CadSoft unveils version 6.0 of the EAGLE PCB design tool

Premier Farnell today announced release plans for EAGLE Version 6.0 including details of the many new features being developed for this release and deliver a major step forward in the PCB designer productivity and efficiency. All users purchasing the current version of EAGLE will receive a free upgrade to the version 6.0 release.

CadSoft's EAGLE V6.0 PCB design software will deliver innovations and improvements over previous versions that will allow design engineers significant flexibility and efficiency. The EAGLE V6.0 release features a redesigned XML database structure – the first to use ASCII text as format, which allows quick and easy design changes, importing of design information as well as translation from other design formats.

"The best-selling CAD tool (EAGLE) for PCB design just got better, and we will deliver the latest EAGLE V6 free to new and upgrading customers after release," said Thomas Liratsch, CadSoft's General Manager. "EAGLE V6.0 is agile and intuitive, so design engineers can easily design, replicate, move, transpose, and even copy/paste schematics – saving them time and effort. Simply put, the new release of EAGLE takes designer friendliness and freedom to a whole new experience."

EAGLE V6.0 includes a variety of user-friendly and productivity-enhancing features, such as:
• XML database structure – Read, edit and parse the data outside of the EAGLE tool
• Design reuse – Ability to merge board/schematic pairs using COPY/ PASTE function with full consistency. This supports easy design, for example, of multi-channel devices.
• Routing capability improvements
  · Optimised work-flow with an Undo/Redo log to control each step going back or forward
  · Flawless conversions between millimetres and inches with increased internal resolution
• Library editor optimisations
  · Ability to connect a pin to multiple pads allows designers to handle components' internal connected pins. Compared to previous versions schematic symbols look nicer and you save time in defining them.
  · Designers can also draw arbitrary pad shapes easily without any problems with Design Rules Check for such objects.
  · EAGLE V6.0 also supports an unlimited number of technologies and package variants – eliminating the need to split a library into two parts.
  · File description functionality – You can now individualise your schematics, sheets, boards and symbols with descriptions, so you don't have to open the file in

an editor window to see its contents.

Customers who buy or upgrade from a former version to EAGLE V5 from 01 September 2011 can upgrade their software edition and modules for free to EAGLE Version 6.0. For more information please visit www.cadsoft.de/conditions_buy_now [1]

Source URL (retrieved on *12/28/2014 - 1:11am*):
http://www.ecnmag.com/product-releases/2011/10/cadsoft-unveils-version-60-eagle-pcb-design-tool

Links:
[1] http://www.cadsoft.de/conditions_buy_now