
Regerberate Documentation

Release 0.0.1.dev

Scott Torborg

November 29, 2015

1	Contents	3
1.1	Quick Start	3
2	Indices and Tables	5

Scott Torborg - scotttorborg.com

This is a tool to let you integrate SVG editing tools like Illustrator or Inkscape into your PCB design workflow.

Contents

1.1 Quick Start

1.1.1 Install

Install with pip:

```
$ pip install regerberate
```

1.1.2 Workflow

The goal of the Regerberate workflow is to make it easy and fast to use an SVG editor to edit layers in a PCB layout, while enabling version control, single-step production builds, and flexibility between EDA packages.

The typical workflow looks like this:

Initial Board Design

Use any EDA package to design a board. The board shouldn't include elements which will be edited in SVG tools: in order to avoid "redoing" work when edits are made in the source EDA package, additional SVG artwork should only be composited, and not subtract from or mutate "base" art.

Generate a set of base Gerber files from the EDA tool. This is identical to the process that you would normally use to generate Gerber files for production.

SVG File Creation

Call regerberate on the set of Gerber files to build an SVG.:

```
$ regerberate prepare -o myboard.svg intermediate/*.ger
```

For each source Gerber file, a pair of layers will be created in the SVG file.

One layer in the pair is the *base* layer as output by the EDA package. It should generally not be modified, and just serves as a visual reference to design other art around. Regerberate can update this layer based on new Gerbers exported by the EDA package.

The other layer in the pair is the *extra* layer, and is where additional artwork should be added.

SVG Editing

Drawing primitives in the *extra* SVG layers will be replicated in the output Gerber files. A wide variety of SVG features cannot be translated into the Gerber format, like colors and zero-width lines. These should be avoided.

Rendering of SVG to Gerber

The SVG file can then be rendered into a new set of output Gerber files ready for production.:

```
$ regerberate render -o build/ myboard.svg
```

Each Gerber file will contain the composite of the corresponding *base* and *extra* SVG layer pair. Gerber filenames will be preserved from the intermediate Gerber files used to create them.

Indices and Tables

- `genindex`
- `modindex`