
SolidPython Documentation

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Contents

1	SolidPython	3
2	SolidPython: OpenSCAD for Python	5
3	Advantages	7
4	Installing SolidPython	9
5	Using SolidPython	11
6	Importing OpenSCAD code	13
7	Example Code	15
8	Extra syntactic sugar	17
8.1	Basic operators	17
8.2	First-class Negative Space (Holes)	17
8.3	Animation	18
9	solid.utils	19
9.1	Directions: (up, down, left, right, forward, back) for arranging things:	19
9.2	Arcs	19
9.3	Extrude Along Path	20
9.4	Basic color library	20
9.5	Bill Of Materials	20
9.6	solid.screw_thread	20
9.7	Jupyter Renderer	21
10	Contact	23
11	License	25
12	Library Reference	27
13	Indices and tables	29

Contents:

- *SolidPython: OpenSCAD for Python*
- *Advantages*
- *Installing SolidPython*
- *Using SolidPython*
- *Importing OpenSCAD Code*
- *Example Code*
- *Extra syntactic sugar*
 - *Basic operators*
 - *First-class Negative Space (Holes)*
 - *Animation*
- *solid.utils*
 - *Directions: (up, down, left, right, forward, back) for arranging things:*
 - *Arcs*
 - *Extrude Along Path*
 - *Basic color library*
 - *Bill Of Materials*
- *solid.screw_thread*
- *Jupyter Renderer*
- *Contact*
- *License*

CHAPTER 2

SolidPython: OpenSCAD for Python

SolidPython is a generalization of Phillip Tiefenbacher's openscad module, found on [Thingiverse](#). It generates valid OpenSCAD code from Python code with minimal overhead. Here's a simple example:

This Python code:

```
from solid import *
d = difference() (
    cube(10),
    sphere(15)
)
print(scad_render(d))
```

Generates this OpenSCAD code:

```
difference() {
    cube(10);
    sphere(15);
}
```

That doesn't seem like such a savings, but the following SolidPython code is a lot shorter (and I think clearer) than the SCAD code it compiles to:

```
from solid import *
from solid.utils import *
d = cube(5) + right(5)(sphere(5)) - cylinder(r=2, h=6)
```

Generates this OpenSCAD code:

```
difference() {
    union() {
        cube(5);
        translate([5, 0, 0]) {
            sphere(5);
        }
    }
}
```

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```
}  
    cylinder(r=2, h=6);  
}
```

CHAPTER 3

Advantages

Because you're using Python, a lot of things are easy that would be hard or impossible in pure OpenSCAD. Among these are:

- built-in dictionary types
- mutable, slice-able list and string types
- recursion
- external libraries (images! 3D geometry! web-scraping! ...)

CHAPTER 4

Installing SolidPython

- Install via [PyPI](#):

```
pip install solidpython
```

(You may need to use `sudo pip install solidpython`, depending on your environment. This is commonly discouraged though. You'll be happiest working in a [virtual environment](#) where you can easily control dependencies for a given project)

Using SolidPython

- Include SolidPython at the top of your Python file:

```
from solid import *  
from solid.utils import * # Not required, but the utils module is useful
```

(See [this issue](#) for a discussion of other import styles)

- OpenSCAD uses curly-brace blocks ({ }) to create its tree. SolidPython uses parentheses with comma-delimited lists.

OpenSCAD:

```
difference() {  
    cube(10);  
    sphere(15);  
}
```

SolidPython:

```
d = difference() (  
    cube(10), # Note the comma between each element!  
    sphere(15)  
)
```

- Call `scad_render(py_scad_obj)` to generate SCAD code. This returns a string of valid OpenSCAD code.
- *or:* call `scad_render_to_file(py_scad_obj, filepath.scad)` to store that code in a file.
- If `filepath.scad` is open in the OpenSCAD IDE and Design => 'Automatic Reload and Compile' is checked in the OpenSCAD IDE, running `scad_render_to_file()` from Python will load the object in the IDE.
- Alternately, you could call OpenSCAD's command line and render straight to STL.

Importing OpenSCAD code

- Use `solid.import_scad(path)` to import OpenSCAD code.

Ex:

`scadfile.scad`

```
module box(w,h,d) {
    cube([w,h,d]);
}
```

`your_file.py`

```
from solid import *

scadfile = import_scad('/path/to/scadfile.scad')
b = scadfile.box(2,4,6)
scad_render_to_file(b, 'out_file.scad')
```

- Recursively import OpenSCAD code by calling `import_scad()` with a directory argument.

```
from solid import *

# MCAD is OpenSCAD's most common utility library: https://github.com/openscad/MCAD
mcad = import_scad('/path/to/MCAD')

# MCAD contains about 15 separate packages, each included as its own namespace
print(dir(mcad)) # => ['bearing', 'bitmap', 'boxes', etc...]
mount = mcad.motors.stepper_motor_mount(nema_standard=17)
scad_render_to_file(mount, 'motor_mount_file.scad')
```

- OpenSCAD has the `use()` and `include()` statements for importing SCAD code, and SolidPython has them, too. They pollute the global namespace, though, and you may have better luck with `import_scad()`,

Ex: `scadfile.scad`

```
module box(w,h,d) {  
    cube([w,h,d]);  
}
```

your_file.py

```
from solid import *  
  
# use() puts the module `box()` into the global namespace  
use('/path/to/scadfile.scad')  
b = box(2,4,6)  
scad_render_to_file(b, 'out_file.scad')
```

CHAPTER 7

Example Code

The best way to learn how SolidPython works is to look at the included example code. If you've installed SolidPython, the following line of Python will print(the location of) the examples directory:

```
import os, solid; print(os.path.dirname(solid.__file__) + '/examples')
```

Or browse the example code on Github [here](#)

Adding your own code to the example file `solid/examples/solidpython_template.py` will make some of the setup easier.

Extra syntactic sugar

8.1 Basic operators

Following Elmo Mäntynen's suggestion, SCAD objects override the basic operators + (union), - (difference), and * (intersection). So

```
c = cylinder(r=10, h=5) + cylinder(r=2, h=30)
```

is the same as:

```
c = union() (
    cylinder(r=10, h=5),
    cylinder(r=2, h=30)
)
```

Likewise:

```
c = cylinder(r=10, h=5)
c -= cylinder(r=2, h=30)
```

is the same as:

```
c = difference() (
    cylinder(r=10, h=5),
    cylinder(r=2, h=30)
)
```

8.2 First-class Negative Space (Holes)

OpenSCAD requires you to be very careful with the order in which you add or subtract objects. SolidPython's `hole()` function makes this process easier.

Consider making a joint where two pipes come together. In OpenSCAD you need to make two cylinders, union them, then make two smaller cylinders, union them, then subtract the smaller from the larger.

Using `hole()`, you can make a pipe, specify that its center should remain open, and then add two pipes together knowing that the central void area will stay empty no matter what other objects are added to that structure.

Example:

```
outer = cylinder(r=pipe_od, h=seg_length)
inner = cylinder(r=pipe_id, h=seg_length)
pipe_a = outer - hole()(inner)
```

Once you've made something a hole, eventually you'll want to put something, like a bolt, into it. To do this, we need to specify that there's a given 'part' with a hole and that other parts may occupy the space in that hole. This is done with the `part()` function.

See [solid/examples/hole_example.py](#) for the complete picture.

8.3 Animation

OpenSCAD has a special variable, `$t`, that can be used to animate motion. SolidPython can do this, too, using the special function `scad_render_animated_file()`.

See [solid/examples/animation_example.py](#) for more details.

SolidPython includes a number of useful functions in `solid/utils.py`. Currently these include:

9.1 Directions: (up, down, left, right, forward, back) for arranging things:

```
up(10) (  
    cylinder()  
)
```

seems a lot clearer to me than:

```
translate( [0,0,10]) (  
    cylinder()  
)
```

I took this from someone's SCAD work and have lost track of the original author.
My apologies.

9.2 Arcs

I've found this useful for fillets and rounds.

```
arc(rad=10, start_degrees=90, end_degrees=210)
```

draws an arc of radius 10 counterclockwise from 90 to 210 degrees.

```
arc_inverted(rad=10, start_degrees=0, end_degrees=90)
```

draws the portion of a 10x10 square NOT in a 90 degree circle of radius 10. This is the shape you need to add to make fillets or remove to make rounds.

9.3 Extrude Along Path

```
solid.utils.extrude_along_path(shape_pts, path_pts, scale_factors=None)
```

See [solid/examples/path_extrude_example.py](#) for use.

9.4 Basic color library

You can change an object's color by using the OpenSCAD `color([rgba_array])` function:

```
transparent_blue = color([0,0,1, 0.5])(cube(10)) # Specify with RGBA]
red_obj = color(Red)(cube(10))                  # Or use predefined colors
```

These colors are pre-defined in `solid.utils`:

Red	Green	Blue
Cyan	Magenta	Yellow
Black	White	Transparent
Oak	Pine	Birch
Iron	Steel	Stainless
Aluminum	Brass	BlackPaint
FiberBoard		

They're a conversion of the materials in the [MCAD OpenSCAD library](https://github.com/openscad/MCAD/blob/master/materials.scad), as seen [here] (<https://github.com/openscad/MCAD/blob/master/materials.scad>).

9.5 Bill Of Materials

Put `@bom_part()` before any method that defines a part, then call `bill_of_materials()` after the program is run, and all parts will be counted, priced and reported.

The example file [solid/examples/bom_scad.py](#) illustrates this. Check it out.

9.6 solid.screw_thread

`solid.screw_thread` includes a method, `thread()` that makes internal and external screw threads.

See [solid/examples/screw_thread_example.py](#) for more details.

9.7 Jupyter Renderer

Render SolidPython or OpenSCAD code in Jupyter notebooks using [ViewSCAD](#), or install directly via:

```
pip install viewscad
```

(Take a look at the [repo page](#), though, since there's a tiny bit more installation required)

CHAPTER 10

Contact

Enjoy, and please send any questions or bug reports to me at evan_t_jones@mac.com.

Cheers!

Evan

CHAPTER 11

License

This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version.

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[Full text of the license.](#)

Some class docstrings are derived from the [OpenSCAD User Manual](#), so are available under the [Creative Commons Attribution-ShareAlike License](#).

CHAPTER 12

Library Reference

CHAPTER 13

Indices and tables

- genindex
- modindex
- **search**

members