PyEAGLE Documentation

Release 0.1.2.dev

Scott Torborg

September 04, 2015

Contents

Scott Torborg

PyEAGLE is an interchange API to read and write Cadsoft EAGLE XML (v6 and later) schematic, library, and PCB layout files.

Some long term goals are:

- Comprehensive and round-trip-capable interface between Pythonic data structures and EAGLE's XML format, eliminating the impedance mismatch involved in manipulating EAGLE files in Python scripts.

- Traversal and net analysis helpers, to serve as a foundation for higher-level tools.

- Rendering capability in SVG.

- Command-line tools, including basic `grep` and `lint` tools.

At this point it is fairly incomplete and not 'production useful'.

Example Usage

```
lib = pyeagle.open('SparkFun-Sensors.lbr')
for part in lib:
    print part
```

Screenshot

Everyone loves screenshots.

Contents

3.1 Quick Start

3.1.1 Install

Install with pip:

```
$ pip install pyeagle
```

3.1.2 Use

Open a library file, print some info about the contents.

```python
import pyeagle

lib = pyeagle.open('SparkFun-Sensors.lbr')

for device_set in lib:  # could also use lib.device_sets
    print device_set

for symbol in lib.symbols:
    print symbol

for package in lib.packages:
    print package
```

3.2 API Reference

3.3 Contributing

Patches and suggestions are strongly encouraged! GitHub pull requests are preferred, but other mechanisms of feedback are welcome.

PyEAGLE hopes to have a comprehensive test suite with 100% line and branch coverage, as reported by the excellent `coverage` module. To run the tests, simply run in the top level of the repo:

```
$ tox
```

This will also ensure that the Sphinx documentation builds correctly, and that there are no PEP8 or Pyflakes warnings in the codebase.

3.3.1 Contributors

Thanks to Bernard "Guyzmo" Pratz for `zc.buildout` support.

Indices and Tables

- genindex
- modindex