Fixed filling large rotated rectangles in the CAM Processor for devices like GERBER.
- Fixed an unjustified DRC width error for very short arcs with round endings.
- Fixed handling directory delimiters for \$HOME and \$EAGLEDIR under Windows to make sure directory names are always separated by forward slashes (/) inside ULPs in order to make things platform independent.

- Fixed exporting PAD and SMD data in library scripts (flags were missing).

Release notes for EAGLE 4.13

=====

* Platforms:

- EAGLE now also runs on Mac OS X (with X11). See the README file for details about system requirements.

* Supply Layers:

- Added a note regarding the size of annulus symbols in supply layers to the online help for the LAYER command.

* User Language:

- Fixed the example in the online help of 'dlgLabel'.

* BOARD command:

- The BOARD command now places elements in the third and, if necessary, fourth quadrant of the newly created board in case there are too many of them to fit into the second quadrant.

* DRC command:

- The DRC no longer checks polygons in packages against objects that have no electrical potential for clearance errors.

* VIA command:

- When placing a via at a point where an SMD exists that is connected to a signal, the via is now automatically added to that signal.

* Miscellaneous:

- With active f/b annotation any operations that would combine two signals in the board and would result in combining two nets in the schematic are now forbidden and need to be done in the schematic.
- Changed the default entries in the drill diameter menus to metric values.
- It is now possible to zoom into a drawing up to a factor where the internal editor resolution (0.1 micron) is visible. The checkbox "Options/User interface/Limit zoom factor" is still present and checked by default, but there is no more warning message when it is turned off. See "Help/Editor commands/WINDOW" for details on a few quirks that may happen when zooming in very far.
- When running as user 'root' under Linux and Mac OS X (which is only necessary to do the licensing and should be avoided under normal operation), EAGLE no longer sets the "Projects" path to avoid a popup message at program start, and it also doesn't save the ~/.eaglerc file any more.
- Improved opening the Control Panel's "File/Open/Project" menu in case there are many subdirectories that do not contain EAGLE projects.
- Coordinates are no longer snapped to the current grid if they are entered textually with the '>' modifier, as in (> 1 2).

* Bugfixes:

- Fixed handling very short wires in the ERC's test about net wires that only appear to be connected.
- Fixed an occasional unjustified "Can't close window while command is running!" error.
- Fixed auto routing towards an arc (it could happen that the last generated wire segment ended somewhere completely different).
- Fixed loading full screen windows that are not maximized from a project under Windows.
- Fixed drifting windows when loading from a project under Linux.
- Fixed wrong window coordinates when restoring maximized windows from a project under Windows.
- Fixed a possible crash when doing 'WINDOW (@);' after a PRINT on Windows (only happened with some printer drivers).
- Fixed the DRC LOAD command in case the file name was not absolute and the file was not located in the current directory.
- Fixed handling file names in the "Open recent" lists on systems that handle file names case insensitive (Windows).
- Fixed a problem in assigning function keys, which has been introduced in version 4.11r04.
- Fixed displaying pads and vias when dragging them on a white or colored background.
- Fixed opening files in ULPs with the output() statement if the file name contains uppercase characters.
- Fixed deleting a part in a schematic with Shift+DELETE in case it has gates on other sheets (these other gates were also drawn on the current sheet when doing UNDO).
- Fixed calculating polygons (the CAM Processor generated slightly different results in consecutive calls with the same board file; these differences were at most the size of the machine resolution, which in practice had no negative effects).
- Since the RATSNEST and the DRC command may produce faulty results if a drawing contains invalid polygons in the layers Top...Bottom, t/bRestrict, t/bKeepout or Dimension, such polygons are now reported as errors (a polygon is invalid if two or more of its edges intersect).
- Fixed keeping the selected package variant visible in the device editor's package variants list.
- Fixed handling the recent files list for boards that have been newly created from a schematic.
- Fixed layer selection in CAM jobs (changes were lost when loading a drawing, or creating a new section, without first saving the changed CAM job).

Release notes for EAGLE 4.12

=====

* Autorouter:

- The Autorouter now optionally runs even if a signal layer that contains objects is not activated.
- When routing from a pad that covers more than one raster point, the Autorouter now tries to lay out the wire at the middle of the pad.
- The Autorouter now applies the cfSmdImpact parameter to SMDs that are placed at angles of 45, 135, 225 or 315 degrees.

* CAM Processor:

- The CAM Processor now also has a "File/Open recent" menu.
- The CAM Processor now always prints all vias (even blind or buried) if the Vias layer is active, but none of the layers 1..16 is active.
- Added a note to the CAM Processor's "Photoplotter Info File" regarding missing apertures in case they were requested in a non-orthogonal angle.
- The CAM Processor no longer deselects all layers when clicking into some empty space of the "Layers" list.
- Implemented some overlap when filling polygons with Postscript in the CAM Processor to avoid small gaps on some output devices.
- The EXCELLON device in the CAM Processor now writes the actual drill sizes into the output file, so you no longer need to send the "drill rack" file to the board manufacturer. If you don't like this, you can disable this feature by deleting or commenting out the line

```
DrillSize = "%sC%0.4f\n"      ; (Tool code, tool size)
```

in the eagle.def file.

- The EXCELLON device in the CAM Processor now automatically generates the drill size definitions according to the actual values used in the board, so you no longer need a drill rack file. If you don't like this feature, you can disable it by deleting or commenting out the line

```
AutoDrill = "T%02d"          ; (Tool number)
```

in the eagle.def file. Existing CAM jobs that have a drill rack file defined will continue to use that file.

- The device EXCELLON_RACK has been introduced to still have the old functionality with user supplied rack file available.
- The new drill station parameter BeginData can be used to define a string that is output before the actual drill data (the EXCELLON device now outputs a '%' here).
- Added M48 and M72 to the EXCELLON Init string.

* Control Panel:

- The Control Panel now has a new menu item "File/Open recent projects".

* Design Rules:

- Increased the maximum copper thickness in the layer setup of the Design Rules to 1mm.

* User Language:

- The new User Language function language() can be used to internationalize ULPs (see "Help/User Language/Builtins/Builtin Functions/Miscellaneous Functions/language()").
- The User Language directive #usage can now handle internationalized texts (see "Help/User Language/Syntax/Directives/#usage").
- The new User Language directive #require can be used to tell the user that a ULP requires at least the given version of EAGLE (see "Help/User Language/Syntax/Directives/#require").

* ADD command:

- Changed the meaning of the wildcards ('*' and '?') in the ADD command. They used to match [a-z0-9_] and will now match any non-whitespace

character.

- The ADD dialog now has a checkbox that allows the pattern search in the descriptions to be turned off.

* DELETE command:

- The DELETE command now combines two wires if applied to their joining point with the Ctrl key pressed. If you want to have this functionality without pressing the Ctrl key, you can append the line

```
Cmd.Delete.WireJointsWithoutCtrl = "1"
```

to the eaglerc.usr file.

* DRC command:

- The new options LOAD and SAVE in the DRC command can be used to load the Design Rules from or save them to a given file.
- The DRC no longer checks objects that have no electrical potential (like wires in packages, rectangles, circles and texts) against each other for clearance errors.
- The DRC dialog now has a 'Check' button instead of 'OK'.

* EXPORT command:

- The EXPORT IMAGE command can now create TIFF files.

* INFO command:

- The INFO command now also displays the data of the part when selecting a text from a smashed part.

* MITER command:

- Changed the default values for "miter radius" to some typical grid values.

* MOVE command:

- Moving texts of a smashed part now draws a line to the part's origin so that the user can see which part this text belongs to.

* NET command:

- When drawing a net wire that connects two segments of different nets, the question "Connect Nets?" was changed to "Connect Net Segments?" in order to make it clear that only the two segments involved are concerned, and not the entire nets.

* PRINT command:

- The selected printer, paper size and orientation are now also saved and restored under Windows.

* RATSNEST command:

- Unnecessary thermal stubs that could occur around pads, vias and smds when calculating signal polygons are now avoided. Note that due to this

modification there may be cases where a pad, via or smd that used to be considered connected to the polygon is now no longer actually connected and the RATSNEST command will generate an airwire.

WARNING: If you send a board file created with this version of EAGLE to a boardhouse for manufacturing, and they will produce the CAM data themselves, please make sure that they use EAGLE version 4.11r05 or higher. Otherwise the manufactured board may contain thermal stubs even though you didn't see them in your version of EAGLE.

* RENAME command:

- If the RENAME command is entered without any additional parameters in a package, symbol or device drawing, a dialog now pops up that requests the input of the new name for this object.

* UPDATE command:

- The UPDATE command's new syntax 'old_library_name = new_library_name' can be used to update a library in a board or schematic with the contents of an other library (see "Help Update").

* VIA command:

- The VIA command now activates the layers that correspond to the length of the via in case none of these layers is active and the Vias layer is set to color 0.

* WINDOW command:

- 'WINDOW (@)' no longer reacts if the cursor is outside the editor window.

* Miscellaneous:

- When renaming a signal the new name is now the default when the user is prompted whether to combine two signals.
- The list of layers in the CAM Processor is now always wide enough to display the full layer names.
- The CAM Processor now only prompts once per job (not once per job section) whether the current file should be reloaded.
- When clicking into the drawing area of an editor window in order to activate that window, an active command in that window now ignores that mouse click to avoid unintended effects (like, for instance, inadvertently deleting an object).
- Speeded up ratsnest calculation for large signals.
- The "File/Open recent" list is now also updated in case of a "File/Save as" operation.
- Changes to the visibility of toolbars made through the toolbar context menu are now also stored in the eaglerc file.
- When mirroring a part where wires connected to that part change their layer, vias at the far ends of these wires are now placed/removed as necessary.
- The program now uses German menu texts if the system is set to a German environment.
- The window title now displays the program version number.
- If an error is detected while calculating polygons, the editor window now zooms to one of the offending polygon edges.

- Fixed the width of some characters in the vector font.
WARNING: Note that due to this change some texts may turn out longer than they used to! If you have vector font texts on any of your signal layers, make sure you do a DRC before manufacturing the board with this new version!
- If the center mouse button is used to pan the editor window, any special function of that mouse button (like bringing up the layer dialog) will now be performed if the panning distance doesn't exceed 10 pixel.
- When panning the editor window with the center mouse button, the movement can now exceed the limits defined by the scroll bars if the Shift key is held down while panning.
- If EAGLE is called with an eagle.epf file as argument, the respective project will now be opened.
- If EAGLE is called with the name of a project that is contained in one of the directories listed in "Options/Directories/Projects", that project will now be opened.
- Improved selecting objects in densely populated areas with click&drag.
- The splitter position in the Control Panel and the Help window (Linux only) is now stored in the eaglerc file.
- If you don't like the special mode in wire drawing commands that allows for the definition of an arc radius by pressing the Ctrl key when placing the wire, you can add the line

`Cmd.Wire.IgnoreCtrlForRadiusMode = "1"`

to the eaglerc.usr file. This will turn this feature off for all commands that draw wires.

* Bugfixes:

- After dropping a library from the ADD dialog, the respective entry in the Control Panel's tree view is now refreshed accordingly.
- Fixed handling the coordinates of rectangles (were sometimes off by one editor unit).
- Fixed selecting a rotated rectangle with Ctrl+MOVE.
- Fixed Gerber data output in case of negative coordinates ('-' was counted against the number of digits).
- Fixed handling UL_GRID.unitdist.
- Fixed a crash under Windows when re-loading a very large text file.
- Fixed '#' substitution in output file names of the CAM Processor.
- Fixed a possible crash when browsing through libraries from older versions in the ADD dialog.
- Fixed an unjustified DRC width error for short arcs with round endings.
- Fixed a crash when doing ROTATE with a part name and then clicking into the drawing with the right mouse button in case there was no group defined.
- Fixed highlighting parts and elements when clicking on their smashed names or values with SHOW.
- Fixed drawing rectangles on HPGL2 and derived devices and DESIGNJET650 in the CAM Processor. Since version 4.11 the "Bar" function in 'eagle.def' requires handling of the angle, which these devices can't do, so they need to emulate rectangles.
- Fixed loading drawings that were created with EAGLE versions before 3.0 directly into version 4.11 (object orientations were not updated correctly).
- Fixed handling rectangles in schematics in dxf.ulp.
- Fixed an unjustified renaming of a net when deleting a junction.
- Fixed handling >PLOT_DATE_TIME in EXPORT IMAGE.
- Fixed handling the character 'mue' (0xB5) when converting strings to

- uppercase. Note, though, that still only the characters below ASCII code 127 are guaranteed to work. Any other characters should not be used!
- Fixed an unjustified change of package or symbol names to "...@1" when doing a library update in case the device set as used in the schematic is no longer present in the library.
 - Fixed mirroring polygons that are completely contained in a group.
 - Fixed handling coordinates in the ERC report.
 - Fixed handling text files containing locale dependent characters like, for instance, the 'euro' symbol.
 - Fixed unjustified airwires in case non-through vias and supply layers are used.
 - Fixed ADDing parts with drag&drop from the Control Panel that have one of the characters '{' or '}' in their name.
 - Fixed selecting vias with the Ctrl key pressed.
 - Fixed a crash when changing the command text menu from within a button that has a submenu.
 - Fixed displaying vias/pads in EXPORT IMAGE in case the Vias/Pads layer is set to color 0.
 - Fixed generating bus names.
 - Fixed handling of '<', '>' and '&' in names and values when displayed in a Rich Text context.
 - Fixed drawing text origins when panning.
 - Fixed handling ASSIGNEd keys. They now really override hotkeys that are used for pulldown menus.
 - Fixed automatic ending of net and bus wires in case the newly drawn and the target wire are in a straight line and get optimized to form a single wire.
 - Fixed selecting vias of the same signal that have been wrongly placed at the same coordinates.
 - Fixed an unexpected popup menu in the CHANGE command.
 - Fixed a crash when printing under Linux without a defined printer.
 - Fixed placing a via in case there is a polygon wire at that location.
 - Fixed automatically setting junctions if "Auto end net and bus" is off.
 - Fixed displaying the Color column in the layer list of the CAM Processor.
 - Fixed handling polygons within packages in the Autorouter.
 - Fixed a possible loss of wire connectivity in the MOVE and PASTE commands with non-orthogonal angles.
 - Fixed assigning window icons when opening a script or ULP editor window from a project.
 - Fixed calculating signal polygons with a very large number of edges.
 - Fixed selecting gates in a device drawing with Ctrl+MOVE.
 - The MITER command now only inserts a new wire if the two existing wires are on the the same layer, have the same width and wire style.
 - Wires are now only optimized if they have the same wire style.
 - Fixed the ROTATE command if an object was selected with click&drag and the mouse button was released outside the editor window.
 - Fixed using the User Language builtin statements board, schematic, sheet, library, deviceset, package and symbol in a variable initialization, as in `int a = board;`
 - Fixed undoing a library update of a library that contained supply devices.
 - Fixed CUT/PASTE in case the pasted drawing contains modified packages that are only referenced by devices that are not contained in the target drawing.
 - Fixed a possible loss of consistency with implicit power pins in case a single segment net with the same name as these power pins was renamed or connected to an other net.
 - Fixed relocating wires after a library update (sometimes they could no longer be selected).

- Fixed handling signal polygons in the EXPORT IMAGE command that would completely vanish after being calculated (their outlines were printed as seen on the screen).
- Fixed an unnecessary window refresh when turning the MARK on or off.
- Fixed setting blind or buried vias at places where there are signal wires on layers that are not covered by those vias.
- Fixed focus handling when closing stacked dialogs in a ULP.
- Fixed slow script execution with NET, MOVE and ROTATE.
- Fixed handling the default button in a ULP dialog that contains a dlgTextView.
- Fixed handling recent files category names in the eaglerc file (were sometimes all uppercase under Windows).
- Fixed exporting monochrome images under Linux and printing with the "black" option under both Linux and Windows in case color 15 did not have the default value.
- Fixed handling the "File/Open recent" menus in script and ULP editor windows when loading them from a project.
- Fixed detecting a loss of consistency if a net class is defined (but not used) while f/b annotation is not active (maybe because the other drawing is not loaded).
- Fixed handling net classes when doing a PASTE into a schematic/board pair.
- Fixed handling micro vias on two layer boards (vias from layer 1 to 16 with a drill diameter less than the minimum drill diameter were wrongfully considered micro vias).
- Fixed handling orientation in the RECT command (now also accepts lowercase).
- Fixed setting the >LAST_DATE_TIME string when modifying the Design Rules.
- Fixed handling command text buttons in case of a "nothing to edit" error.
- Fixed the DRC distance check in case of dimension wires with a width of zero and a Copper/Dimension distance value of 0.2 or 0.3 micron.
- Fixed handling eaglerc entries that contain '='.
- Texts in signal layers with proportional or fixed font are no longer checked against each other for clearance errors in the DRC (same as has already been the case with vector font texts).
- Fixed selecting holes and junctions with Ctrl+MOVE.
- Fixed displaying the currently pressed button in the command text menu under Windows XP.
- Fixed placing a wire in the ROUTE command in case the length of the remaining airwire is below the Snap_Length, but a finishing wire can't be placed automatically due to layer constraints.
- Fixed the automatic ratsnest calculation in the ROUTE command (it didn't always calculate the shortest airwire).
- Fixed drawing wires in signal layers with the WIRE command to or from smds in case the smd part is mirrored or the wire is not in the same layer as the smd.
- Fixed setting the visible routing layers in the DISPLAY dialog according to the Design Rules for newly created boards.
- Fixed going into click&drag mode for off-grid objects with coarse grid.
- Fixed drawing smds with roundness 100 and orthogonal rotations with any of the CAM Processor devices that are derived from the "Generic" device (like EPS, for instance).
- Fixed handling circles with zero width (typically used in restrict layers) in the autorouter (it didn't route as close to them as it would have been allowed according to the design rules).
- Fixed a possible crash when moving one end of the last wire of a net segment onto the other end of that wire.

=====

* Library Management:

- Packages and Device Sets can now be copied into the currently edited library from other libraries, either through Drag&Drop from the Control Panel or by using the COPY command's new extended syntax (see "Help Copy").
- New package variants can now be created by directly using packages from other libraries, either through Drag&Drop from the Control Panel or by using the PACKAGE command's new extended syntax.
- The packages of the currently edited library can now be updated with those from other libraries, either through Drag&Drop from the Control Panel or by using the UPDATE command's new extended syntax.

* Blind & buried vias:

- The program can now handle so-called "blind & buried" vias. "Blind" vias are those that are not drilled all the way through the current layer stack. "Buried" vias are produced by drilling through the entire current layer stack. Vias that go all the way through the complete board are basically the same as "buried" vias, but sometimes are also referred to as "through" vias. "Micro vias" are small blind vias that go from one layer to the next inner layer. These are typically used to connect SMD pads to an inner layer, without having to run a wire away from the SMD.
- The Design Rules dialog now has a new tab named "Layers", in which the layer setup can be defined. The minimum drill size and aspect ratio for blind vias can be defined on the "Sizes" tab.
- When updating an existing board from an older version, the layer setup will be determined by the layers that are actually in use (either because there are objects in them, or they are supply layers, or they are used by the Autorouter setup). The layer stack will consist of a sequence of "core" and "prepreg" material (with the thickness of the individual layers chosen so that the final board results in a thickness between 1mm and 1.5mm), and which allows a via that goes through all layers. After loading an old board into this version you should verify the layer setup in the Design Rules and adjust it to your actual needs.
- The DISPLAY and LAYER dialogs (and related combo boxes) will only display those signal layers that are used in the layer setup.
- The CHANGE LAYER and ROUTE command only set the minimum necessary vias (according to the layer setup in the Design Rules). It may happen that an already existing via of the same signal is extended accordingly, or that existing vias are combined to form a longer via if that's necessary to allow the desired layer change.
- The VIA command has a new parameter that defines the layers this via shall cover. The syntax is from-to, where 'from' and 'to' are the layer numbers that shall be covered. For instance 2-7 would create a via that goes from layer 2 to layer 7 (7-2 would have the same meaning). If that exact via is not available in the layer setup of the Design Rules, the next longer via will be used (or an error message will be issued in case no such via can be set).
- The Autorouter cannot work with supply layers and non-through vias at the same time. In such cases you need to replace the supply layers with signal polygons accordingly.
- The CHANGE command has a new option named VIA, which can be used to change the layers a via covers. The syntax is

CHANGE VIA from-to *

where 'from' and 'to' are the layer numbers the via shall cover. If that exact via is not available in the layer setup of the Design Rules, the next longer via will be used (or an error message will be issued in case no such via can be set).

- The User Language object UL_VIA, now has two new data members 'start' and 'end', which return the layer numbers in which that via starts and ends. The value of 'start' will always be less than that of 'end'. Note that the data members 'diameter' and 'shape' will always return the diameter or shape that a via would have in the given layer, even if that particular via doesn't cover that layer (or if that layer isn't used in the layer setup at all).
- The DRC now checks whether all vias and objects in signal layers correspond to the actual layer setup. If they don't, a "Layer Setup" error is flagged.
- If the layer setup of a board contains blind or buried vias, the CAM Processor generates a separate drill file for each via length that is actually used in the board (see "CAM Processor").
- The DRC performs new checks for blind vias: vias that don't pass the check against the "Minimum Drill" parameter and are blind vias that are exactly one layer deep (so-called "micro vias") are checked against the "Min. Micro Via" parameter. Blind vias that pass these tests will further be checked to see whether they have a drill diameter that conforms to the "Min. Blind Via Ratio" parameter in "Edit/Design Rules/Sizes".

* Arbitrary angles:

- Texts and elements in a board context can now be rotated by any angle, in steps of 0.1 degrees (see "Help Add" for a description of the "orientation" flags).
- The new "Spin" flag in orientations can be used to disable the function that keeps texts readable from the bottom or right side.
- Pads and SMDs can now be placed with arbitrary angles.

* Arcs and Wires:

- In many aspects Arcs are now treated the same way as Wires. They are part of a signal when drawn in a signal layer, they can be used when drawing a polygon, and they now also have a wire style.
- The endings of arcs can now be either round or flat. You should use flat arc endings only when absolutely necessary (round endings have advantages when generating, e.g., Gerber files).
- The end points of an arc can now be moved independently, just like those of wires. When moving such points, the radius of the arc will be scaled accordingly.
- All commands that draw wires can now draw arcs by using the new 'curve' or '@radius' parameter (see "Help/Editor Commands/WIRE").
- There are no more 'arcs()' loop members in the User Language. Any ULPs that used to loop through arcs must now check the new data member UL_WIRE.arc when looping through the wires (see "Help/User Language/Object Types/UL_WIRE"). The "User Language" section below contains an example that shows how to adapt existing ULPs.
- The new command MITER can be used to take the edge off wire joins (see "Help Miter").
- The wire bend styles 0, 1, 3 and 4 now use an additional miter radius as defined with the MITER command.

* Additional flags for pads, vias and smds:

- Pads, Vias and SMDs now have additional flags that control the generation of the stop and cream masks, the thermals and the shape of the "first" pad within a package.
- The User Language objects UL_PAD, UL_VIA and UL_SMD have a new data member 'flags', which returns the setting of these flags (see "Help/User Language/Object Types/UL_PAD", "Help/User Language/Object Types/UL_VIA" and "Help/User Language/Object Types/UL_SMD").
- The PAD and SMD commands support the new options NOSTOP, NOTHERMALS, NOCREAM, and FIRST, respectively, to define these flags. The VIA command supports the new option STOP
- The CHANGE command has the new options STOP, CREAM, and FIRST to modify these flags (the THERMALS option already exists).

* User definable colors:

- The layer, background and grid colors are now completely user definable.
- There are now three "palettes" for black, white and colored background, respectively. Each palette has 64 color entries, which can be set to any RGB value. The palette entry number 0 is used as the background color (in the "white" palette this entry cannot be modified, since this palette will also be used for printing, where the background is always white).
- The color palettes can be modified either through the dialog under "Options/Set.../Colors" or by using the command

```
SET PALETTE <index> <rgb>
```

where <index> is a number in the range 0..63 and <rgb> is a hexadecimal RGB value, like 0xFFFF00 (which would result in a bright yellow). Note that the RGB value must begin with "0x", otherwise it would be taken as a decimal number. You can use

```
SET PALETTE BLACK|WHITE|COLORED
```

to switch to the black, white or colored background palette, respectively.

Note that there will be no automatic window refresh after this command, so you should do a WINDOW; command after this.

- By default only the palette entries 0..15 are used and they contain the same colors as previous versions.
- The palette entries are grouped into "normal" and "highlight" colors. There are always 8 "normal" colors, followed by the corresponding 8 "highlight" colors. So colors 0..7 are "normal" colors, 8..15 are their "highlight" values, 16..23 are another 8 "normal" colors with 24..31 being their "highlight" values and so on. The "highlight" colors are used to visualize objects, for instance in the SHOW command.
- The background color for layout and schematic can now be set to any color. Note, though, that in case the background color is neither pure black nor pure white, the drawing will be displayed layer by layer, which usually makes a window refresh slower than with black or white background.
- Changes to the "Options/Set..." dialog:
 - + The "Grid" Tab has been renamed to "Colors".
 - + The minimum visible grid size parameter has been moved to the "Misc" tab.
- The new User Language builtin function 'palette()' can be used to determine the currently used palette as well as the palette entries (see "Help/User Language/Builtins/Builtin Functions/Miscellaneous Functions/palette()").

* Control Panel:

- The tree view in the Control Panel can now be sorted by 'name' or by 'type' via the pulldown menu option "View/Sort".
- The Control Panel's pulldown menu option "File/Refresh tree" has been moved to "View/Refresh".
- Directory entries in the Control Panel's tree view which can contain libraries now all have the "Use all" and "Use none" options in their context menus.
- New context menu options for libraries, device sets and packages as well as Drag&Drop features for copying and updating library objects, and for creating new package variants.
- Drag&Drop of a board, schematic or library file into the appropriate editor window now loads the file into that window for editing. The previous functionality of performing a library update when dropping a library into any editor window has been removed.

* Design Rules:

- The new Design Rule parameters Shapes/Elongation can be used to define the elongation of Long and Offset shaped Pads. Valid values are from 0 to 200, where 0 results in a regular octagon shape (no elongation) and 100 gives you a side ratio of 2:1 (100% elongation), which is the ratio that has been hard-coded in previous program versions.
- The Design Rules dialog now has a new tab named "Layers", which defines the layer setup for multilayer boards (see "Help/Design Checks/Design Rules").
- The Design Rules tab "Shapes" contains a new combo box named "First", which defines the shape of the "first" pad within a package.
- The Design Rules tab "Sizes" contains the two new parameters "Min. Micro Via" and "Min. Blind Via Ratio".
- The Design Rules tab "Restrings" contains a new set of restring parameters for micro vias.

* User Language:

- The User Language member functions UL_PAD.shape and UL_VIA.shape now return PAD_SHAPE_ANNULUS, PAD_SHAPE_THERMAL, VIA_SHAPE_ANNULUS and VIA_SHAPE_THERMAL, respectively, if their shape is requested for a supply layer (see Help/User Language/Object Types/UL_PAD and UL_VIA).
- The User Language dialog object dlgListView now accepts a new parameter that defines the column and direction to use for sorting.
- The User Language functions strchr(), strstr(), strrchr() and strrstr() now accept an 'index' parameter to start the search at a given position.
- Opening the same file concurrently in two output() statements in a User Language Program is now treated as an error.
- The User Language objects UL_HOLE, UL_PAD and UL_VIA now have a new data member 'drillsymbol'.
- A User Language Program can now be aborted even if it is currently executing a lengthy 'for' or 'while' loop.
- The new ULP function status() can be used to display a message in the editor window's status bar.
- The User Language dialog function dlgTextView now accepts a second parameter to support hyperlinks in Rich Text (see "Help/User Language/Dialogs/Dialog objects/dlgTextView()").
- The User Language dialog function dlgMessageBox can now add an icon to the message box by prepending the message string with one of the characters '!', ';' or ':'. (see "Help/User Language/Dialogs/Predefined Dialogs/dlgMessageBox()").
- Due to the implementation of arbitrary angles and "spin" the following new

member functions have been added to the User Language:

UL_PAD.angle, UL_SMD.angle, UL_RECTANGLE.angle, UL_ELEMENT.angle,
UL_ELEMENT.spin and UL_TEXT.spin.

Make sure you take these into account in your own ULPs as necessary,
otherwise boards containing objects with these new features may be handled
incorrectly. See the 'dxf.ulp' for an example.

- Due to the modifications of pad shapes, the User Language constants
PAD_SHAPE_XLONGOCT and PAD_SHAPE_YLONGOCT have been replaced with
PAD_SHAPE_LONG, and the new constant PAD_SHAPE_OFFSET has been introduced.
- The new User Language member function UL_PAD.elongation returns the
elongation value for pads with shapes Long or Offset.
- The User Language object UL_VIA, now has two new data members 'start' and
'end', which return the layer numbers in which that via starts and ends.
The value of 'start' will always be less than that of 'end'. Note that the
data members 'diameter' and 'shape' will always return the diameter or shape
that a via would have in the given layer, even if that particular via doesn't
cover that layer (or if that layer isn't used in the layer setup at all).
- Due to the implementation of different arc cap styles the member function
UL_ARC.cap has been added to the User Language.
- The loop member functions UL_BOARD.arcs(), UL_PACKAGE.arcs(), UL_SHEET.arcs()
and UL_SYMBOL.arcs() no longer exist, since arcs are now treated a lot like
wires. Any ULPs that used to loop through arcs must now check the new data
member UL_WIRE.arc when looping through the wires (see "Help/User
Language/Object Types/UL_WIRE").

To convert an existing ULP that uses the arcs() loop member functions
consider the following example:

Assume you have a ULP that looks like this:

```
void ProcessArc(UL_ARC A) { /* do something with the arc */ }
void ProcessWire(UL_WIRE W) { /* do something with the wire */ }
board(B) {
    B.arcs(A) ProcessArc(A);
    B.wires(W) ProcessWire(W);
}
```

To make it run with EAGLE version 4.1 you need to eliminate the 'arcs()' call and move the actual arc processing into the ProcessWire() function:

```
void ProcessArc(UL_ARC A) { /* do something with the arc */ }
void ProcessWire(UL_WIRE W)
{
    if (W.arc)
        ProcessArc(W.arc);
    else
        /* do something with the wire */
}
board(B) {
    B.wires(W) ProcessWire(W);
}
```

Note that you only need this explicit handling of arcs if you actually
need to gain access to parameters only the UL_ARC can provide. If you are
not interested in that kind of information, you can handle the arcs just
like ordinary wires, using the parameters the UL_WIRE provides.

- To be able to handle any UL_ARC on UL_WIRE level the UL_WIRE object now
has the two additional members 'cap' and 'curve'.

- The User Language objects UL_PAD, UL_VIA and UL_SMD have a new data member 'flags', which returns the setting of the flags that control mask and thermal generation (see "Help/User Language/Object Types/UL_PAD", "Help/User Language/Object Types/UL_VIA" and "Help/User Language/Object Types/UL_SMD").
- The User Language object UL_HOLE has a new data member 'diameter[]' which returns the diameter of the solder stop masks.
- The output() statement in a User Language Program now supports the new mode character 'D', which causes the file to be automatically deleted at the end of the EAGLE session (see "Help/User Language/Builtins/Builtin Statements/output()").
- The User Language object UL_GRID now has an additional data member named 'unitdist', which returns the grid unit that was used to define the actual grid distance (see "Help/User Language/Object Types/UL_GRID").

* Autorouter:

- The Autorouter no longer attempts to route within the borders of the signal's surrounding rectangle first, because that way it sometimes was forced to take an "expensive" path, which it would have avoided if it had been allowed to use the entire board area in the first place. This may cause longer routing times in some cases, but may just as well speed up the routing, especially on complex boards.

* CAM Processor:

- The new parameter MaxApertureSize can be used in the 'eagle.def' file to define an upper limit for the size of the generated apertures for the GERBERAUTO and GERBER_RS274X devices. If objects larger than this limit are to be displayed, apertures will be emulated for them.
- If the board contains blind or buried vias, the CAM Processor generates a separate drill file for each via length that is actually used in the board. The file names are built by adding the number of the start and end layer to the base file name, as in

boardname.drd.0104

which would be the drill file for the layer stack 1-4. If you want to have the layer numbers at a different position, you can use the placeholder %L, as in

.%L.drd

which would result in

boardname.0104.drd

The drill info file name is always generated without layer numbers, and any '.' before the %L will be dropped.

Any previously existing files that would match the given drill file name pattern, but would not result from the current job, will be deleted before generating any new files. There will be one drill info file per job, which contains (among other information) a list of all generated drill data files.

- The aperture wheel file is now checked for duplicate D-codes (see "Help/Generating Output/CAM Processor/Output Device/Device Parameters/Aperture Wheel File").

* Text editor:

- Setting the font in a text editor window is now done via the pulldown menu option "File/Font..." and no longer via the printer setup. The selected font is now also used in the text editor window.

* ADD command:

- The ADD command now mirrors the object that is attached to the cursor when the center mouse button is pressed.

* ARC command:

- Arcs are now part of a signal if drawn in a signal layer of a board. When updating an existing board drawing, arcs in signal layers are transferred into signals (either newly generated ones or the ones that the arcs are apparently connected to by sharing the same end points).
- The ARC command now accepts a signal name (just like the WIRE command).
- The endings of arcs can now be either round or flat (the ARC command therefore accepts the new parameters ROUND and FLAT). When updating an existing drawing, the 'cap' parameter of all arcs in boards, packages and symbols, that have their endings covered by other objects (like wires or vias) will be set to 'round'. This allows them to be drawn more easily on the various output devices.

* BUS command:

- The BUS command now has an extended syntax to allow drawing arcs (see "Help/Editor Commands/BUS").

* CHANGE command:

- When changing the layer of a signal wire, only the minimum necessary via will be set (according to the layer setup in the Design Rules). It may happen that an already existing via of the same signal is extended accordingly, or that existing vias are combined to form a longer via if that's necessary to allow the desired layer change.
- The CHANGE command has a new option named VIA, which can be used to change the layers a via covers. The syntax is

CHANGE VIA from-to *

where 'from' and 'to' are the layer numbers the via shall cover. If that exact via is not available in the layer setup of the Design Rules, the next longer via will be used (or an error message will be issued in case no such via can be set).

- The CHANGE command can now change the cap style of arcs by using

CHANGE CAP ROUND | FLAT

- The CHANGE command has the new options STOP, CREAM, and FIRST to modify the new pad/smd flags (the THERMALS option already exists).
- The parameters Spacing and Isolate in the CHANGE popup menu now present a list of predefined values (just like, for instance, the Width parameter). All such popup menus now contain the entry "..." at the bottom, which brings up a dialog to enter a new value.

* COPY command:

- The COPY command can now copy parts in a schematic (see "Help Copy").
- The COPY command can now copy packages and device sets from other libraries into the currently edited library (see "Help Copy").
- The COPY command now mirrors the object that is attached to the cursor when the center mouse button is pressed.

* DELETE command:

- The option "SIGNALS" must now be written in full.

* DISPLAY command:

- The automatic enabling/disabling of related layers when activating or deactivating the t/bPlace or Symbols layer in the DISPLAY command can now be turned off by appending the line

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

to the eaglerc.usr file.

* DRC command:

- The DRC now checks for objects in the Pads and Vias layer that are not Pads or Vias (i.e. wires, rectangles etc.) and flags them as "Layer Abuse" errors. The reason for this is that EAGLE does not handle these object in any special way, so they might cause short circuits. If you get such an error from the DRC, you should move the object in question into the proper signal layer(s).
- The DRC now checks objects in the t/bKeepout layers only if the respective layer is activated.
- The DRC now checks whether all vias and objects in signal layers correspond to the actual layer setup. If they don't, a "Layer Setup" error is flagged.

* GRID command:

- The GRID command accepts the new option 'alt', which allows you to define an "alternate" grid that will be used whenever you press the Alt key while selecting or moving objects. The alternate grid can have its own size and unit, and is typically used to temporarily switch into a finer grid if the normal grid is too coarse. See "Help/Editor Commands/GRID".
- The GRID dialog has been changed to allow the user to enter the alternate grid parameters.

* INVOKE command:

- The INVOKE command now mirrors the object that is attached to the cursor when the center mouse button is pressed.

* MIRROR command:

- The MIRROR command now accepts the name of an element in a board, just like the MOVE command.

* MITER command:

- The new command MITER can be used to "take the edge off" wire joins (see "Help Miter").

* MOVE command:

- When picking up an object with the MOVE command, the status line now displays the same information as the SHOW command (currently this only works if the "User guidance" is turned off).
- If an Arc is selected at one of its end points, that point can now be moved freely (just like that of a Wire). The Radius of the Arc is then scaled accordingly.
- The MOVE command now mirrors the object that is attached to the cursor when the center mouse button is pressed.
- The MOVE command can now select objects at their origin by pressing the Ctrl key (see "Help/Editor Commands/MOVE").

* NET command:

- The NET command now displays information about the current net in the status line.
- The NET command now has an extended syntax to allow drawing arcs (see "Help/Editor Commands/NET").

* PACKAGE command:

- The PACKAGE command can now create package variants with packages from other libraries (see "Help Package").

* PAD command:

- The PAD command can now create pads with arbitrary angles and therefore accepts an "orientation" parameter (See "Help Pad").
- The pad shapes XLongOct and YLongOct have been renamed to Long. When updating an existing drawing from a previous version, XLongOct pads will be converted to Long pads with an angle of 0 degrees, and YLongOct pads will become Long with 90 degrees.
- The new pad shape "Offset" can be used to have pads that have the shape as defined by Long, but extend only to one side.
- The PAD command supports the new options NOSTOP, NOTHERMALS and FIRST to define the new 'flags' (see "Help/Editor Commands/PAD").

* PASTE command:

- The PASTE command now mirrors the object that is attached to the cursor when the center mouse button is pressed.

* POLYGON command:

- The 'width' and 'layer' can now be changed at any time while drawing a polygon.
- The POLYGON command now has an extended syntax to allow drawing arcs (see "Help/Editor Commands/POLYGON").

* RATSNEST command:

- The RATSNEST command now processes all points of a signal, even if that signal is very complex (in previous versions it dropped wire end points

from processing if the total number of connection points exceeded 254). This requires more memory when calculating the ratsnest. In case this is a problem on your system, you can revert to the original method by appending the line

```
Option.RatsnestLimit = "254"
```

to the eaglerc.usr file. The value given here is the number of connection points up to which all wire end points will be taken into account and thus limits the amount of memory used (processing will use up to the square of this value in bytes, so a value of 1024 will limit the used memory to 1MB). A value of "0" means there is no limit. A value of "1" will result in airwires being connected only to pads, smds and vias.

- RATSNEST no longer marks the board drawing as modified, since the calculated polygon data (if any) is not stored in the board, and the recalculated airwires don't really constitute a modification of the drawing.

* ROTATE command:

- The ROTATE command now accepts an "orientation" parameter (e.g. SMR359.9).
- The ROTATE command now accepts the name of an element in a board, just like the MOVE command.
- The ROTATE command can now be used with Click&Drag to rotate objects or groups by any angle (see "Help Rotate").

* ROUTE command:

- The ROUTE command now dynamically recalculates the current airwire while routing. In doing so, it also takes into account points along a wire, if those are closer to the cursor than the ends of that wire. If there is a pad, via or smd that is at most Snap_Length away from the end of the airwire (in the current layer), that end will now snap to the center of the object.
- The ROUTE command no longer automatically sets a Via at the end point of a wire. If you want to place a Via at the end point of a routed wire you can do so by holding the Shift key down while clicking at the end point.
- When determining the layer in which to route, the ROUTE command now also considers Wires (not only SMDs).
- When changing the layer in the ROUTE command, only the minimum necessary via will be set (according to the layer setup in the Design Rules). It may happen that an already existing via of the same signal is extended accordingly, or that existing vias are combined to form a longer via if that's necessary to allow the desired layer change.
- The ROUTE command now has an extended syntax to allow drawing arcs (see "Help/Editor Commands/ROUTE").
- The ROUTE command now creates a new airwire if necessary when Ctrl is pressed while selecting the starting point (see "Help/Editor Commands/ROUTE").

* SET command:

- The SET USED_LAYERS command also takes into account the layers from the new multilayer setup in the Design Rules and keeps them in the menus.
- The SET WIRE_BEND command accepts the two new values 5 and 6 to define bend styles that start or end in a 90 degree arc, plus the new value 7 for a bend style that results in an arc that exactly fits to the wire at

the starting point. If there isn't exactly one wire at the starting point, a straight wire is drawn. This bend style can be used to draw wires in sort of a "freehand" way.

- The special character '@' can be used with the SET WIRE_BEND command to define which bend styles shall actually be used when switching with the right mouse but (as in SET WIRE_BEND @ 1 2 4 5;).
- The SET command now restores the program default values for the parameter menus when executed as, for instance,

```
SET WIDTH_MENU;
```

(i.e. without any values). This applies to all *_WIDTH parameters.

* SHOW command:

- The SHOW command now displays the net class (in case of a net or signal) and the gate name (in case of a multi gate part).

* SMASH command:

- The SMASH command can now be applied to a GROUP
- Pressing the Shift key while clicking on a part or group with the SMASH command will now "un-smash" the object.
- The >PART and >GATE parameters are now also smashed.

* SMD command:

- The SMD command supports the new options NOSTOP, NOTHERMALS and NOCREAM to define the new 'flags' (see "Help/Editor Commands/SMD").

* SPLIT command:

- The SPLIT command now has an extended syntax to allow drawing arcs (see "Help/Editor Commands/SPLIT").

* UPDATE command:

- The UPDATE command can now update packages in a library (see "Help Update").

* VIA command:

- The VIA command has a new parameter that defines the layers this via shall cover. The syntax is from-to, where 'from' and 'to' are the layer numbers that shall be covered. For instance 2-7 would create a via that goes from layer 2 to layer 7 (7-2 would have the same meaning). If that exact via is not available in the layer setup of the Design Rules, the next longer via will be used (or an error message will be issued in case no such via can be set).
- The VIA command supports the new option STOP to define the new 'flags' (see "Help/Editor Commands/VIA").

* WIRE command:

- The WIRE command now has an extended syntax to allow drawing arcs (see "Help/Editor Commands/WIRE").

* Miscellaneous:

- If a net gets renamed because a Supply pin was placed on it, the user is now notified of this.
- Improved part placement in BOARD and PASTE command.
- The files created with EXPORT IMAGE now contain the image resolution in case the image format supports this.
- The RIPUP command can now be interrupted.
- The cursor is now switched to the "hour glass" while the Autorouter run.
- The size of the text origins is now limited to the actual size of the text.
- There is now a new item "Stop command" in the "Edit" pull down menu which has the same effect as the "Stop" button in the action toolbar.
- When printing on DOS based Windows systems (Windows 95, 98, ME) EAGLE can now render the drawing in memory and send the complete bitmap to the printer in order to overcome problems with printing texts on some printer drivers. This slows down printing, but at least it produces correct results. If you happen to have a printer driver that doesn't work correctly, you can turn this workaround on by setting the parameter

Printer.InternalRendering

in the eaglerc.usr file to a value other than the default "0".

The individual bits in the number each stand for a specific Windows version:

00000001 = Win32s
 00000010 = Windows 95
 00000100 = Windows 98
 00001000 = Windows Me
 00010000 = Windows NT
 00100000 = Windows 2000
 01000000 = Windows XP

You can use any combination of these bits in order to turn InternalRendering on or off for specific platforms. For instance the setting

Printer.InternalRendering = "6"

would turn this feature on only for Windows 95 and Windows 98.

If you had "Options/User interface/Always vector font" active because your printer wouldn't print non-vector fonts correctly, you may want to turn that option off now and try printing non-vector fonts. You may also need to turn off the "Persistent in this drawing" option for a particular drawing. Selecting the "Black" option in the PRINT dialog may speed up printing (in case you are printing to a black&white device).

- Printing under Linux now supports CUPS.
- When selecting an object in a densely populated area, the "Select highlighted object" message now also displays the information about that object, as would the SHOW command.
- Opening the same file concurrently in two output() statements in a User Language Program is now treated as an error.
- Error message dialogs now use the system defined sound effects.
- When connecting net segments the user is now always informed about the resulting name.
- The SIGNAL and PINSWAP commands now offer a selection if there are, for instance, two SMD pads on Top and Bottom at the same location.
- The DELETE command can now be interrupted when deleting a GROUP.
- The cursor now changes to the "hour glass" while processing polygons.

- Improved selecting smashed names/values in densely populated areas.
- If changing a package in a board results in connected pads of that element being moved outside of the allowed board area of the Light or Standard edition, the wires attached to these pads are now ripped up in order to comply with the board area limitations.
- Fixed handling of '\' at the end of script lines (the '\' inserted an additional blank, which caused problems with 'Description' lines in multiple EXPORT/SCRIPT of a library).
- An airwire in a dense area now triggers the selection mechanism even if the other objects belong to the same signal.
- Avoiding flickering status bar in library script with many EDIT commands.
- Changed mouse button handling under Windows to improve application button selectability.
- The progress bar in the status line of an editor window is now only displayed when it is actually active, and the percentage value is displayed outside of the bar under Windows.
- When moving a part in a schematic causes net wires to be automatically generated, both ends of these wires are now checked to see if any junctions are missing or can be deleted (this only works if "Options/Set.../Misc/Auto set junction" is active).
- Improved ERC's checks for unconnected net wires and missing junctions.
- The parameter toolbar in board context now contains a combo box where angles can be selected and entered (instead of the former four buttons for R0...R270).
- Panning is now done through Click&Drag with the center mouse button (no longer with the Ctrl button). If you want to have the old functionality back you can add the line

`Interface.UseCtrlForPanning = "1"`

to your eaglerc.usr file. Note, though, that the

Ctrl key is now used for special functions in some commands, so when using these special functions (like selecting an object at its origin in MOVE) with this parameter enabled you may inadvertently pan your draw window.

- Zero length airwires are now displayed as X-shaped crosses to improve their visibility.
- The new fill styles Stipple1, Stipple2, Stipple3 and Stipple4 (numerical values 12..15) can be used to have layers be drawn and erased without disturbing each other.
- Improved library update in case of device sets with a large number of package variants.
- When switching through the wire bend styles with the right mouse button (for instance in the WIRE command), the Shift key reverses the direction and the Ctrl key toggles between corresponding bend styles.
- When a Mark is active, the relative coordinates are now also displayed as "polar coordinates" (radius + angle), indicated by "(P ...)" in the coordinate display. This can also be used to measure the distance between any two points.
- Coordinates entered in the command line or in scripts can now be given relative to the mark, as polar coordinates, and can simulate a right mouse button click, which is mainly useful to select a group (see "Help/Editor Commands/Command Syntax").
- Dialog input fields that accept decimal numbers now automatically convert the ',' (or whatever the local decimal point is set to) into a real decimal point ('.').
- When loading a board/schematic pair that doesn't indicate consistency, a consistency check will now be performed automatically.

- The sheet selection combo box in the action toolbar now contains an entry named "remove", which can be used to remove the current sheet from the schematic.
- The highlighted objects from SHOW now stay highlighted if a WINDOW command is given.
- The Rich Text tag <author> no longer uses a smaller font.
- The relative coordinate display now uses at least the mark's precision.
- Editor windows now have a new menu entry "File/Open recent" which allows to easily reload recently used files.
- Increased grid display precision for mil and inch.
- No more popup message for undefined or empty group, rather 'beep' and status message.
- Avoiding unnecessary backups (for instance "*.b#1") when saving files that have not been modified.
- Fixed EXPORT PARTLIST in case of long names or values.

* Bugfixes:

- Fixed delimiting the undo buffer in CHANGE PACKAGE and CHANGE TECHNOLOGY.
- The "Change Isolate" menu option is now also available in the package editor.
- A polygon that consists only of two defined points is now rejected.
- Fixed removing an ASSIGNED key from the 'eaglerc' file.
- Fixed setting the default grid when changing the drawing type (package, symbol, device) inside a library.
- Fixed canceling a command by pressing the "Cancel" button in a dialog (for instance the NET command was not completely terminated).
- Fixed searching the 'eagle.scr' file in the current directory and handling directory paths in the -S option.
- Fixed skipping unsuitable objects when pasting between different drawing types.
- The User Language now generates an message if accessing a contact from a pin without a device context.
- Fixed switching off the DRC check "copper against t/bRestrict" (polygons in the t/bRestrict layers were still checked).
- Fixed renaming a package variant in the library editor.
- Fixed displaying pads/vias in case an additional signal layer is activated with the DISPLAY command.
- Fixed UL_AREA for packages and symbols in an UL_ELEMENT or UL_INSTANCE context, respectively.
- Fixed detecting an error condition in accessing an uninitialized User Language variable while it is being defined.
- Fixed storing NET_WIRE_WIDTH and BUS_WIRE_WIDTH in the eaglerc file.
- Fixed double file name entries in the Control Panel under Windows after renaming files with upper-/lowercase.
- Fixed refreshing the Control Panel tree after creating a new CAM job.
- Fixed handling CAM job descriptions that contain double quotes.
- Fixed handling mouse wheel events when zooming in/out (previously the given "Mouse wheel zoom" factor was applied twice for each event).
- Fixed setting the focus in the DISPLAY and LAYER dialogs.
- Fixed displaying 'not saved' in >LAST_DATE_TIME on printed output.
- Fixed entering negative values in 'real' dialog fields.
- Fixed replacing an identical package from a different library.
- Objects that don't cover any raster point are now virtually enlarged by the Autorouter to make them cover at least one raster point, so that the Autorouter can "see" them.
- Fixed a problem when deleting a layer from within the DISPLAY dialog.

- Fixed updating library devices with newly added packages.
- Fixed handling value on/off in the library update.
- The BOARD command no longer copies all package variants into the newly created board, but rather just the ones that are actually used. When loading a board from a previous version, the superfluous packages will automatically be deleted.
- The Autorouter sometimes didn't route between objects where it actually should have been able to do so.
- The 'Cancel' button sometimes didn't work when a message box was displayed while, for instance, running a script.
- The ADD dialog didn't display all used libraries after a device has been added from the Control Panel.
- Fixed handling layer of mirrored text wires in UL_TEXT.wires.
- Fixed drawing objects attached to the cursor when doing Ctrl+MouseMove (wasn't drawn correctly under Windows).
- Fixed the Technology parameter in EXPORTed library scripts (the values were not quoted).
- Fixed mirroring labels in a schematic.
- Fixed a crash if INVOKE was used in a board.
- Fixed 'Replace all' in the text editor for several occurrences in the same line with different lengths of find and replace text.
- Fixed a bug in the library update of standalone boards in case a pad was removed from a package.
- Fixed CHANGE PACKAGE in case both devices have no connects.
- Fixed flickering smashed texts when dragging parts.
- Fixed displaying wires when zooming far into the drawing.
- Fixed handling empty packages and symbols in CUT/PASTE.
- Fixed setting the size of the ERRORS window.
- The 'scale factor' and 'page limit' in the PRINT dialog are now limited to values ≥ 0 .
- Fixed handling of net classes when renaming nets.
- Added missing 'L' parameter to the format statement (FS) of GERBER_RS274X in eagle.def.
- Fixed closing a project if the user doesn't have write access to the directory.
- Fixed setting user defined default Design Rules when loading an existing library.
- Fixed handling of objects that don't belong to any signal in the DRC, in case there are no signals with the default net class.
- Fixed the header for Postscript files in 'eagle.def'.
- Fixed UL_VIA.diameter[] for LAYER_TSTOP and LAYER_BSTOP in case the drill isn't larger than the stop limit in the Design Rules.
- Fixed ADDING devices with '(' or ')' in their name.
- Fixed a crash in GROUP/CUT in case only net labels where in the group.
- Fixed handling packages that contain both pads and smds.
- Fixed handling multiple smashed '>NAME' etc. texts in User Language Programs.
- Fixed handling the layer of polygons when doing GROUP/CUT/PASTE with mirroring.
- Fixed printing hatched polygons with "pos. Coord" option in the CAM Processor.
- The CAM Processor no longer complains about layers that are enabled in the job but not used in the drawing.
- Fixed 'dxf.ulp' to correctly display solder stop masks for pads, vias and holes.
- Fixed rotating with right mouse button while moving an object with click&drag.

- Fixed deleting a net wire that was connected to a supply pin (the remaining net wasn't renamed accordingly).
- Fixed setting the >DRAWING_NAME parameter when creating a board from a schematic.
- Fixed handling wires that ended exactly on a vertical polygon edge (the polygon was not considered to be connected to that wire).
- Fixed closing ULP include file after an error message (text editor was unable to save the edited file under Windows).
- Fixed checking empty (value) texts in the DRC.
- Fixed snapping bended wires if "Options/Set/Misc/Snap bended wires" is on.
- Fixed buffer overflow in printf() with very long '%*' formats in ULPs.
- Fixed handling SET WIRE_BEND while drawing a wire.
- Fixed calculating zero length airwires in case there is another wire end point very close to these coordinates.
- Fixed searching the library in the library path in case of "ADD part@libraryname".
- Fixed execution of 'eagle.scr' in case there is a command following the EDIT command in the command line.
- Fixed EXPORT SCRIPT in case the library description contains the backslash character.
- Fixed emulating thin arcs on photoplotters.
- Fixed some superfluous '*' in the Gerber RS274X output format.
- Fixed an abort if a ULP accesses a schematic's nets from a board via project.schematic(SCH).
- Fixed the location of pad names for pins in mirrored instances in the User Language.

Release notes for EAGLE 4.09r2

=====

* CAM Processor:

- The new command line option '-N' can be used to turn off the command line message prompts in the CAM Processor. This is mostly useful for fully automated CAM batch jobs.

* Bugfixes:

- Fixed a case where the original pad/via shape was drawn in a supply layer (in addition to the thermal/annulus symbol). This also may have caused the autorouter to route a signal that actually had a supply layer (and therefore didn't have to be routed).
- Fixed the DRC check between copper and Dimension lines with zero width.
- Fixed reacting on system color changes in the Help window under Windows.
- Fixed handling Pwr pins in the library update (consistency could be lost if the pins had a different sequence in the new symbol).

Release notes for EAGLE 4.09r1

=====

* Bugfixes:

- Fixed drawing SMDs with Roundness values other than 0 or 100 on photo plotters with fixed aperture wheels. Note that aperture tolerances are relative to the final object dimensions, so when drawing SMDs with Roundness the aperture actually used may be larger, even if only a negative tolerance value was specified. Also, the difference between the

initially requested aperture and the actually used one may be a lot bigger than the absolute value of the tolerance would indicate. If you are processing CAM data for a board that contains SMDs with Roundness and you get an "Aperture missing" error regarding a Draw aperture, you can try specifying a negative draw tolerance of a few percent to enable drawing of the rounded SMDs (which will then be drawn as if they had a larger Roundness value).

- Fixed handling GROUP rectangles with zero width (which caused a program crash).
- Fixed printing drill symbols with Postscript in the CAM Processor in case the Pads or Vias layer was not activated.
- Avoiding unjustified 'close but unconnected...' warnings in the ERC.
- Fixed net class handling in forward annotation when splitting a net.
- Fixed library update in case of CUT/PASTE from one schematic into another (there was no error message if a package variant was no longer available).
- Added the missing "Cam Processor" entry to the pulldown menu in the schematic editor.
- Fixed handling description when checking for modified CAM job.

Release notes for EAGLE 4.09

=====

Version number changed for final release.

Release notes for EAGLE 4.08r2

=====

* Bugfixes:

- Fixed deleting net segments from pins that overlap with another segment (in such cases it could have happened that the consistency between board and schematic was lost).

Release notes for EAGLE 4.08r1

=====

* Bugfixes:

- Fixed a problem when closing a window and canceling the "Save?" dialog.
- Fixed library update in ADD when package variant is missing.

Release notes for EAGLE 4.08

=====

* Bugfixes:

- Fixed another unjustified error message "Load error 293".

Release notes for EAGLE 4.07

=====

* Bugfixes:

- Fixed unjustified error message "Load error 293".
- Fixed a possible crash when closing a window in a script.

Release notes for EAGLE 4.06

=====

* Bugfixes:

- Fixed checking rectangular objects versus circular objects in the DRC.
- Fixed installing executable file and manual page under Linux.

Release notes for EAGLE 4.05

=====

* Bugfixes:

- Fixed polygon calculation under Windows.
- Fixed canceling the Autorouter during ripup.
- Fixed handling circles in the autorouter (sometimes it routed right through them).

Release notes for EAGLE 4.04

=====

* DELETE command:

- Net segments connected to a bus now keep their name when splitting segments.
- Added a note to the online help of the DELETE command, regarding ripup of signal wires when deleting a part with SHIFT+Delete.

* ERC command:

- In order to handle board/schematic pairs that have only minor inconsistencies, the user can now enable a dialog that allows him to force the editor to perform forward-/backannotation, even if the ERC detects that the files are inconsistent. This can be done by appending the line

`Erc.AllowUserOverrideConsistencyCheck = "1"`

to the eaglerc.usr file.

PLEASE NOTE THAT YOU ARE DOING THIS AT YOUR OWN RISK - if the files get corrupted in the process, there may be nothing anybody can do to recover them. After all, the ERC *did* state that the files were inconsistent!

- The ERC now reports an error if supply pins with the same name are overwritten with more than one signal.

* EXPORT command:

- The EXPORT SCRIPT command now sets the grid unit to 'mm' if the current grid isn't metric, in order to avoid loss of precision.

* INFO command:

- The INFO command now displays the VIA diameter separately for outer and inner layers (as determined by the Design Rules), plus the value that has been originally defined by the user.

The format is "Diameter = outer/inner (original)".

* OPTIMIZE command:

- The OPTIMIZE command now also optimizes the "flat" wires in a board (i.e.

those not being part of a signal).

* PACKAGE command:

- The PACKAGE dialog now accepts an empty "Variant name" field and no longer requires the explicit entry of " (two single quotes) to define a package variant with an "empty" name.

* TECHNOLOGY command:

- The TECHNOLOGY command no longer asks before removing a Technology from a device.

* WIRE command:

- In order to speed up execution of large scripts that produce many wires, 'Set Optimizing Off' now also disables automatic wire splitting in scripts.

* User Language:

- When running a ULP the internal search path for images will now be set to the ULP's directory.
- The help page for Rich Text now contains a list of the supported image formats for the "<img...>" tag.
- The new member function UL_LAYER.used can be used in a ULP to check if a given layer is actually used in a drawing.
- The new builtin constants "path_...[]" and "used_libraries[]" can be used to determine the entries in the "Options/Directories" dialog and the currently used libraries from within a ULP.
See "Help/User Language/Builtins/Builtin-Constants".
- The new mode character 'F' in the 'output()' statement of the User Language can be used to force opening a file with an otherwise protected extension (*.brd, *.sch or *.lbr).

* Miscellaneous:

- Improved speed of CAM Processor.
- NET and SIGNAL commands are no longer terminated when canceling a 'Connect...?' dialog.
- Pad and via drill holes are no longer subtracted from polygons.
- The position of the "splitters" in dialogs like for instance "ADD" are no longer stored as percentage values in the file eaglerc.usr, but rather as absolute pixel values. The first time such a dialog is opened with this program version the splitter positions may therefore not be the same as when the dialog was left last time.
- Polygons that would completely disappear after being calculated with RATSNEST are now shown on the screen with their original outlines. This allows editing or deleting them.
- The string edit dialog now adjusts its width to completely show longer strings (for instance when editing a long bus name).
- Command text buttons (defined with the MENU command) now stay pressed to indicate the currently active command (this does not apply to buttons that result in a submenu).
- The Windows file dialog no longer checks for the existence of a file, thus allowing the user to leave out the filename extension when typing in a file name.
- The command button groups are now separated by horizontal lines, which saves

- space in the command menu.
- The default value for the solder stop mask ("Edit/Design rules/Masks/Stop") has been changed to a fixed value of 4mil.
- The size of SMD names displayed when SET PAD_NAMES is on is now limited to 20mil.
- The "Description" field in the library editor and the right pane in the Control Panel can now be completely minimized with the splitter.
- If a package variant or technology is no longer there (e.g. because it has been deleted or renamed) the user can now select a different one in the library update.
- The automatic unit determination in dialog input fields can now be controlled by appending the line

Interface.PreferredUnit = "x"

to the eaglerc.usr file, where "x" can be
 "0" for automatic unit determination (default)
 "1" for imperial units
 "2" for metric units.

- The 'dxf.ulp' now has radio buttons to choose the unit of the generated data ('mm' or 'inch'), and outputs data in 'mm' with 4 decimal digits instead of 3.
- When using the CAM Processor in command line mode it no longer stops if a warning message is given, but rather offers a prompt that allows the user to decide whether to continue or not.

* Bugfixes:

- Fixed handling of upper/lowercase umlauts in removing/renaming library objects.
- Fixed error handling if a non existing member function was used in a ULP.
- Fixed library update error 403-1362.
- Fixed library update in case a device set is no longer present in the library.
- Fixed printing Thermal symbols.
- Technologies can no longer contain lowercase characters.
- Fixed handling devices with double quotes in their name in the Control Panel.
- Fixed focus handling in 'Connect Nets' dialog.
- The internal object counters are now checked against a safety limit in order to avoid overflows.
- Fixed optimizing wires after MOVE.
- Fixed layer in UL_POLYGON.contours and UL_POLYGON.fillings in case of mirrored polygons.
- Fixed handling of PWR pins in library update.
- Fixed exporting images.
- Fixed handling printer specific page sizes under Windows.
- Fixed updating libraries in case packages had been replaced in the board.
- Empty values of smashed parts are no longer selectable by mistake.
- Fixed handling '\ ' in the "Options/Directories" dialog under Windows.
- Fixed displaying User Language dialogs under Windows.
- Fixed handling Technology values that add up to exactly 13 characters.
- The EXPORT SCRIPT command now explicitly generates a "-" if a device does not contain the empty Technology.
- Fixed handling lowercase characters when deleting Technologies.
- Fixed splitting net segments with supply symbols.
- Fixed EXPORTing net class for implicit PWR pins.
- Fixed optimizing wires after deleting a group.

- Fixed handling the PRINT option '-0'.
- Fixed handling lines longer than 1000 characters in eaglerc.
- Fixed handling "SET BEEP ON|OFF".
- Fixed a possible crash that happened under Windows if the system date was outside the range 1970..2037.
- Fixed parsing '(' in the command line (caused an error message when renaming a package variant that contained this character).
- Fixed a possible crash that could happen if a part on a sheet was deleted with SHIFT+Delete, and that part was the last of this type in the schematic.
- Fixed 'Shift-Deleting' parts with gates on different sheets.
- The terminating ';' was missing for "Hole..." in exported scripts.
- Fixed handling "WRITE @filename;".
- Fixed handling directory paths after navigating with the file dialog.
- Fixed the "...not saved" dialog in case an other board should be edited after a modification that changed both the board and the schematic (it didn't list the schematic as modified).
- Fixed setting the net class of a newly drawn net or signal in case the net/signal name is specified in the command line.
- Fixed library update in case a non-pwr pin became a pwr pin.
- Fixed handling unconnected supply pins. They are now implicitly mapped to the same net name as any other overwritten supply pin with the same name.
- Fixed handling "&xxx;" tags in the tree view of the control panel.
- When drawing a new net/signal segment towards an existing one, the net class of the existing net/signal is no longer overwritten.
- "File/Open/Text..." now always opens a text editor window, even if the file name has an extension that would indicate a different kind of window (like ".cam").
- The library editor window is now refreshed after executing a script.
- Closing a "Description" or "Connect" dialog with the system's close button now reacts correctly if the user selects not to discard changes.
- When opening a file from the Control Panel with "File/Open/..." the proper filename extension is now added if it is missing.
- Fixed naming nets if the NET command contains an explicit net name.
- Fixed delimiting the undo buffer in the REPLACE command.
- Fixed setting the directory in "Save as...".
- Fixed handling command line PRINT parameters.
- Fixed selecting objects that are close together.
- Fixed using expressions as title in a dlgGroup.
- The User Language no longer returns wires in layer 0.
- Fixed PASTEing nets with different net classes than an existing net of the same name.
- Fixed toolbar positioning.
- Fixed checking user defined print borders against the printer driver's minimums.
- Fixed library update in case of pins with '@' in their name.
- Fixed handling pin names with '@' in the User Language.
- Fixed handling parameter strings (like '>NAME') in ULPs in case the editor window was refreshed after a dialog.
- Fixed handling the '>GATE' parameter String in ULPs.
- Fixed UL_TEXT.wires() for mirrored texts of smashed elements.
- Fixed calculating ratsnest when adding a part with many Pwr pins (this took a very long time).
- Fixed page size of PS_DINA3 device in CAM Processor.
- Fixed the polygon subtractor to handle cases where many identical objects have to be subtracted from the same polygon.
- Fixed closing a window in a script that has been called from an other window.

- Fixed version 3.5 compatibility mode in PACKAGE command.
- Fixed the UL_PIN.contact function if called from a UL_PINREF context.

Release notes for EAGLE 4.03

=====

WARNING: Due to some necessary changes in the data structure once you edit a file with version 4.03 you will no longer be able to edit it with earlier versions!

PLEASE MAKE BACKUP COPIES OF YOUR CURRENT BOARD-, SCHEMATIC- AND LIBRARY-FILES BEFORE EDITING THEM WITH VERSION 4.03!

* ADD command:

- The ADD dialog now has a 'Drop' button that removes a library from the list of used libraries.
- The ADD command can now fetch a particular gate from a device, without automatically fetching the MUST and ALWAYS gates, too.
- Some users always want to use the device name as part value, even if the part needs a user supplied value. Those who want this can now append the line

```
Sch.Cmd.Add.AlwaysUseDeviceNameAsValue = "1"
```

to the eaglerc.usr file.

* DELETE command:

- The DELETE command now deletes an entire part when clicking on a gate with the Shift key pressed. In that case, the wires connected to the element in the board will not be ripped up.
- The DELETE command now deletes the entire polygon when clicking on a polygon wire with the Shift key pressed.
- The DELETE command now deletes the entire net or bus segment when clicking on a net or bus wire with the Shift key pressed.

* ERC command:

- Since some people did not appreciate the new checks implemented into the ERC, these can now be suppressed by appending the line

```
Erc.SuppressAdditionalWarnings = "1"
```

to the eaglerc.usr file.

- The ERC now reports uninvoked MUST gates as errors.
- The ERC now checks whether nets and signals have consistent net classes.
- The ERC no longer checks pin overlaps within the same gate.

* EXPORT command:

- The EXPORT IMAGE command now accepts the new keyword MONOCHROME to produce a black&white image.
- EXPORT IMAGE now displays a 'wait' cursor if the operation takes some time.
- EXPORT NETSCRIPT now outputs the net classes.

* INFO command:

- The INFO command now displays the layer name.

* LAYER command:

- The new option '??' can be used to suppress error messages when deleting a layer.

* UPDATE command:

- The UPDATE command now accepts the parameters '+@' and '-@' which allow renaming the libraries contained in a drawing.

* VALUE command:

- Some users don't want the warning message about a part not having a user definable value, so that warning can now be disabled by appending the line

```
Warning.PartHasNoUserDefinableValue = "0"
```

to the eaglerc.usr file.

* Design Rules:

- The DRC check "copper against t/bRestrict" can now be completely turned off through "Misc/Check restrict" in the Design Rules dialog. By default any copper objects in the Top and Bottom layers are checked against objects in t/bRestrict.

- There are now checkboxes on the "Restrstring" tab that can be set to enable using the actual pad or via diameter in inner layers, too. By default this option is off for newly created boards, but will be set when updating a board from version 3.5 or earlier, since previous versions used the same pad/via diameter on all layers.

TO IMPLEMENT THIS FEATURE IT WAS NECESSARY TO MODIFY THE DATA STRUCTURE IN A WAY SO THAT FILES EDITED WITH VERSION 4.03 CAN NO LONGER BE LOADED INTO EARLIER VERSIONS (OTHERWISE THE RESULTS MIGHT NOT BE CORRECT).

- The Restrstring parameters in the Design Rules are no longer set to 0 when updating a board file from version 3.55 or earlier. They are now left at their default values and if the file contains any pads or vias that would become larger due to these new parameters, the user will be prompted to confirm this and to adjust the Design Rules if necessary. When the CAM Processor loads a version 3.55 (or earlier) file, the Restrstring Parameters will still be set to 0, which makes sure that the result will be the same as in previous versions. When loading a file from version 4.0 or 4.01, the restrstring parameters will be checked, and if they are 0 the user will be prompted to confirm this and to adjust the Design Rules if necessary.

- The DRC now checks the Design Rules for plausibilities.

- The DRC now only checks pads and smds, as well as any copper objects connected to them, against dimensions.

* Polygons:

- The polygon named _OUTLINES_, which is used for calculating outlines, no longer needs to have an Isolate parameter of 0 (See "Help/Generating Output/Outlines data").

- When calculating the _OUTLINES_ polygon, objects in the layers t/bRestrict and Dimension were wrongly subtracted.

* User Language:

- New data members UL_PART.deviceset, UL_DEVICESET.library, UL_DEVICE.library and UL_SYMBOL.library.

* CAM Processor:

- The CAM Processor can print schematics again.

* Miscellaneous:

- Improved selectability of arcs.
- Speeded up saving a file when there are many entries in the CP tree and the network is slow.
- The "Always vector font" flag that is stored in a drawing file can now be controlled via "Options/Set/Persistent in this drawing" (previously this was only possible by entering the SET command in the command line).
- The vector font characters '&', '@' and '~' have been improved.
- The 'dxf.ulp' now outputs texts always with the builtin vector font by default (this can be changed through a new checkbox in the dialog).
- Junctions are now also set automatically when a part is placed, and they will be automatically deleted if a part or a net wire is deleted.
- The automatic opening of the project folder at program start (or when activating a project by clicking on its gray button) can now be disabled by appending the line

```
ControlPanel.View.AutoOpenProjectFolder = "0"
```

to the eaglerc.usr file.

* Bugfixes:

- The Autorouter sometimes routed too close towards pads, which could result in short circuits (this only happened when the pad shape was different in the various routing layers).
- A crash could happen in EXPORT IMAGE unter Linux with 8bpp X-servers.
- A crash could happen when removing a sheet with the REMOVE command.
- When using the CAM Processor device GERBER_RS274X to generate data for a supply layer, the thermal gaps were missing.
- When printing several sheets the caption was not printed correctly under Windows with some printers.
- The CAM Processor's GERBERAUTO didn't always create all apertures when pads or vias had different shapes in the various layers.
- Resource usage under Windows was drastically reduced.
- Entering command names that were too long caused a crash.
- The values used with the CAM Processor command line options -h, -w, -x and -y have been wrongly rejected.
- Values for Width and Height defined in 'eagle.def' were not copied into the CAM Processor window, and values entered in the CAM Processor were not used.
- CUT didn't work with the right mouse button.
- The library update mechanism sometimes didn't recognize different symbols or packages, which made it impossible to add certain parts to a schematic.
- If the resulting bitmap was too large, the EXPORT IMAGE sometimes crashed or didn't create any output (without an error message).
- When creating new library objects it was possible to enter names that

- contained blanks.
- When EXPORT IMAGE was used in a newly loaded library with no object being edited, the program crashed.
 - A possible crash under Windows that happened when the program was left with Alt-X was fixed.
 - The User Language function fileglob() didn't work if the filename pattern contained a blank.
 - Sometimes some layers were not displayed in the CHANGE PACKAGE preview.
 - The selected window area was not restored correctly when a schematic was loaded from a project file.
 - Avoiding ERC warnings about net wires being too close if there is a bus wire at that point.
 - Under Windows the UPDATE command didn't search for libraries case insensitively.
 - Fixed changing the layer of a polygon in a package.
 - When printing more than one sheet on Windows 2000 the offset of the second and following sheets was incorrect with some printer drivers.
 - Empty lines in text files were not printed.
 - Editing the DESCRIPTION of a directory produced faulty line ends under Windows.
 - There was no error message when the DESCRIPTION file couldn't be written.
 - Speeded up DRC processing and deleting errors when the large cursor is used.
 - Reading a Design Rules file didn't work if the description contained the '=' character.
 - When a package was renamed, the package variant list wasn't updated.
 - The Control Panel did not open the tree to the initial project if the "Projects" path contained more than one entry.
 - The Control Panel tree was not refreshed after operations like rename, copy etc.
 - The chapter describing the User Language function 'lookup()' was missing from the German online help under Windows.
 - Relative symbolic links were not expanded correctly under Linux.
 - Pads and vias didn't keep their drills open inside signal polygons.
 - The DRC now reports texts in signal layers that don't use the builtin vector font (this check can be disabled through "Check texts" on the Design Rules' "Misc" page).
 - The DRC and Autorouter sometimes crashed when the board contained a large arc or circle.
 - Polygon wires no longer react on CHANGE STYLE.
 - The SIGNAL command did not ask the user whether to combine newly defined signals with existing ones, but rather silently connected them.
 - When a dialog is canceled, any running script will now also be canceled.
 - Previously entered text in the text editor's 'Find' dialog is now selected.
 - Avoiding 'wait' cursor in dialogs during lengthy operations.
 - EXPORT IMAGE didn't use the board's Design Rules.
 - A UL_GATE that has been derived from a UL_INSTANCE did not provide the correct x and y values.
 - When loading an Autorouter control file that contained a routing grid with a precision that exceeded the internal editor resolution of 0.0001mm, the routing grid was rounded to the internal resolution.
 - When a file was opened using the Control Panel's context menu, the buttons in the editor window's toolbars didn't work correctly under Windows.
 - The program crashed when a device was added to a schematic via Drag&Drop from the Control Panel, and the library needed to be updated from an older version.
 - Some paper sizes were not handled correctly in the PRINT command under Linux.

- When a device that supports colors has been used in the CAM Processor, selecting "Layer/Show selected" or "Layer/Show all" disabled the display of the color numbers in the layer list.
- Removing a key assignment with "ASSIGN key;" didn't work.
- The ADD command didn't add a new part if a part name was given and there were still unused NEXT gates of that type available in an already existing part.
- The 'Stop' button wasn't disabled correctly when pressed in the LABEL command.
- Supply layers were not printed correctly if the print was done rotated.
- Fixed missing cursor in text editor if the first character in a line is something like '{'.
- When using the MOVE command with "click&drag" mode, releasing the mouse button outside the drawing area is now ignored (as was the case in version 3.55).
- The next command was sometimes skipped when an error message was given in a script file.
- When routing a large board with many layers and a fine grid, the autorouter sometimes needed extremely much memory. This has been improved to make it work with less memory.
- When using a fine routing grid the Autorouter sometimes inserted an extra (jagged) wire segment at the starting point of a trace.
- When the routing grid was not an integer multiple of the internal editor resolution of 1/10000mm (e.g. 6.25mil), the Autorouter sometimes didn't round the resulting coordinates correctly.
- Fixed updating the contents of a dlgComboBox when a statement is executed.
- Fixed the behaviour of the "Again" function in the text editor in case nothing has been searched yet.
- Fixed setting colors, fill styles and long strings when adding new layers from a library to a board or schematic drawing.
- Fixed processing textual coordinate values (sometimes in script files only every other coordinate pair was recognized).
- Fixed recalculating the "Window Fit;" area of a sheet after a library update.
- User Language Dialogs sometimes became inoperative after canceling a message box.
- The bounding box of texts will be recalculated in order to fit exactly to the text size when using vector font.
- The CAM Processor didn't correctly write the offsets into the info files.
- The PASTE command in a schematic didn't always forward annotate the net classes of the pasted nets correctly.
- Fixed handling empty strings in dlgComboBox.
- Fixed updating libraries in the PASTE command.
- Sometimes small white "dots" appeared when a file was newly loaded. These also caused zero-length/zero-width apertures to be generated in Gerber output.
- When renaming a file in the Control Panel, a possibly existing target file is no longer overwritten.
- Sometimes under Windows a click on a push button was not recognized when the mouse button was released while moving the mouse pointer away from the push button.
- Fixed handling parameters (e.g. package variant names) containing '='.
- Fixed creating new projects and folders in case several branches in the Control Panel tree view contained the same directory.
- Sometimes thermal and annulus symbols were not correctly generated for supply layers if a window refresh in the board editor was caused while the CAM Processor was running.

- CHANGE PACKAGE sometimes didn't map the pad assignments correctly, which could cause wrong connections in the board and a loss of consistency.
- The ROUTE command no longer generates a via at points where there is already a wire in the same layer.
- The layer of SMDs was sometimes not recognized correctly.

Release notes for EAGLE 4.01

=====

* Bugfixes:

- The 'bom.ulp' didn't print the "Description" if no database was in use.
- 'bom.ulp' now uses '.htm' as default extension when saving as HTML.
- Added missing warning messages to the ERC help page.
- Holes will be subtracted from signal polygons even if the Distance parameter for Copper/Dimension in the Design Rules is 0 (for better compatibility with version 3.55).
- Fixed restoring the visible area in a schematic when loading a project.
- Fixed handling umlauts in NAME command.
- Sometimes text files were not correctly recognized and thus not listed in the Control Panel's tree view.
- Making sure a very small circle with Width=0 is drawn as at least one pixel on the screen (this is sometimes used as a reference marker).
- Fixed not limiting the zoom factor for small drawings.
- Avoiding flickering hour glass cursor when loading many libraries in ADD.
- No longer displaying text origins in device set drawing.
- Fixed checking whether a pad or via is connected to a polygon.
- Fixed calculating airwires between partial polygons.
- Checking if coordinates are within the allowed range.
- Fixed calculating polygons with widths less than 1 micron.
- The "Use all/none" option was missing in the context menu of the "Libraries" entry in the Control Panel if there was only one entry in the "Options/Directories/Libraries" path.
- Sometimes a faulty "used" indicator appeared in the Control Panel on directories that are not project directories.
- After loading Design Rules from a file into the Design Rules dialog, the percentage values of e.g. the Restricting parameters were not updated correctly.
- Fixed switching back the mouse cursor from the four-way selection cursor.
- Fixed editing the library description from a symbol drawing.
- The "mouse click" symbol was not displayed correctly in the Linux help.
- Changing "Options/Set/Display mode" did not propagate the parameter change to the other windows.
- Polygons and non-continuous wires in packages and symbols were not handled correctly in the User Language.
- Since version 4.0 a bus name that is used with a range can no longer end with a digit. However, if such a name was already present in a drawing from a previous version and a net was extracted from that bus, the program crashed. Now there will be an error message in such a case.
- If a library was saved and parts from that library were added very quickly (within the same second, for example in a script) it could happen that the current version of the library was not used, but rather the previously loaded version.
- The obsolete description of DISPLAY_MODE FAST has been removed from the online help.
- Fixed a bug in the library update mechanism that caused an error code of 400-1363.

Release notes for EAGLE 4.0

=====

* Control Panel:

- The Control Panel now has a "Tree View" which provides an overview over all areas of EAGLE, like Libraries, User Language Programs, Projects etc.
- The Control Panel's tree view supports "Drag&Drop" to copy or move files and directories. Files (like ULPs or scripts) can also be dragged over an editor window and dropped there; this results in the file being RUN or executed by the SCRIPT command inside the respective window. Devices and packages can also be dragged and dropped onto editor windows. Dropping an entire library onto a board or schematic editor window will perform a library update.
- Objects in the tree view have a context menu that can be accessed by pressing the right mouse button.
- The menu option "Save project as..." is no longer available. New projects can now be created via the context menu in the "Projects" tree item, or by selecting "File/New/Project" from the Control Panel.
- The path settings in "Options/Directories" can now use the special names "\$HOME" and "\$EAGLEDIR" to access the user's home directory or the EAGLE installation directory, respectively.
- The new "Auto backup" feature will automatically save any modified drawing into a safety backup file after a certain time. See "Help/Automatic backup" for details.

* New Project Structure:

- The names of files that are under the current project directory are no longer written as absolute paths into the 'eagle.epf' file, but rather relative to the project directory. This allows for complete project directories to be easily copied or renamed.
- A project is now held in a subdirectory that contains a file named 'eagle.epf' (which stores the location and settings of open windows).

* Converting projects from previous versions:

- Previous versions of EAGLE used individually named project files (*.epf) to store project information. Beginning with version 4 a project is stored entirely in a subdirectory, and that directory contains a file with the fixed name 'eagle.epf'.
- The easiest way to convert projects from older versions is to create one directory for each project and copy that project's '.epf' file into this directory under the name 'eagle.epf'. The name of the project in version 4 is the name of the project's directory.
- In the Control Panel use the "Options/Directories" dialog to enter the name of the directory that contains your project subdirectories into the "Projects" field.

* User Interface:

- The textual command menu can now be configured to display aliased command buttons as well as submenus (see HELP MENU for details).
- Changes made in the "Options/User interface" dialog now take effect immediately for open editor windows.
- The cursor inside a layout or schematic editor window can now be set

to a "large" crosshair cursor (see "Options/User interface").

- The "Delete" icon was changed from a pencil with an eraser to an 'X'.
- The "Split" icon was changed to better indicate what will happen.

* Keyboard and mouse control:

- Alt-0 no longer pops up the window list, but leads directly to the Control Panel.
- Pressing the Ctrl key while moving the mouse now scrolls the draw window in any direction.
- The mouse wheel now zooms in and out in editor windows (zoom factor can be adjusted in "Options/User interface/Mouse wheel zoom", a value of '0' disables this feature and the sign of this factor defines the direction of the zoom operation).

* Screen display:

- The default for "minimum visible text size" has been changed to 3.
- The display mode parameter FAST has been dropped.
- By default the zoom factor in editor windows is limited so that the resulting virtual drawing area does not exceed the 16-bit coordinate range. This is necessary to avoid problems with graphics drivers that are not 32-bit proof. If the graphics driver on a particular system can handle coordinates that exceed the 16-bit range, "Options/User interface/Limit zoom factor" can be switched off allow larger zoom factors.

* Design Rules:

- EAGLE now supports a full set of Design Rules that are stored inside the board file (and can also be saved to disk files). Both the Design Rule Check and the Autorouter will use the same set of rules.
- Newly created boards take their design rules from the file 'default.dru', which is searched for in the first directory listed in the "Options/Directories/Design rules" path.
- Cream mask values are now measured "inwards" and thus have a positive sign.
- The parameters AnnulusConduct and ThermalConduct are no longer available. There are now checkboxes in the Design Rules dialog's "Supply" tab that define whether a Thermal or Annulus symbol shall have a "Restrict" or not.
- In order to assure that existing boards yield the same results when producing CAM data after they have been updated to version 4, the minimum restrict parameters for the outer layers are set to 0 in the design rules (this allows the existing pads and vias to keep their defined diameters). The user should adjust these parameters to some reasonable values and run a design rule check after adding new parts from version 4 libraries.

NOTE: The above has been changed in version 4.03 - see release notes for version 4.03!

Also, the new design rule parameter that controls the minimum distance between objects in signal layers and the board dimensions (default: 40mil) will be set to 0 if a board that is updated from an older version contains any signal polygons. The reason for this is that in previous versions polygons didn't take the board dimensions into account when they were calculated, but starting with version 4 polygons keep the minimum distance defined in "Edit/Design rules/Distance/Copper/Dimensions" from the boards dimensions. In order to guarantee that updated boards will yield the same results when producing CAM data with version 4 this parameter is set to 0.

Note that this will also have an impact on the autorouter, so the user should adjust this parameter to some reasonable value.

* Net Classes:

- Nets and Signals now have a new parameter called "Net Class".
- The new command CLASS is used to define and select net classes.
- The CHANGE command has a new option 'class' to change the net class of a net or signal.

* Polygons:

- When calculating polygons, the minimum distances defined in the design rules and net classes will be taken into account. Therefore the default value for the Isolate parameter of newly created polygons is now 0. If a particular polygon is given an Isolate value that exceeds that from the design rules and net classes, the larger value will be taken.
- The new parameter 'rank' defines if and how polygons are subtracted from each other. When updating existing files, polygons in signals will get a rank of '1', while polygons in packages will get rank '7'.
- Polygons are now checked in the Design Rule Check if they have the same 'rank'.
- Sometimes the polygon subtractor didn't go through a gap where, according to the actual widths and minimum distances, it should have.
- Polygons in the t/bRestrict layers are now subtracted from signal polygons in the Top and bottom layer, respectively.

* Design Rule Check:

- The DRC now runs a lot faster.
- Progress is now displayed in a progress bar (the progress rectangles are no longer displayed and the SET variables DRC_SHOW and DRC_COLOR are now obsolete).
- Since the DRC is now much faster its error messages are no longer stored in a separate '*.drc' file (this separate file sometimes caused board and DRC error messages to be out of sync).
- Polygons from different signals with the same 'rank' are checked against each other.
- All objects in layers Top..Bottom (including arcs, circles etc.) are now checked.
- The 'overlap' and 'minimum distance' check are no longer separate checks.
- The DRC no longer checks an individual signal against everything else. The newly introduced "Net Classes" can be used to do this.
- The rectangle for a selective DRC can now be defined with "click&drag" (just as in the WINDOW command).
- Holes are no longer checked in the "Grid" check (only pads, vias, smds and wires in signal layers are checked).
- Any objects in signal layers within a package are now checked against each other.
- Several new checks have been added (see the DRC dialog for more information about the new parameters).
- Due to a calculation problem the DRC sometimes reported very small errors where in reality there were no errors.

* Long strings:

- All names, values and texts can now be of any length.

- The User Language constants regarding name lengths still exist, but the program uses these constants only for formatted output as in the EXPORT command. They are still present for compatibility only.
- There is no more limit to the number of members in a bus (bus index values are limited to 0..511).
- Bus member names can now contain any characters, except ':', ',', '[', ']' and blanks.

* Wire styles:

- Wires now have a new parameter 'Style', which can be set to one of the following values:

Continuous	_____	(default)
LongDash	___ _ _ _	
ShortDash	-----	
DashDot	___ . ___ . ___	

- The variable for setting the bend type of a wire has been renamed from Wires_Style to Wire_Bend to avoid confusing the two parameters.
- Note that the DRC and Autorouter will always treat wires as "Continuous", even if their style is different. Wire styles are mainly for electrical and mechanical drawings and should not be used on copper layers. It is an explicit DRC error to use a non-continuous wire as part of a signal that is connected to any pad.

* Text fonts:

- Texts can now have three different fonts:
 - 'Vector' the program's internal vector font (as used in previous versions)
 - 'Proportional' a proportional pixel font (usually 'Helvetica')
 - 'Fixed' a monospaced pixel font (usually 'Courier')
- When updating drawings from older versions, all texts are converted to 'Proportional' font, except for those in layers Top...Bottom, tRestrict and bRestrict, since these texts probably need to be subtracted from signal polygons, which only works with the 'Vector' font.
- The program makes great efforts to output texts with fonts other than 'Vector' as good as possible. However, since the actual font is drawn by the system's graphics interface, 'Proportional' and 'Fixed' fonts may be output with different sizes and/or lengths. See HELP TEXT if you don't want texts to be displayed with fonts other than 'Vector'. You can set the user interface option "Always vector font" to always have texts output with the builtin vector font.
- When creating output files with the CAM Processor, texts with fonts other than 'Vector' may be output using the 'Vector' font instead. This happens if the actual output device is unable to produce texts with different fonts.
- If a text with a font other than 'Vector' is subtracted from a signal polygon, only the surrounding rectangle is subtracted. Due to the above mentioned possible size/length problems, the actually printed font may exceed that rectangle. Therefore, if you need to subtract a text from a signal polygon it is recommended that you use the 'Vector' font.
- The 'Ratio' parameter has no meaning for texts with fonts other than 'Vector'.
- The CHANGE command has a new option "Font".

* Pads and Vias:

- The diameter of pads and vias is now derived from the drill diameter using the Design Rules (the pad and via diameter '0' is now allowed and results in a diameter that is derived from the current design rules). If a pad is defined with a diameter that exceeds the one that would result from the current design rules, the larger diameter is taken. The default value for the diameter of newly created pads and vias is now '0' to allow the Design Rules to define the actual diameters.
- Pads can have different shapes on Top and Bottom (they will always be 'round' on the inner layers).
- The via shape now only applies to the outer layers (they will always be 'round' on the inner layers).
- The diameter of pads with shape X/YLongOct now defines the smaller side of the pad (formerly the wider side). Existing files will be modified accordingly during the update.
- By default vias no longer generate Thermal symbols in supply layers. There is a new design rules parameter that enables Thermal symbols to be generated for vias in supply layers and signal polygons.
- When updating files from older versions, pads and vias with diameter \leq drill will be replaced with a 'hole' of that drill diameter. This only works for unconnected pads; if a pad is connected to a pin (in a library or schematic) or to a signal (in a board) it can't be converted to a hole and the user must decide what to do in such a case. This conversion has become necessary because pads and vias now always have a 'restring' that is determined by the design rules.
If a consistent board/schematic pair is updated to version 4 and such pads are replaced with holes in only one of the drawings, the board/schematic pair will become inconsistent. If that happens you will need to modify the respective package/device definition to make things consistent again.

* Round SMDs:

- SMDs have a new parameter "Roundness", which can range between 0 and 100 and defines the percentage by which the corners are "rounded". A value of 0 (default) results in a rectangle, while a value of 100 results in a circular shape (if the x and y dimension of the SMD are the same), which can be used for BGAs.
- The SMD command accepts roundness values as numbers with a leading '!' (to be able to distinguish it from the SMD size values).
- The CHANGE command has a new option "Roundness".

* New Library structure:

- What was called a "Device" in previous versions is now called a "Device Set". A "Device Set" consists of the gate definitions and several actual devices, implemented through "Package Variants"
- The PACKAGE command can now assign several different package variants to a device (as in 7400N, 7400D,...).
- The new command TECHNOLOGY can be used to define various "technology" variants for a device's package variants (as in 74LS00N, 74S00N,...).
- The CHANGE command has the new options PACKAGE and TECHNOLOGY, which can be used to select from the packages and/or technologies a device set defines. This can be done from within the schematic or board.
- The new command DESCRIPTION can be used to provide detailed textual information about a device, package or library.

- The CONNECT dialog now allows copying pin/pad connections from an other package variant. Only those package variants are offered in the "Copy from" combo box that have the same pad names as the current package variant (only connected pads are checked).
- The CONNECT dialog now asks the user if he want's to discard any changes before canceling the dialog.
- The CONNECT command can now handle gate names that contain periods.
- The device editor now displays a list of package variants, a preview of the current package and the description of the device.
- Since it is now possible to "completely" define a device with all package and technology variants, the default setting of the "Value" parameter has been changed to "Off".
- The meaning of the "Value" parameter in a device set is now as follows:
 - + "Value Off" means there is no user definable value, i.e. the value of a part is defined by the dievice name (including, if present, technology and package variant). A device like "74LS00N" would be an example.
 - + "Value On" means this device needs a user defined value to be fully specified. A resistor is an example.
 Even in the "value off" case the user can (after a confirmation dialog) change a part's value to handle any special cases that might otherwise cause problems. Any CHANGE TECHNOLOGY or CHANGE PACKAGE command that is executed after such a change will set the value back to the device name.

* Automatic Library update:

- If a library has been modified after parts or packages from it have been added to a schematic or board, the new command UPDATE can be used to automatically update all used library objects with their latest version (see "Help Update").
- The UPDATE command can be selected from the "Library" pulldown menu in a board or schematic, or from the context menu of a library in the Control Panel. It is also possible to drag&drop a library from the Control Panel onto a schematic or board drawing and perform the update that way.

* Bill Of Material:

- The User Language Program 'bom.ulp' to generate the "Bill Of Material" has been rewritten. It now has a dialog in which the user can interactively generate the BOM, pulling in additional data from a user defined database file. Use "RUN bom.ulp" and click on the "Help" button for more information.

* Generating Outlines for milling prototypes:

- The User Language Program 'outlines.ulp' can be used to generate the data necessary to control a milling machine for generating a prototype board.

* User Language:

- The User Language now supports user defined dialogs as well as standard file dialogs and message boxes.
- The RUN command now accepts additional arguments that are available to the ULP as 'argc' and 'argv' parameters.
- Data can now be read into a ULP.
- The new lookup() function can be used to perform database lookups.
- The new fileglob() function can be used to do a directory search.
- The new fileerror() function can be used check for I/O errors.

- The 'exit()' function can now have a string parameter which is sent to the editor window and executed as a command string.
- ULPs can now include other ULP files with the new #include directive.
- The new #usage directive can be used to provide information about a ULP.
- The new object UL_DEVICESET is used to access device sets in a library.
- The builtin statement device() has been renamed to deviceset() to conform with the new library structure.
- UL_POLYGON has a new member 'rank'.
- UL_POLYGON has new members 'contours()' and 'fillings()' to access the calculated polygon data. The 'dxf.ulp' now uses these new members to draw the actual shape of calculated polygons.
- The new object UL_CLASS is used to access net classes.
- UL_BOARD and UL_SCHEMATIC have new members 'classes()'.
- UL_NET and UL_SIGNAL have new members 'class'.
- UL_WIRE has new members 'style' and 'pieces()'.
- UL_TEXT has a new member 'font'.
- The data members 'diameter' and 'shape' of UL_PAD and UL_VIA are now layer dependent and thus require the layer number for which the data shall be retrieved, because depending on the new Design Rules diameters and shapes of pads and vias can be different in the various layers. The syntax is now, e.g., pad.diameter[LAYER_TOP] to get the pad diameter in layer 1. Existing User Language Programs need to be edited to conform to this new syntax. See "Help UL_PAD/UL_VIA" for further details. The 'dxf.ulp' now generates the pads and vias separately in each active layer, including stopmask layers.
- The data members UL_SMD.dx and UL_SMD.dy can now have an optional layer index to retrieve the dimensions in the t/bStop and t/bCream mask layers. The 'dxf.ulp' now generates mask data if those layers are active.
- New builtin constants INT_MAX, INT_MIN, REAL_EPSILON, REAL_MAX and REAL_MIN (see "Help/User Language/Builtins/Builtin Constants").

* Script files:

- Script files can now call other scripts (as long as no recursive call is made).
- Script files can now contain comments. Everything after (and uncluding) a '#' character will be ignored. '#' characters inside quotes have no special meaning. Note that script labels (e.g. "BRD:") and continuation characters ("\") must not be followed by anything else than white space, and therefore can not be followed by comments (otherwise they loose their special meaning).
- The 'eagle.scr' file is now first searched for in the current project directory (which is equal to the current working directory in case there is no project open) and then in the directories listed in the Control Panel's "Options/Directories/Scripts".

* Autorouter:

- The Autorouter can now route "through" signal polygons (this can be controlled by the new cost factor 'cfPolygon'). A side effect of this is that a connection that has been made by the polygon before the Autorouter was started may be lost if a track generated by the Autorouter "splits" the polygon. Therefore, even if the Autorouter reaches 100%, the final result (after recalculating the polygon by RATSNEST) may be below 100%.
- The Autorouter control parameters are now stored inside the board file. They can be saved to and loaded from external files via the Autorouter

dialog. Existing control files will be automatically read and stored in the board file when updating files from previous versions (the old *.ctl file then becomes obsolete and will not be read when the autorouter is started - any changes made to such a file with a text editor are ignored!).

- The Autorouter and DRC now use the same set of Design Rules. The old Autorouter parameters are stored in their corresponding DRC parameters, except for the following:
 - + Instead of the separate 'mdWireDimension' and 'mdViaDimension' there is now only a single 'Copper/Dimension' parameter. This parameter will be set to the last of the two 'md*' parameters from the *.ctl file (which typically is 'mdViaDimension').
 - + The 'mdWireRestrict' and 'mdViaRestrict' parameters are ignored because the minimum distance between any copper (except pads and smds) and a restrict area is now 0.
 - + The 'tpViaDiameter' parameter is ignored because the actual via diameter is now generated from the via drill and the restring parameters.
 - + The 'tpWireWidth' and 'tpViaDrill' parameters are stored in the default net class 0.

When saving Autorouter control parameters to disk, the minimum distance parameters are no longer part of that file.

- There can now be any number of 'Optimize' passes. By default there are now 4 'Optimize' passes.
- Each pass can be separately activated or deactivated.
- The Autorouter can now route different wire widths and minimum distances simultaneously by using "Net Classes".
- The minimum distance parameters are no longer defined in the Autorouter dialog, but rather in the Design Rules dialog.
- The track parameters (wire width and via diameter) are now defined in the Design Rules dialog (absolute minimums) and the Net Classes dialog.
- The minimum routing grid is now 0.02mm (about 0.8mil).
- The default control parameters and the internal handling of cfChangeDir have been modified to avoid jagged tracks.
- The value 99 for the cfNonPref parameter now causes the router to completely avoid traces that are not in the preferred direction of the respective layer. You should carefully decide if this behaviour is really what you want. To avoid unexpected effects with existing boards, the value 99 will be silently changed to 98 when updating older files.
- With very small routing grids the Autorouter sometimes routed a little bit too close towards round pads/vias.
- Sometimes the autorouter didn't go through a gap where, according to the actual widths and minimum distances, it should have.

* ADD command:

- The ADD command can now be used with wildcards ('*' or '?') to find a specific device. The ADD dialog offers a tree view of the matching devices, as well as a preview of the device and package variant.
- To add directly from a specific library, the command syntax

ADD devicename@libraryname

can be used. 'devicename' may contain wildcards and 'libraryname' can be either a plain library name (like "ttl" or "ttl.lbr") or a full file name (like "/home/mydir/myproject/ttl.lbr" or "../lbr/ttl").

- If a device or package shall be added, and there is already such an object

(with the same name from the same library) in the drawing, an automatic library update will be performed which replaces the existing object in the drawing with the current version from the library.

- The new command UPDATE can be used to update all parts in a board or schematic with modified library versions (see "Help Update").

* CHANGE command:

- CHANGE LAYER for wires and polygons now works between any layers within packages and symbols.

* CONNECT command:

- Pressing the SPACE key in the CONNECT dialog while a list element has the focus will now perform the 'connect' or 'disconnect' action, respectively.

* DELETE command:

- If the last supply symbol of a given type is deleted from a net segment that has the same name as the deleted supply pin, that segment is now given a newly generated name (if there are no other supply symbols still attached to that segment) or the name of one of the remaining supply symbols.

* DISPLAY command:

- The new parameters '?' and '??' can be used to control what happens if a layer that is given in a DISPLAY command does not exist in the current drawing. See "Help Display" for details.

* GROUP command:

- If the selected group is empty, the GROUP command no longer displays a message box saying "Group is empty". It rather prompts that message in the status bar (with a beep) and stays active for a new group definition.

* ERC command:

- The ERC now lists the package names when reporting parts/elements with inconsistent packages.
- The ERC now detects inconsistencies between the implicit power and supply pins in the schematic and the actual signal connections in the board. Such inconsistencies can occur if the supply pin configuration is modified after the board has been created with the BOARD command. Since the power pins are only connected "implicitly", these changes can't always be forward annotated. If such errors are detected, forward-/backannotation will still be performed, but the supply pin configuration should be checked!
- The ERC now checks for missing junctions and overlapping wires and pins.

* ERRORS command:

- The ERRORS dialog is no longer modal (it stays "on top" of the editor window) and can be kept open while resuming normal editing in the editor window.
- The various error types are now listed more detailed.

* EXPORT command:

- The EXPORT can now export image files (BMP, PNG, etc.). See "Help/EXPORT" for details.

* NET and BUS command:

- If a net wire is placed at a point where there is already another net or bus wire or a pin, the current net wire will be ended at that point (in previous versions the user had to click twice to end a net wire). The same applies to a bus wire that is placed at a point where there is already another bus wire. This function can be disabled with "SET AUTO_END_NET OFF;", or by unchecking "Options/Set/Misc/Auto end net and bus".
- If a net wire is placed at a point where there are at least two other net wires and/or pins, a junction will automatically be placed. This function can be disabled with "SET AUTO_JUNCTION OFF;", or by unchecking "Options/Set/Misc/Auto set junction".
- If a bus name is used with a range, that name must not end with digits, because it would become unclear which digits belong to the Name and which belong to the range.

* PASTE command:

- When pasting objects into a drawing that already contains earlier (different) versions of these objects, an automatic library update will be performed which replaces the existing objects in the drawing with the new versions from the paste buffer.

* PRINT command:

- The PRINT dialog's "Page setup" now allows border values that are smaller than the initial values derived from the printer driver. To get back to the original default you can enter '0'. Note, though, that your actual printer may not be able to print that close to the page limits.
- The printer settings are no longer stored in the project file, but are now stored in the user parameters ('eaglerc').
- Version 3.5 stored the binary printer setup in the Windows registry in order to remember which printer was last used in EAGLE. This, however, has frequently caused problems, sometimes even total system crashes. As of version 4 EAGLE (like most other Windows applications) no longer stores the used printer and always starts up with the system default printer.

* REMOVE command:

- The REMOVE command can now handle device, symbol and package names with extension (for example REMOVE name.pac). If the name is given without extension, you have to be in the respective mode to remove an object (i.e. editing a package if you want to remove packages).

* RENAME command:

- The RENAME command now allows '.' in names.
- The RENAME command can now handle device, symbol and package names with extension (for example RENAME name1.pac name2[.pac] - note that the

extension is optional in the second parameter). If the first parameter is given without extension, you have to be in the respective mode to rename an object (i.e. editing a package if you want to rename packages).

* REPLACE command:

- The REPLACE command can no longer be used with active forward- and backannotation. This is due to the now complete definition of a device set with all its package variants. Use the CHANGE PACKAGE command to select one of the defined package variants, or use the UPDATE command to update a package with a modified version from the same library.

* SET command:

- The SET options for Thermal and Annulus parameters as well as the Solder Stop and Cream mask data have been removed. These values are now defined in the Design Rules.
- The SET variables DRC_SHOW and DRC_COLOR are now obsolete (progress in the Design Rule Check is now displayed in a progress bar).
- The SET variable MAX_ERROR_ZOOM is now obsolete. The ERRORS dialog is no longer modal (it stays "on top" of the editor window) and zooming can be done with the usual WINDOW commands or buttons.

* SHOW command:

- Highlighted objects are now kept highlighted during subsequent window operations.
- Pressing ESCape in the SHOW command now lowlights the currently highlighted object.

* USE command:

- The USE command is now mainly for use in script files.
- The actually used libraries can now be comfortably selected in the Control Panel.

* CAM Processor:

- The CAM Processor no longer supports matrix printers. Use the PRINT command to print to the system printer.
- The CAM Processor no longer prints schematics. Use the PRINT command instead. (As of version 4.03 the CAM Processor can print schematics again).
- The command line option -I is no longer available, since the CAM Processor can now be used freely in the Freeware version.
- Output is now only possible into files. If data shall be sent to a COM or LPT port under Windows the UNC filename of a queue attached to that port has to be used.
- If the "Section" text in a CAM Processor section consists of a string like "Title: Descriptive text...", the "Title" will appear on the section's tab, while the "Descriptive text..." will only be visible in the "Section" field.
- CAM Processor jobs can now have a description.
- The diameter of octagonal pads in RS274-X has been increased by a factor of 1.08239 to compensate for the different interpretation of pad diameters in EAGLE and RS274-X.
- Wires are no longer shortened in the CAM Processor in order to keep the drills open. Only devices that can actually remove pixels from the output

can now keep the drills open (currently only the Postscript devices "PS" and "EPS" can do this).

- The new CAM Processor device PS_INVERTED can be used to produce inverted Postscript output.

* Parameter storage:

- User specific parameters are now stored in an "eaglerc" file.
At program start, parameters are read (in the given sequence) from the files

<prgdir>/eaglerc	(Linux and Windows)
/etc/eaglerc	(Linux only)
\$HOME/.eaglerc	(Linux)
\$HOME/eaglerc.usr	(Windows)

where <prgdir> means the directory that contains the EAGLE program file. If no environment variable HOME is defined, the <prgdir> will be used instead. When the program ends, the current values of all parameters (if any of them have changed) are written to the eaglerc file in the HOME directory.

Under Linux "\$HOME" stands for the environment variable HOME.

Under Windows "\$HOME" is either the environment variable HOME (if set) or the value of the registry key "HKEY_CURRENT_USER\Software\Microsoft\Windows\CurrentVersion\Explorer\Shell Folders\Personal", which contains the actual name of the "My Documents" directory.

- The file 'eagle.cfg' is not read any more.
- Key assignments made with the ASSIGN command are now stored in the user specific parameters.

* Command line options:

- The options '-A' and '-T' are now obsolete (thermal and annulus data is now defined in the Design Rules).
- The options '-B' and '-M' are now obsolete (solder stop and cream mask data is now defined in the Design Rules).
- The option '-C' is now obsolete, since the CAM Processor no longer supports matrix printers (all printing is done with the PRINT command).
- The options '-Z' and '-Y' are now obsolete (drill symbols are configured in "Options/Set/Drill" and are stored in the user specific "eaglerc" file).

* Bugfixes:

- When printing several sheets at once, the >SHEET always displayed the number of the currently edited sheet.
- When saving a file with more than one dot in its name the name was cut off.

* Miscellaneous:

- The DOS and OS/2 platforms is no longer supported.
- Due to changes in the file data structure you will most likely be asked whether to run the ERC when loading a board/schematic pair created with an earlier version of EAGLE.
- Files from earlier versions of EAGLE may contain library objects with the same names. This was caused by PASTE or ADD operations with modified

devices or packages. Version 4 no longer allows this to happen, and therefore needs to make sure updated files do not contain multiple objects with the same name. In order to assure this, the update routine adds the '@' character and a number to the names of such library objects.

- The library editor can now edit devices and symbols, even if the user's license does not contain the schematic module.
- Avoiding multiple 'Save?' prompts for boards and schematics that are connected via f/b annotation.
- When a file is modified while updating from a previous version the resulting update report is now presented in a separate text window.