

Eagle PCB Design Software



Summary

1. Introduction

Eagle is a proprietary Computer Assisted Drawing (CAD) software for designing Printed Circuit Boards (PCBs) from Schematics to Gerber Files. This is the software that have primarily been used to design the MBHP PCB modules. It has limitations on PCB size on the freeware version.

You can find the official website here : <http://www.cadsoftusa.com/>



2. Installation

First, download the correct installer for your operating system at

<http://www.cadsoftusa.com/download-eagle/>

Follow the instructions to get the program installed

3. Getting Started Video Tutorials

A serie of official Getting Started videos to be found here:

<http://www.cadsoftusa.com/eagle-pcb-design-software/guided-tour/#c760>

MIDIbox Library

[MIDIbox Eagle Library](#)

Tips & Tricks

[HowTo Custom Library](#)

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.

From:

<http://www.midibox.org/dokuwiki/> - **MIDIbox**

Permanent link:

<http://www.midibox.org/dokuwiki/doku.php?id=eagle>

Last update: **2016/06/14 00:54**

