PCB Design Guide

GT Off-Road Racing | Data Acquisition

Andrew Hellrigel

[06/01/2021]

Table of Contents

[1.0 Overview 2](#_Toc73403069)

[1.1 Introduction 2](#_Toc73403070)

[1.2 Point of Contact 2](#_Toc73403071)

[2.0 Hardware Reference 3](#_Toc73403072)

[2.1 [First section] 3](#_Toc73403073)

[2.1.1 [Subsection] 3](#_Toc73403074)

[3.0 Software Theory of Operation 4](#_Toc73403075)

[4.0 References 5](#_Toc73403076)

[5.0 Revision History 6](#_Toc73403077)

# 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

## 2.3 Contributing to GTOR’s EAGLE Libraries

# 3.0 Recommended Design Rules

## 3.1 Trace Width

### 3.1.1 Designing for JLCPCB

### 3.1.2 Designing for ME Electronics

# 5.0 Revision History

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