

---

# **PCB Tools Documentation**

*Release 0.1*

**Hamilton Kibbe**

June 23, 2017



<b>1</b>	<b>About PCB Tools</b>	<b>3</b>
1.1	Image Rendering . . . . .	3
1.2	Future Plans . . . . .	4
<b>2</b>	<b>Feature Support</b>	<b>5</b>
<b>3</b>	<b>PCB Tools Reference</b>	<b>7</b>
3.1	rs274x — RS-274X file handling . . . . .	7
3.2	excellon — Excellon file handling . . . . .	7
3.3	operations — Cam File operations . . . . .	8
3.4	render — Gerber file Rendering . . . . .	8
<b>4</b>	<b>Indices and tables</b>	<b>9</b>
	<b>Python Module Index</b>	<b>11</b>



Contents:



---

## About PCB Tools

---

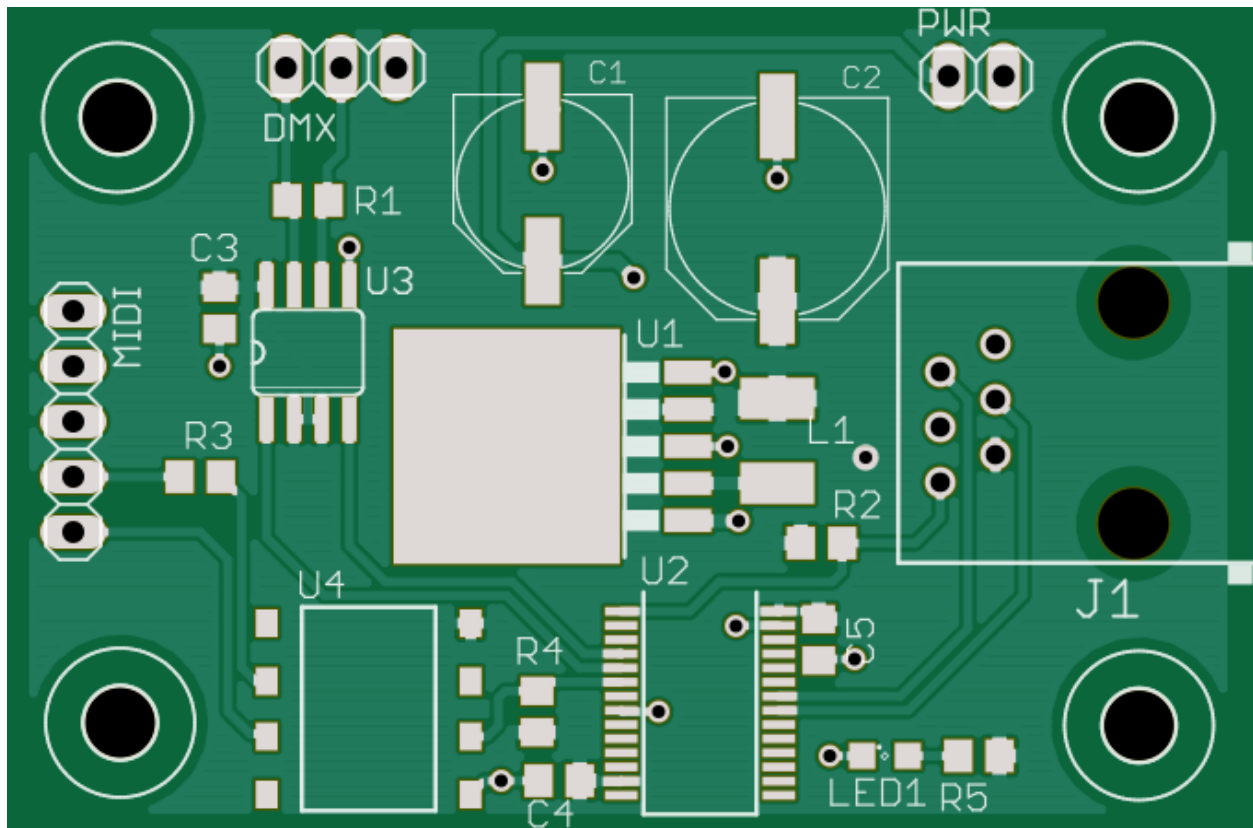
PCB Tools provides a set of utilities for visualizing and working with PCB design files in a variety of formats. The design files are generally referred to as Gerber files. This is a generic term that may refer to *RS-274X* (Gerber), *ODB++*, or *Excellon* files. These file formats are used by the CNC equipment used to manufacture PCBs.

PCB Tools currently supports the following file formats:

- Gerber (RS-274X)
- Excellon

with planned support for IPC-2581, ODB++ and more.

### Image Rendering



The PCB Tools module provides tools to visualize PCBs and export images in a variety of formats, including SVG and PNG.

## **Future Plans**

We are working on adding the following features to PCB Tools:

- Design Rules Checking
- Editing
- Panelization



---

## Feature Support

---

Currently supported features are as follows:

File Format	Parsing	Rendering	Unit Conversion	Scale	Offset	Rotate
RS274-X	Yes	Yes	Yes	No	Yes	No
Excellon	Yes	Yes	Yes	No	Yes	No
ODB++	No	No	No	No	No	No



---

## PCB Tools Reference

---

### **rs274x — RS-274X file handling**

The RS-274X (Gerber) format is the most common format for exporting PCB artwork. The Specification is published by Ucamco and is available [here](#). The `rs274x` submodule implements classes to read and write RS-274X files without having to know the precise details of the format.

The `rs274x` submodule's `read()` function serves as a simple interface for parsing gerber files. The `GerberFile` class stores all the information contained in a gerber file allowing the file to be analyzed, modified, and updated. The `GerberParser` class is used in the background for parsing RS-274X files.

#### **Functions**

The `rs274x` module defines the following functions:

#### **Classes**

The `rs274x` module defines the following classes:

### **excellon — Excellon file handling**

The Excellon format is the most common format for exporting PCB drill information. The Excellon format is used to program CNC drilling machines for drilling holes in PCBs. As such, excellon files are sometimes referred to as NC-drill files. The Excellon format reference is available [here](#). The `excellon` submodule implements classes to read and write excellon files without having to know the precise details of the format.

The `excellon` submodule's `read()` function serves as a simple interface for parsing excellon files. The `ExcellonFile` class stores all the information contained in an Excellon file allowing the file to be analyzed, modified, and updated. The `ExcellonParser` class is used in the background for parsing RS-274X files.

#### **Functions**

The `excellon` module defines the following functions:

## Classes

The `excellon` module defines the following classes:

## `operations` — Cam File operations

The `operations` module provides functions which modify `gerber.cam.CamFile` objects. All of the functions in this module return a modified copy of the supplied file.

## Functions

The `operations` module defines the following functions:

## `render` — Gerber file Rendering

### Render Module

---

**Indices and tables**

---

- *genindex*
- *modindex*
- *search*



**e**

excellon, 7

**o**

operations, 8

**r**

render, 8

rs274x, 7





## E

excellon (module), 7

## O

operations (module), 8

## R

render (module), 8

rs274x (module), 7