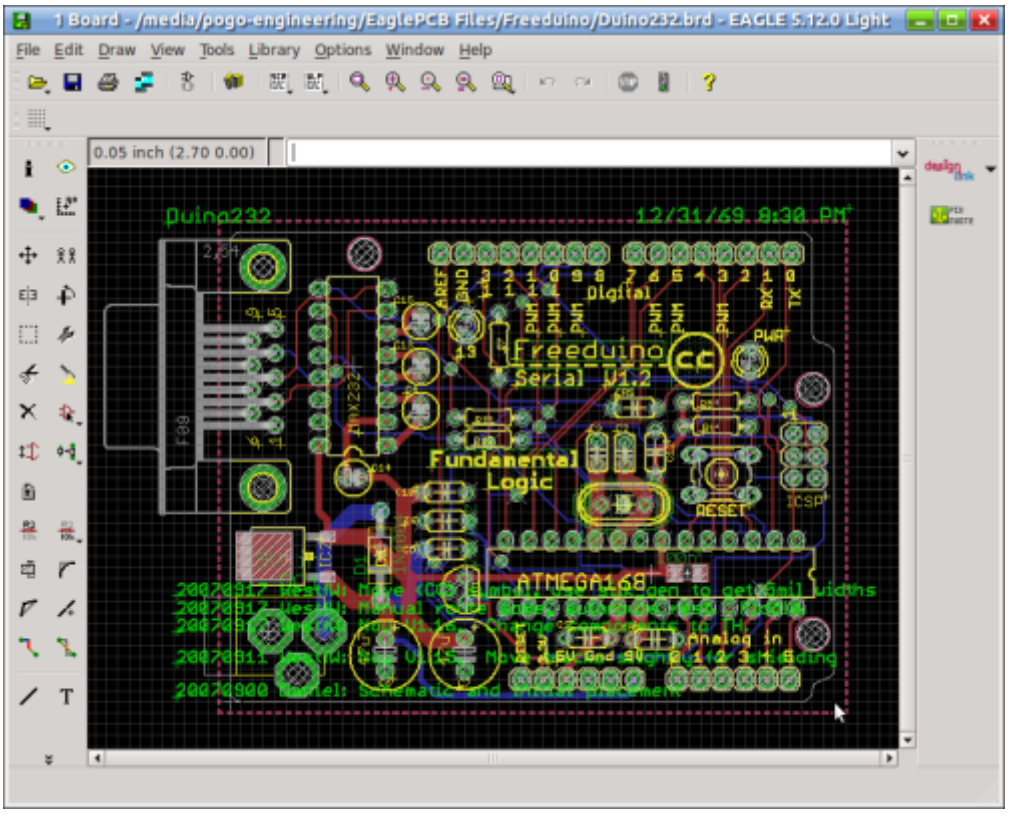
Eagle PCB Design Software

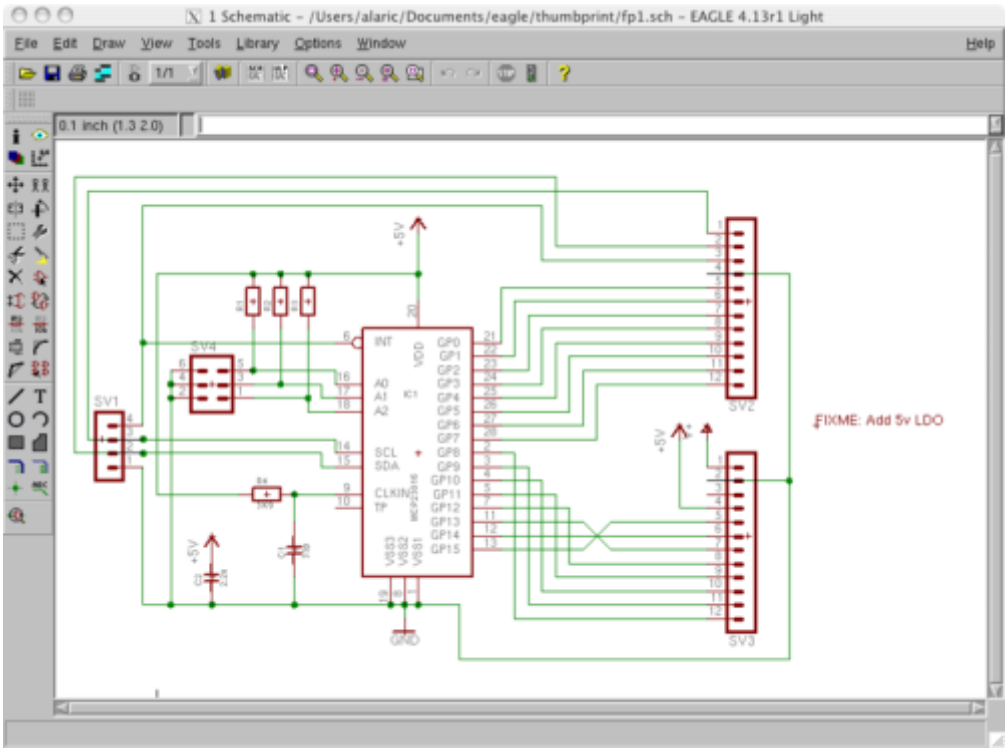


Summary

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

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

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

Eagles External Libs

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

Permanent link:
https://wiki.midibox.org/doku.php?id=eagle&rev=1465132088

Last update: **2016/06/05 13:08**