Computational Design + Fabrication: SolidPython

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- python layer to openscad
- allows full power of python to script solid creation
- better syntax
- adds few utilities

```
import solid, numpy
```

- circle, square
- sphere, cube, cylinder

```
circle(10)
circle(d=10)
square(10)
```

Compound Functions

- union, intersection, difference
- linear_extrude, rotate_extrude

```
union()( circle(d=10), square(size=9) )
```

versus

```
union(){ circle(d=10); square(size=9) }
```

Abbreviations

- `+`, `*`, `-`
- union, intersection, difference

```
circle(d=10) + square(size=9)
```

versus

```
union()( circle(d=10), square(size=9) )
```

Translations Shortcuts

- up,down,left,right,forward,back

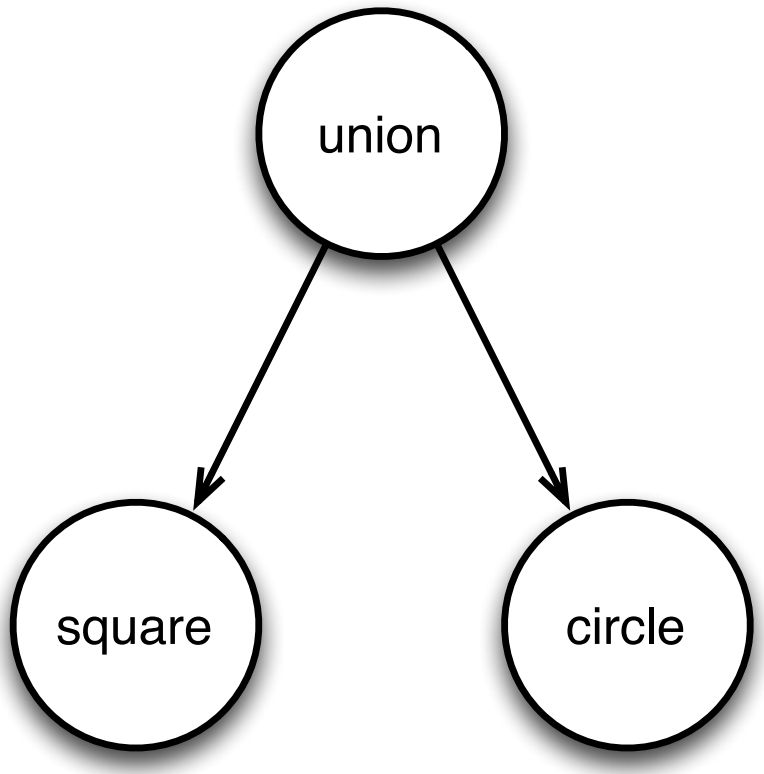
```
up(10)( circle(10) )
```

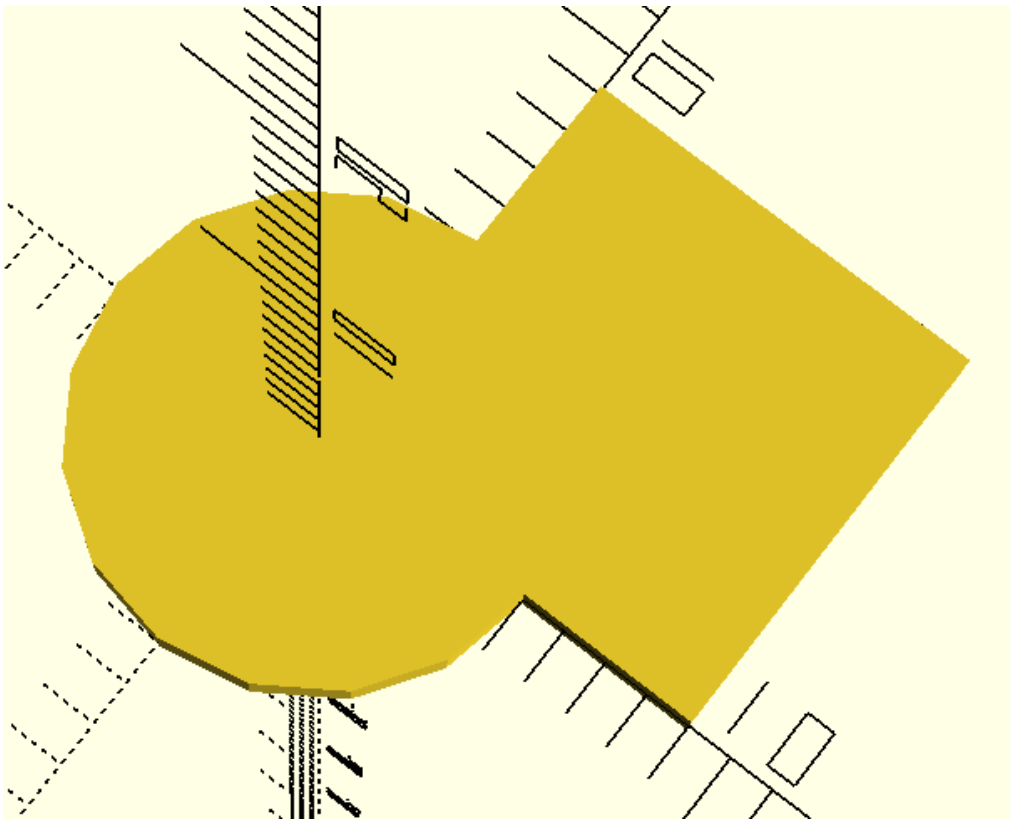
versus

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

Expression Tree One

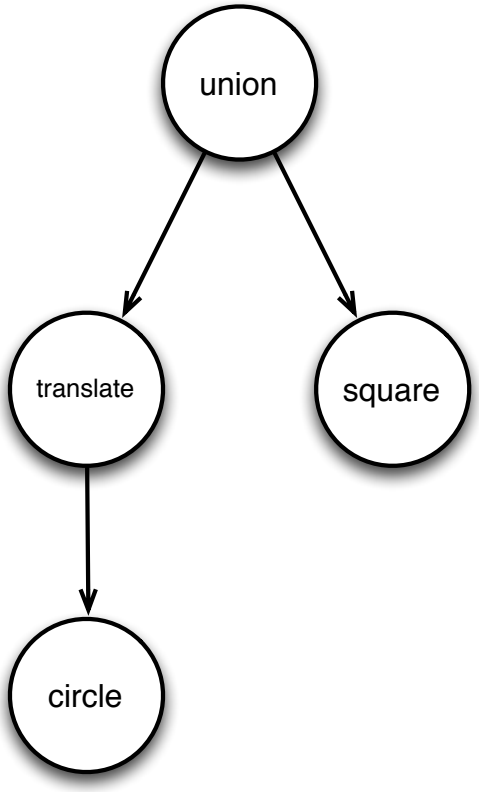

```
circle(d=10) + square(size=9)
```

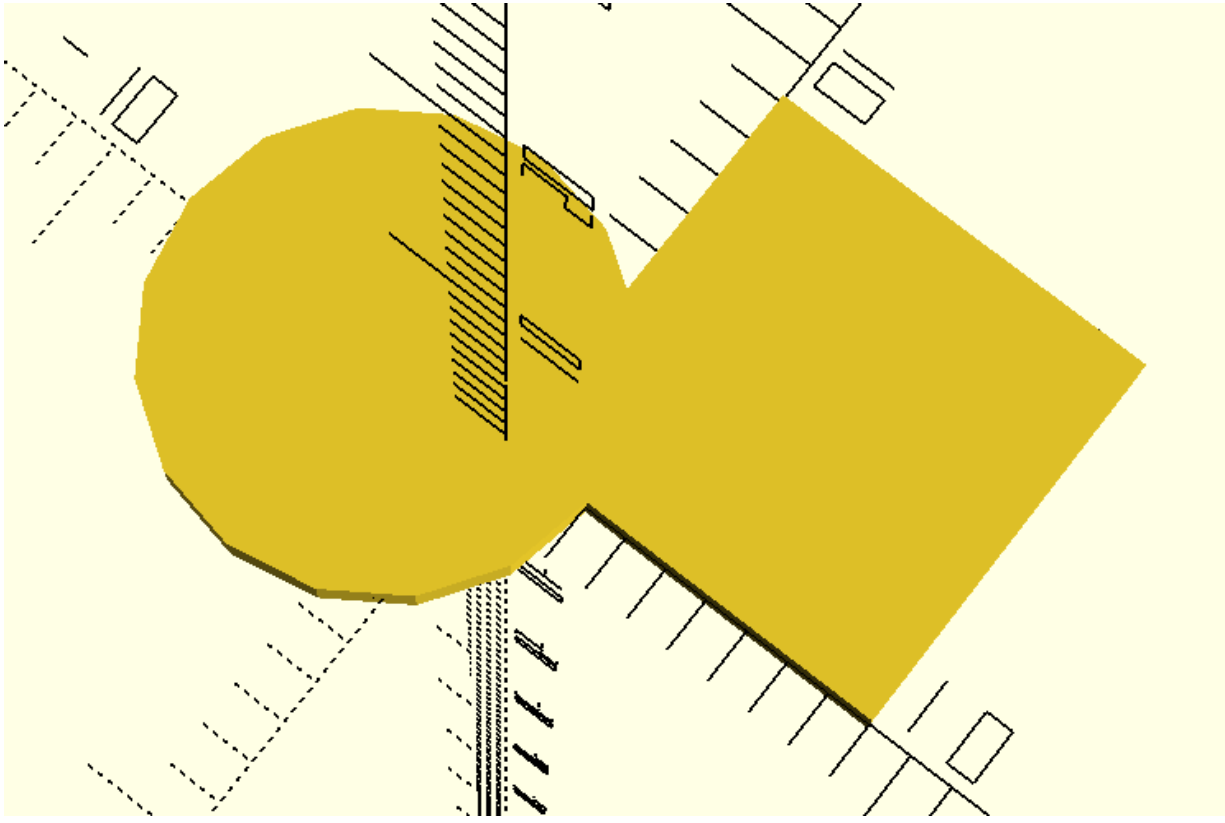




Expression Tree Two

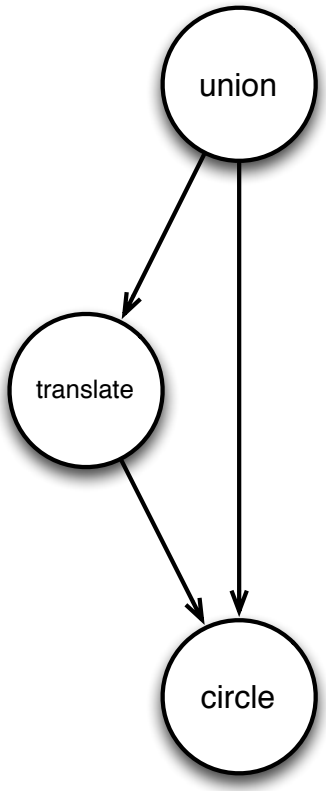

```
left(3)( circle(d=10) ) + square(size=9)
```

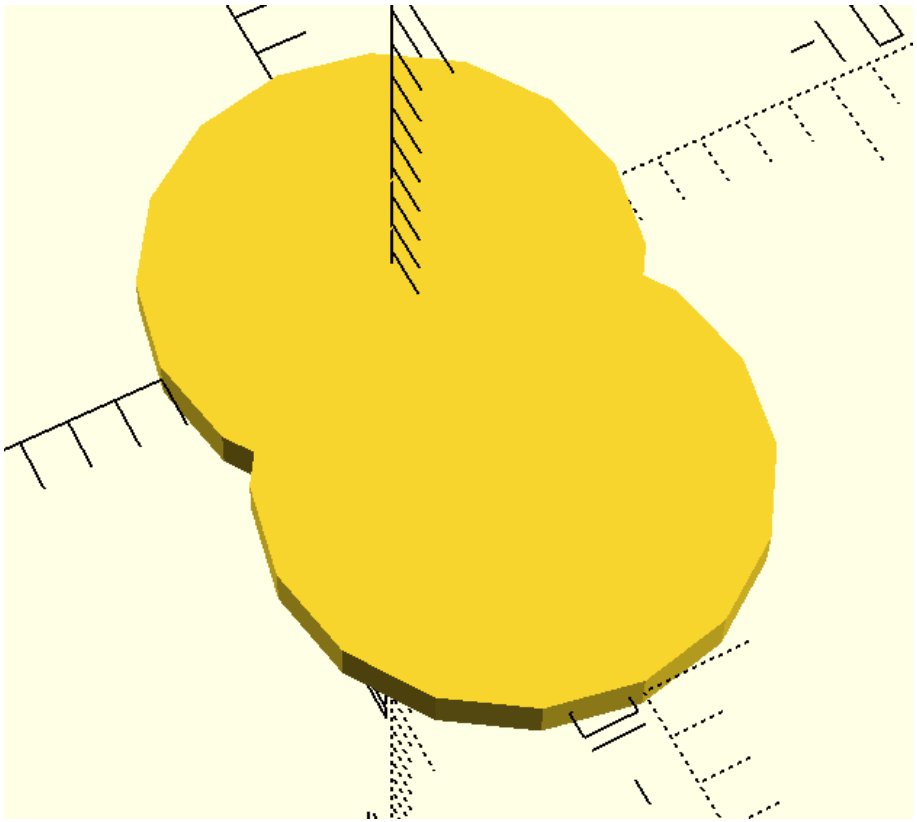




Expression Tree Three


```
c = circle(d=10)
left(3)( c ) + c
```

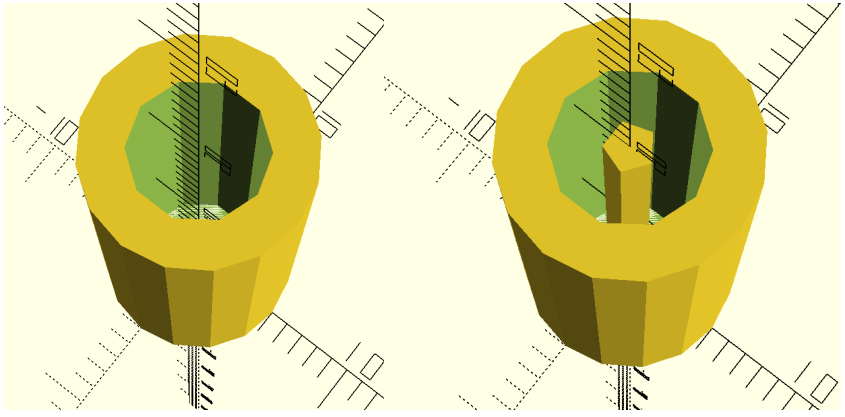




- keeping holes holes
- move hole to end
- allow screw in hole with parts

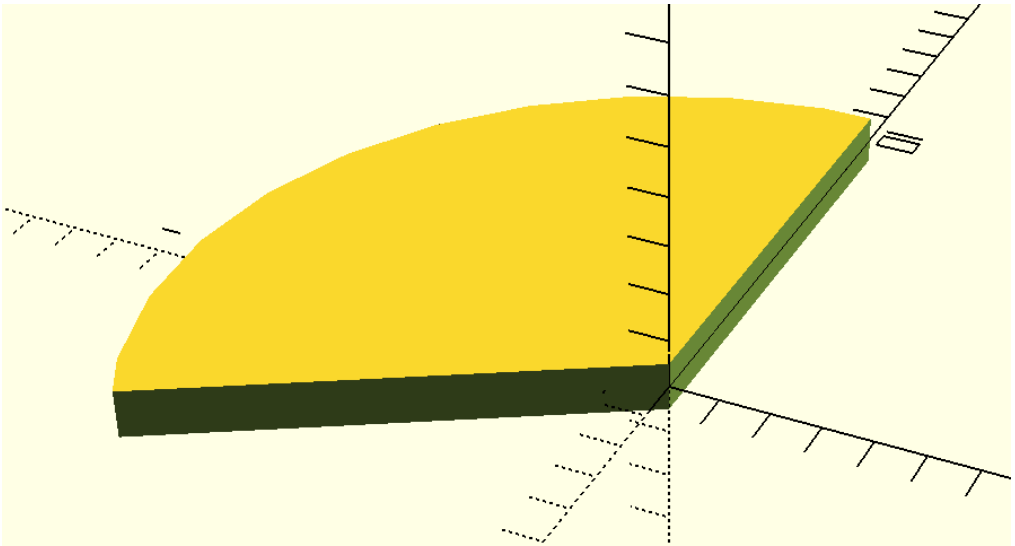
```
cylinder(r=5,h=20) + hole()( cylinder(r=3,h=21) )
```

```
part()( cylinder(r=5,h=20) + hole()( cylinder(r=3,h=21) ) ) +
  cylinder(r=1,h=20)
```



- arc
- for fillets and rounds
- rad, start_degrees, end_degrees
- arc_inverted – not in the square

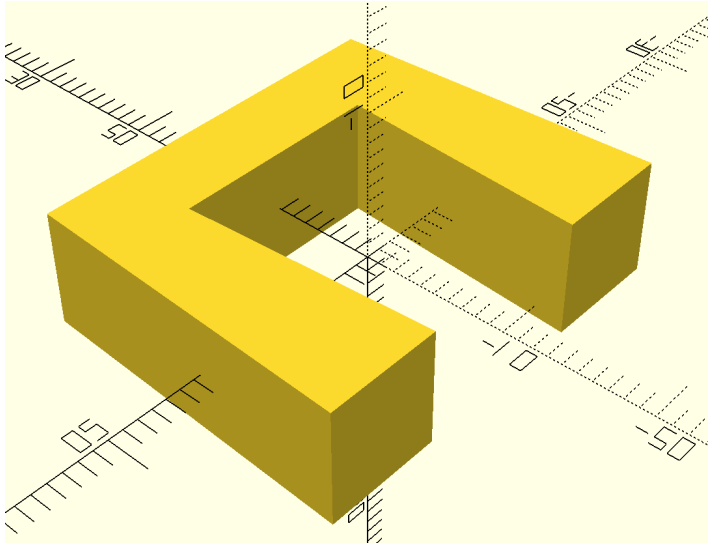
```
arc(rad=10, start_degrees=90, end_degrees=210)
```



Extrude Along Path


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

```
sqr = [[-4,-4],[-4,4],[4,4],[4,-4],[-4,-4]]
pth = [[-10,-10],[-10,10],[10,10],[10,-10]]
extrude_along_path(sqr, pth)
```



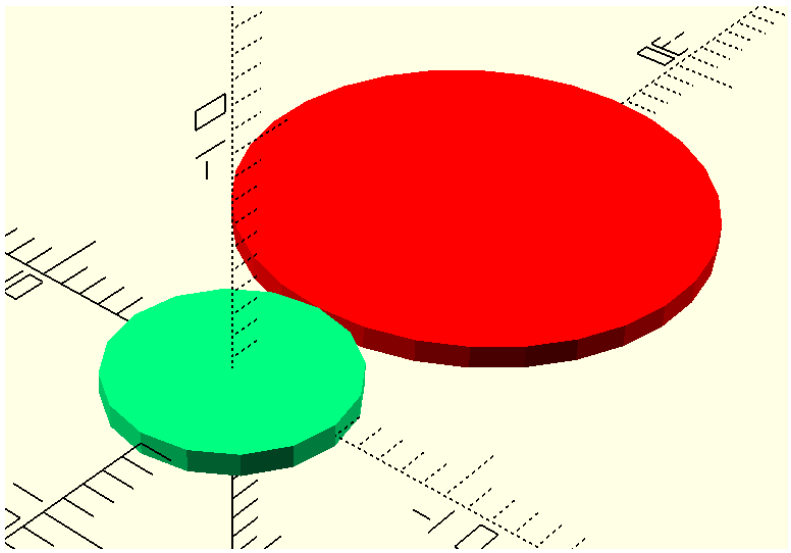
Offset

- not yet able to access openscad's offset

```
offset_points(point_arr, offset, inside=True)
```

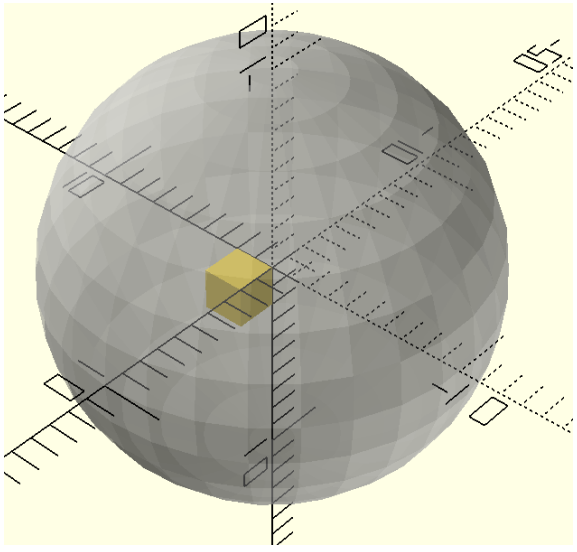
- color
- built in colors

```
color([0.0,1.0,0.5])( circle(5) ) +
  left(15)( color(Red)( circle(10) ) )
```



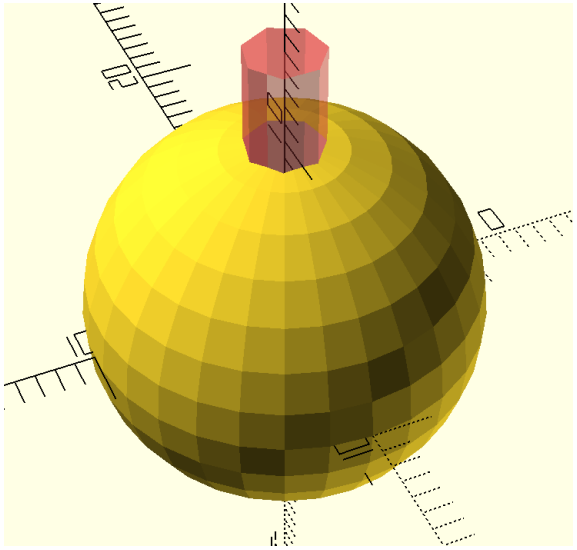
- background, debug, root, disable

```
background( sphere(10) ) + cube(2)
```



Debug

```
sphere(10) - debug( cylinder(r=2,h=15) )
```



References

- SolidPython – `https://github.com/SolidCode/`
- Open SCAD – `http://www.openscad.org`