PCB Design Guide

GT Off-Road Racing | Data Acquisition

Andrew Hellrigel

06/01/2021

Table of Contents

[1.0 Overview 2](#_Toc74933540)

[1.1 Introduction 2](#_Toc74933541)

[1.2 Manufacturing Methods 2](#_Toc74933542)

[1.2.1 JLCPCB 2](#_Toc74933543)

[1.2.2 ME Electronics 2](#_Toc74933544)

[1.2 Recommended EAGLE Tutorial 3](#_Toc74933545)

[2.0 Installation/Environment Set-Up 4](#_Toc74933546)

[2.1 Installing EAGLE 4](#_Toc74933547)

[2.2 Using GTOR’s EAGLE Libraries 4](#_Toc74933548)

[2.3 Setting up Design Rules 4](#_Toc74933549)

[2.3.1 JLCPCB Design Rules 4](#_Toc74933550)

[2.3.2 ME Electronics Design Rules 4](#_Toc74933551)

[3.0 Recommended Design Guidelines 5](#_Toc74933552)

[3.1 Trace Width and Spacing 5](#_Toc74933553)

[3.1.1 Designing for JLCPCB 5](#_Toc74933554)

[3.1.2 Designing for ME Electronics 5](#_Toc74933555)

[3.2 Ground/Power Planes 5](#_Toc74933556)

[3.3 Mounting Holes 5](#_Toc74933557)

[3.4 Using Vias 5](#_Toc74933558)

[3.5 Considerations for PCB Debugging 5](#_Toc74933559)

[3.5.1 Extra headers 5](#_Toc74933560)

[3.5.2 Testing points 5](#_Toc74933561)

[3.5.3 LED indication 5](#_Toc74933562)

[3.5.4 Well designed silk screen 5](#_Toc74933563)

[3.6 Considerations for Non-Obvious Design Errors 5](#_Toc74933564)

[3.6.1 3D Part geometry interference 5](#_Toc74933565)

[3.6.2 Connectors and wires that may be in the way 5](#_Toc74933566)

[3.6.3 Part orientations 6](#_Toc74933567)

[3.6.4 Breaks in ground plane 6](#_Toc74933568)

[5.0 Revision History 7](#_Toc74933569)

# 1.0 Overview

## 1.1 Introduction

The purpose of this document is to give an intro into using EAGLE for PCB design specific to GTOR. This will include how to install EAGLE software, and how use the team’s EAGLE libraries. It will also give some general design rules that should be used for common types of PCB’s we design (such as recommended trace widths, spacings, etc.) This guide will **not** teach how to use EAGLE, but I will give a recommended YouTube series that will give an intro into designing a basic board.

## 1.2 Manufacturing Methods

There are two main methods that GTOR uses for manufacturing our PCB’s. The first is by using JLCPCB which is a Chinese PCB manufacturer and the second is ME Electronics which is an in-house service at Georgia Tech that can manufacture simple PCB’s. I will discuss the pros and cons of each method.

### 1.2.1 JLCPCB

JLCPCB orders can be placed through their website: <https://jlcpcb.com/>. The order should be quoted first using the “quote now” feature and then it can be added to the purchasing spreadsheet. Since PCB orders need to be placed using specific configurations (such as PCB color, etc.) and with the specific PCB files, it is best if someone from DAQ actually places the order to avoid errors.

**Pros**

* Pretty cheap compared to other PCB manufacturers ($2-4 for 5 boards and lower prices in quantity)
* High quality 2- and 4-layer boards with small trace widths and complex designs

**Cons**

* Expensive shipping (~$10 for 3-week shipping and ~$20 for 1-week shipping)
* Shipping delay even for expedited shipping

### 1.2.2 ME Electronics

ME Electronics requests can be placed through their website or by emailing them directly. More information can be found on their website: <https://www.me.gatech.edu/facilities/electronic_lab>.

**Pros**

* Free
* Generally, can get 1–2-day turnaround on board designs

**Cons**

* Only very simple 1–2-layer boards (but should really only do 1-layer boards **without** plated through holes). Anything more complex should really be made from a professional PCB manufacturer. Careful consideration should be taken in the design of these boards as you can’t solder to the back of the board because there is no copper on the back.
* Requires relatively large trace widths and very simple designs (pretty much limits this method to only breakout type boards).

## 1.2 Recommended EAGLE Tutorial

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 Using GTOR’s EAGLE Libraries

To use and contribute to GTOR’s EAGLE libraries, refer to the Eagle\_Libraries\_Guide document to set it up. This document will show you how to both use parts from the GTOREagleLib and how to contribute to the GTOREagleLib if you need to use a part in your design that is not already a part of the library.

## 2.3 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”.

### 2.3.1 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.

### 2.3.2 ME Electronics Design Rules

ME Electronics can’t design within EAGLE’s default design rules. Set the minimum trace width and all of the minimum spacings to 10 mil and that should be pretty safe for them to manufacture, although if you can go bigger, you should. Based on the way that they mill the traces, I’ve noticed that sometimes the traces can be quite a bit smaller than originally intended. A good rule of thumb that I use is to add 4 mil to the trace width that I need. So, if I normally design with 6 mil traces for signal and 12 mil traces for power, then do 10 mil traces for signal and 16 mil traces for power.

# 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 Other EAGLE Tips

## 4.1 The Grid

## 4.2 The Auto-router

# 5.0 Revision History

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