CADSoft Eagle PCB Layout Editor

CADSoft Eagle is a Windows application for doing schematics and PCB layouts. It is widely used in the DIY community due to a freely available version which allows you to design small boards with certain limitations.

Installation

Usage

There are some great usage tips on the Eagle forum thread, as well as elsewhere on the forum. If you find some useful information please copy it into this document.

Libraries

- SmashTV Encoder with Switch

Print an image suitable for toner-transfer resist

Open the .brd file in Eagle

Under the “view” menu choose “Display/Hide Layers...”

Click the layer numbers on the left. Find the layers you want to print. Click “OK”.

Click in the command line (above the image area“ and enter the command RATSNEST to show the ground plane.

Under the file menu, choose “Export”, then choose “Picture”.

Select an appropriate DPI (300+), click “monochrome”, click “browse” to name your image file.

Eagle will generate a .png file suitable for printing from an image editor.

Maybe there's an easier way?

External Links

http://www.cadsoft.de

http://www.instructables.com/id/How-to-make-a-custom-library-part-in-Eagle-CAD-too/

http://www.gearslutz.com/board/geekslutz-forum/147041-eagle-cad-libraries.html

From:
https://wiki.midibox.org/ - **MIDIbox**

Permanent link:
https://wiki.midibox.org/doku.php?id=cadsoft_eagle&rev=1224569194

Last update: **2008/11/25 18:25**