PCB Design Guide

Limbo | Hand I/O

11/16/2021

Table of Contents

[1.0 Overview 2](#_Toc87957534)

[1.0 Recommended EAGLE Tutorials 2](#_Toc87957535)

[2.0 Installation/Environment Set-Up 3](#_Toc87957536)

[2.1 Installing EAGLE 3](#_Toc87957537)

[2.2 Using GTOR’s EAGLE Libraries 3](#_Toc87957538)

[2.3 Setting up Design Rules 3](#_Toc87957539)

[2.3.1 JLCPCB Design Rules 3](#_Toc87957540)

[2.3.2 ME Electronics Design Rules 3](#_Toc87957541)

[3.0 Recommended Design Guidelines 4](#_Toc87957542)

[3.1 Trace Width and Spacing 4](#_Toc87957543)

[3.2 Ground/Power Planes 4](#_Toc87957544)

[3.3 Mounting Holes 4](#_Toc87957545)

[3.4 Using Vias 4](#_Toc87957546)

[3.5 Considerations for PCB Debugging 4](#_Toc87957547)

[3.5.1 Extra headers 4](#_Toc87957548)

[3.5.2 Testing points 4](#_Toc87957549)

[3.5.3 LED indication 4](#_Toc87957550)

[3.5.4 Well designed silk screen 4](#_Toc87957551)

[3.6 Considerations for Non-Obvious Design Errors 5](#_Toc87957552)

[3.6.1 3D Part geometry interference 5](#_Toc87957553)

[3.6.2 Connectors and wires that may be in the way 5](#_Toc87957554)

[3.6.3 Part orientations 5](#_Toc87957555)

[3.6.4 Breaks in ground plane 5](#_Toc87957556)

[4.0 Adding Limbo Eagle Libraries to Eagle Path 6](#_Toc87957557)

[4.1 Introduction 6](#_Toc87957558)

[2.2 Adding an Eagle library file to Eagle library directory path 6](#_Toc87957559)

[5.0 Revision History 9](#_Toc87957560)

# 1.0 Overview

## 1.0 Recommended EAGLE Tutorials

If you prefer reading and working through a tutorial, SparkFun has a really good series of two tutorials that walk you through a board design and give some helpful tips and tricks along the way.

* Schematic Design - <https://learn.sparkfun.com/tutorials/using-eagle-schematic/all>
* Board Layout - <https://learn.sparkfun.com/tutorials/using-eagle-board-layout>

And if you prefer watching videos here is a YouTube series that will be helpful.

* Schematic Design - <https://www.youtube.com/watch?v=GGBcdoFhdWs>
* Board Layout - <https://www.youtube.com/watch?v=a1l6N7BVINA>
* Finalizing Design - <https://www.youtube.com/watch?v=Eu5XMEh79XM>

# 2.0 Installation/Environment Set-Up

## 2.1 Installing EAGLE

To install Eagle, first get educational access to Autodesk products [here](https://www.autodesk.com/education/edu-software/overview?_ga=2.132789582.1533660205.1618200898-1990998181.1614128526&sorting=featured&page=1). Then, download Eagle [here](https://www.autodesk.com/products/eagle/free-download?plc=F360&term=1-YEAR&support=ADVANCED&quantity=1) and log in using the same credentials that were used to get educational access to Autodesk products.

## 2.2 Setting up Design Rules

Setting up the design rules is an important first step when designing a PCB. It ensures that your design will be manufacturable in the end so that you don’t have to redesign parts of the PCB that can’t be manufactured.

The design rules can be changed by going to Edit>Design Rules from within the board editor. The two tabs that you should care most about are “Clearance” and “Sizes”.

### 2.2.1 JLCPCB Design Rules

While JLCPCB can manufacture with tighter tolerances, I generally keep the design rules at EAGLE’s default since there is usually no reason to design with tighter tolerances and smaller traces anyways.

# 3.0 Recommended Design Guidelines

## 3.1 Trace Width and Spacing

When deciding on the width for your traces, the main factor to consider is the amount of current the trace will be carrying. A good rule of thumb is that anything less than 100mA of current will be fine on 6 mil traces. Anything less than 1A of current will be fine on 12 mil traces. For everything outside of these ranges, or if tighter tolerances are needed, a proper trace width calculator should be used. If the PCB is going to be manufactured with ME Electronics, 4 mil should be added on to the whatever trace width is needed due to manufacturing tolerances.

A good rule of thumb for the spacing between traces is just to use the trace with. So, 12 mil traces should have 12 mil spacing, etc. Again, if tighter tolerances are needed or something is outside of this range, just use a proper calculator. Many can be found online with relative ease.

## 3.2 Ground/Power Planes

A technique for designing higher quality circuits is to use ground and/or power planes when it makes sense to. They are good for reducing noise and EMI within your circuit. For most of our applications this isn’t very necessary, but since it is easy to do, we usually just add a ground plane to the back side of our PCB’s.

## 3.3 Mounting Holes

When designing a PCB you should also be thinking about how it is going to be enclosed/mounted. You will likely want some sort of mounting holes for your PCB.

## 3.4 Using Vias

## 3.5 Considerations for PCB Debugging

The recommendations in this section don’t do much to improve the design of the PCB, however they improve the debugging process of the PCB which can save a lot of time in the long run.

### 3.5.1 Extra headers

### 3.5.2 Testing points

### 3.5.3 LED indication

### 3.5.4 Well designed silk screen

## 3.6 Considerations for Non-Obvious Design Errors

The recommendations in this section aren’t necessarily rules of thumb for designing, rather they are things to think through when designing a PCB to ensure that when the PCB is being assembled and tested you can avoid a face-palm moment. Most of these recommendations come from actually making these mistakes before and hopefully they can be avoided in the future.

### 3.6.1 3D Part geometry interference

### 3.6.2 Connectors and wires that may be in the way

### 3.6.3 Part orientations

### 3.6.4 Breaks in ground plane

# 4.0 Adding Limbo Eagle Libraries to Eagle Path

## 4.1 Introduction

Eagle libraries contain information on each integrated circuit (IC) or component that is used on the PCB. Limbo has a specific Eagle library for its members located in the *LimboEagleLibraries* folder.

In order to have Eagle identify where the Eagle library file is, we have to explicitly tell Eagle where to search by adding the file path to Eagle’s directories.

When modifying the Limbo Eagle Libraries, make sure to pull from the Hand I/O repository before starting your work and pushing after adding a device to the library **immediately**. This way, we can avoid having to work with merge conflicts as much as possible.

## 2.2 Adding an Eagle library file to Eagle library directory path

For Eagle to recognize the file as a library, we need to change the directory for Eagle’s library in the settings.

On the control panel, navigate **Control Panel > Options > Directories**.

Graphical user interface, website

Description automatically generated

Then on the **Directories** window, click on the field next to libraries, and then click on **Browse…** to change the root directory of Eagle’s libraries.

Graphical user interface, application

Description automatically generated

Next navigate to your home directory for the folder containing the .lbr file for an Eagle library, and then click **Select Folder**. In this case, the GTOREagleLib folder contains the .lbr file.

**Graphical user interface, text, application

Description automatically generated**

Now, you can look at your new libraries in Eagle by returning to the **Control Panel** and looking under **Libraries**

Graphical user interface, text, application

Description automatically generated

To use the parts in your schematics, right click the folder and click “Use all” so that they will show up when you go to add parts.

Graphical user interface, application

Description automatically generated

# 5.0 Revision History

6/1/2021 (Andrew Hellrigel) – Created the first revision for this document

11/16/2021 (Ryan Chen) – Adapted guide to Limbo and added how to add Eagle library to path