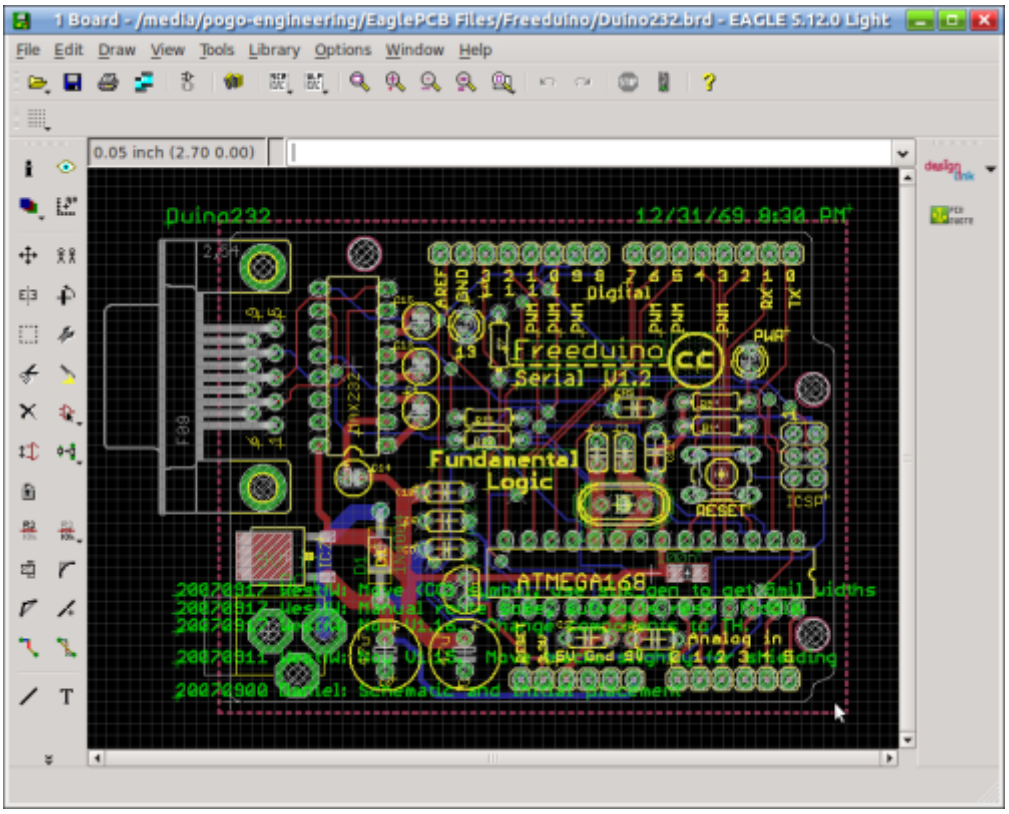
Eagle for PCB Design

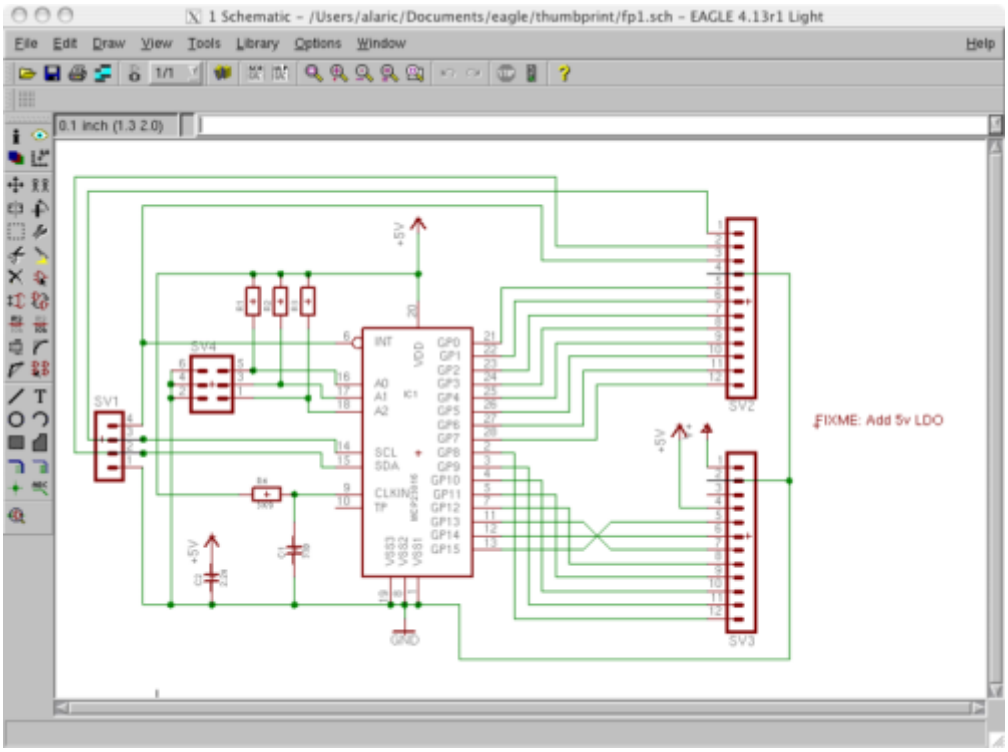


Summary

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

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

Permanent link:
https://wiki.midibox.org/doku.php?id=eagle&rev=1465130513

Last update: **2016/06/05 12:41**