

OpenSCAD

The Programmers Solid 3D CAD Modeller

The screenshot shows the OpenSCAD application interface. On the left is the 'Editor' window containing the following SCAD code:

```
1 bodywidth = dxf_dim(file = "example009.dxf", name = "bodywidth");
2 fanwidth = dxf_dim(file = "example009.dxf", name = "fanwidth");
3 platewidth = dxf_dim(file = "example009.dxf", name = "platewidth");
4 fan_side_center = dxf_cross(file = "example009.dxf", layer = "fan_side_center");
5 fanrot = dxf_dim(file = "example009.dxf", name = "fanrot");
6
7 % linear_extrude(height = bodywidth, center = true, convexity = 10)
8 - import(file = "example009.dxf", layer = "body");
9
10 * for (z = [-(bodywidth/2 + platewidth/2),
11 ---(bodywidth/2 + platewidth/2)]) {
12     translate([0, 0, z])
13     linear_extrude(height = platewidth, center = true, convexity = 10)
14     import(file = "example009.dxf", layer = "plate");
15 }
16
17 intersection() {
18     linear_extrude(height = fanwidth, center = true, convexity = 10, twist = -fanrot)
19     import(file = "example009.dxf", layer = "fan_top");
20 ...
21 // NB! We have to use the deprecated module here since the "fan_side"
22 // layer contains an open polyline, which is not yet supported
23 // by the import() module.
24 rotate_extrude(file = "example009.dxf", layer = "fan_side",
25                 origin = fan_side_center, convexity = 10);
26 }
27
```

Below the code editor is a status bar with the text: 'Viewport: translate = [0.61 -1.31 -2.07], rotate = [55.00 0.00 25.00], distance = 142.23'. At the bottom right of the interface, it says 'OpenSCAD 2015.03'.

The main window on the right displays a 3D preview of a mechanical part, specifically a fan assembly, showing the yellow fan blades and the grey housing. The preview includes a coordinate system and various selection and transformation tools at the bottom.



Customizable Garden Sign

by [TheNewHobbyist](#) April 23, 2013



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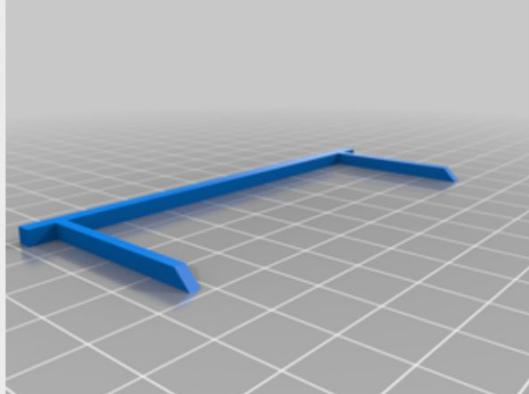


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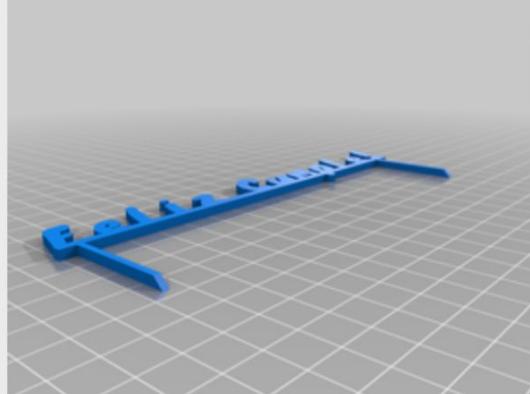
OREGANO Garden Si...

Jun 29, 2020



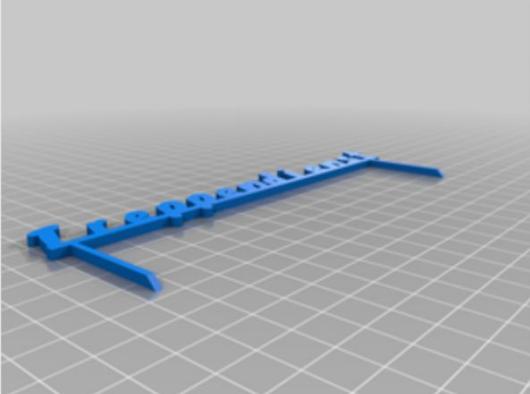
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My Customized Garde...

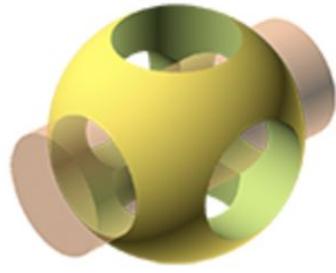
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DILL Garden Sign

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OpenSCAD

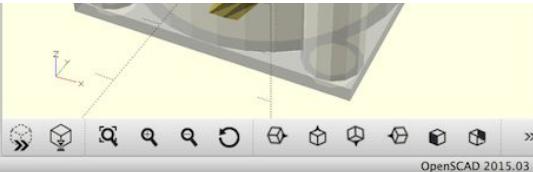
The Programmers Solid 3D CAD Modeller



Totally parametric!!!

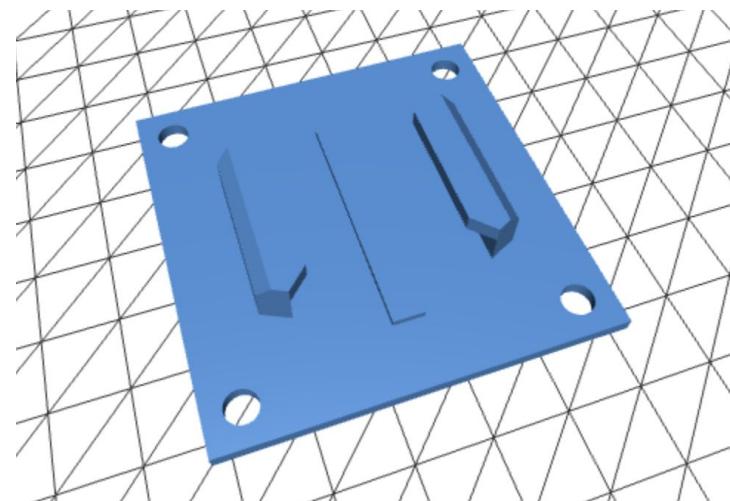
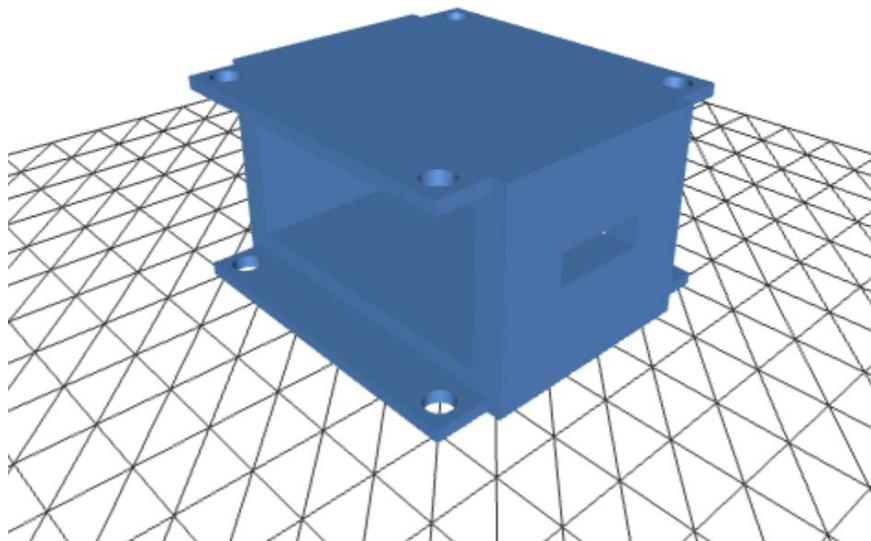
```
1 // intersection();
2
3 linear_extrude(height = fanwidth, center = true, convexity = 10, twist = -fanrot)
4   import(file = "example009.dxf", layer = "fan_top");
5
6
7 // NB! We have to use the deprecated module here since the "fan_side"
8 // layer contains an open polyline, which is not yet supported
9 // by the import() module.
10 rotate_extrude(file = "example009.dxf", layer = "fan_side",
11   origin = fan_side_center, convexity = 10);
12
13 }
```

Viewport: translate = [0.61 -1.31 -2.07], rotate = [55.00 0.00 25.00], distance = 142.23



what can you make

drone gopro mount and battery housing



architectural models

Providence, R. I.
March 19, 1916

Special Features

The Providence Sunday Journal

Fifth Section

A STRIKING PLAN FOR DIGNIFYING CIVIC CENTRE

Design Copyright 1916 by Raymond M. Hood

The Main Entrance to the Court House.

Typical Floor Plan of the Year.

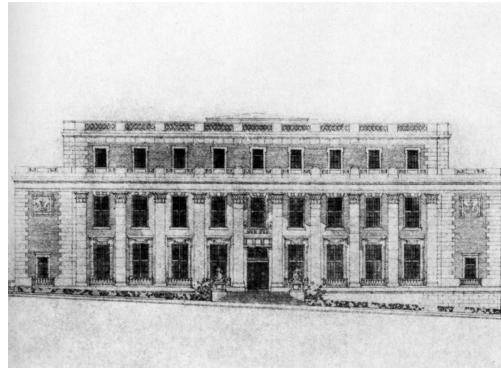
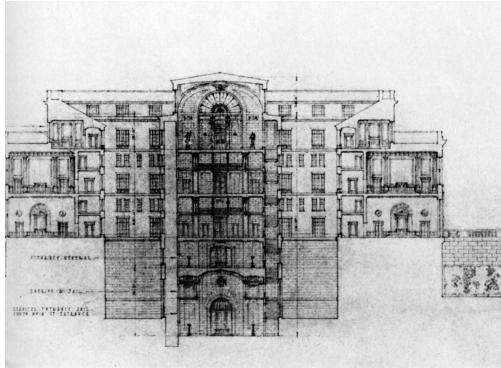
Former Rhode Islander Suggests Imposing State and Municipal Group. With Tower to Occupy Entire Square South of Exchange Place

SARTORIUS, architect, has submitted his architectural plans for a proposed group of buildings in the civic center of Providence, which he believes will add greatly to the beauty and dignity of the city. The project, which is to be known as the "State and Municipal Group," will consist of a tall, slender tower, a large rectangular building, and a smaller one, all facing Exchange Place. The tower will be topped with a spire and will have a clock face on each of its four sides. The main entrance to the group will be located on the side facing Exchange Place. The rectangular building will contain the state capitol, the municipal building, and the post office. The smaller building will be used for public meetings and will have a balcony overlooking the square. The entire group will be surrounded by a fence and will be illuminated at night.

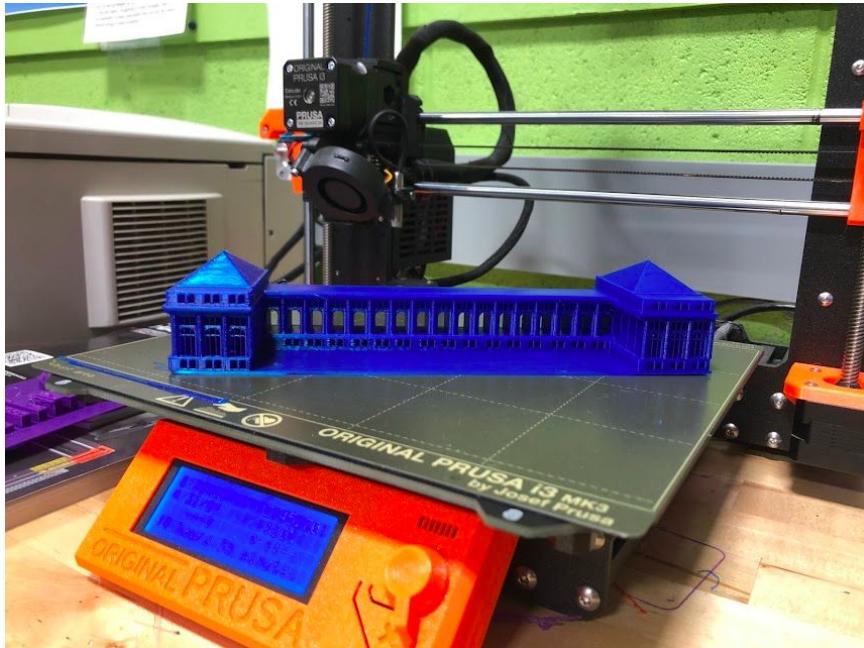
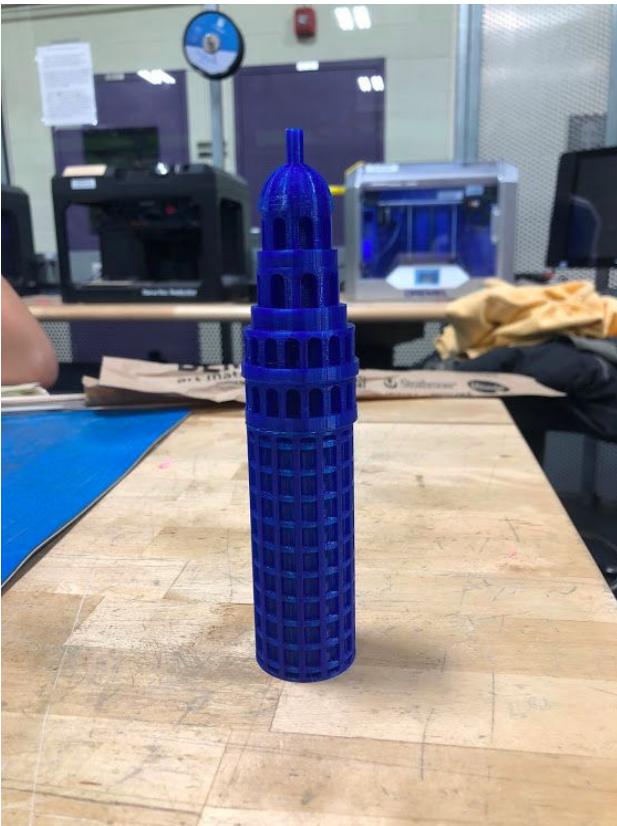
View of the proposed group from the North

Plan Showing Relation of Suggested Buildings to the City Hall and Post Office

Lieuation of Court House, Federal and Municipal Buildings

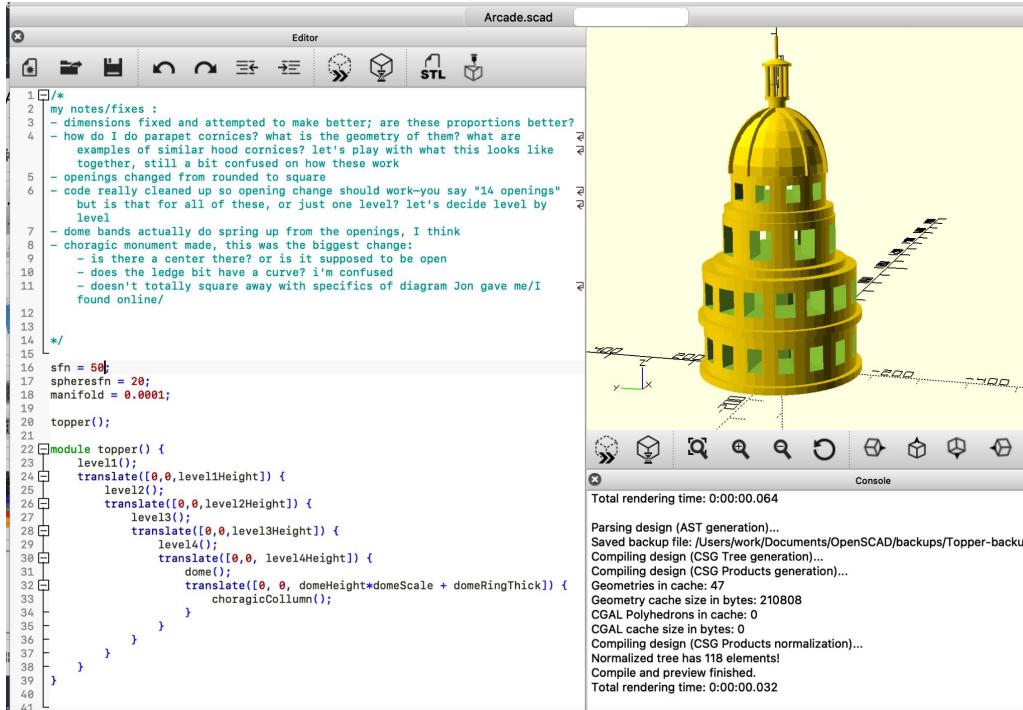


architectural models



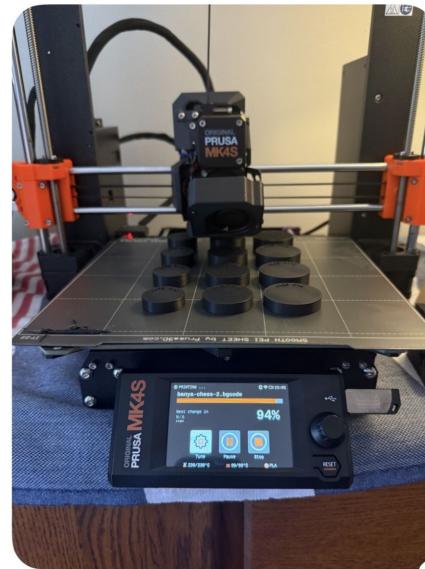
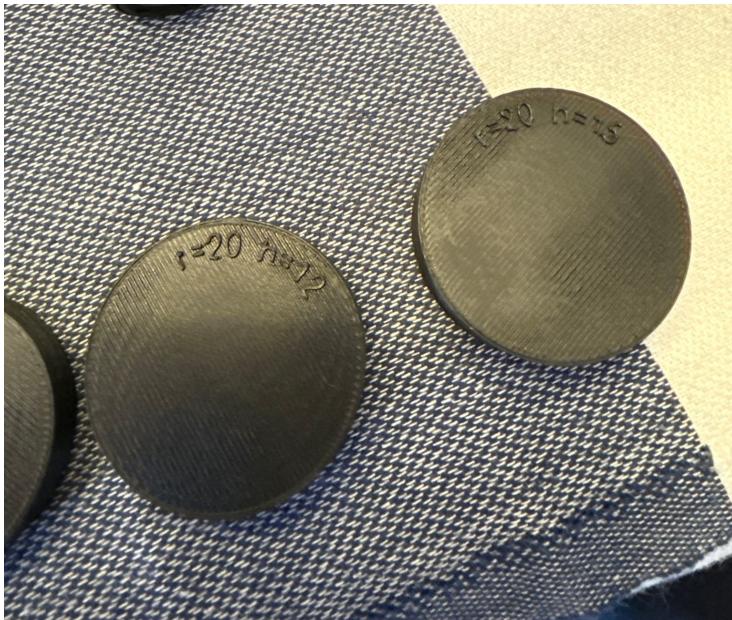
architectural models

<https://github.com/lucasgelfond/Providence-City-Center-Model>



Monday 3:06 PM

replacement banya chess pieces



printed 12 cylinders of varying widths and height to bring to banya for chess piece replacement



labeled with radius and height so I can tell

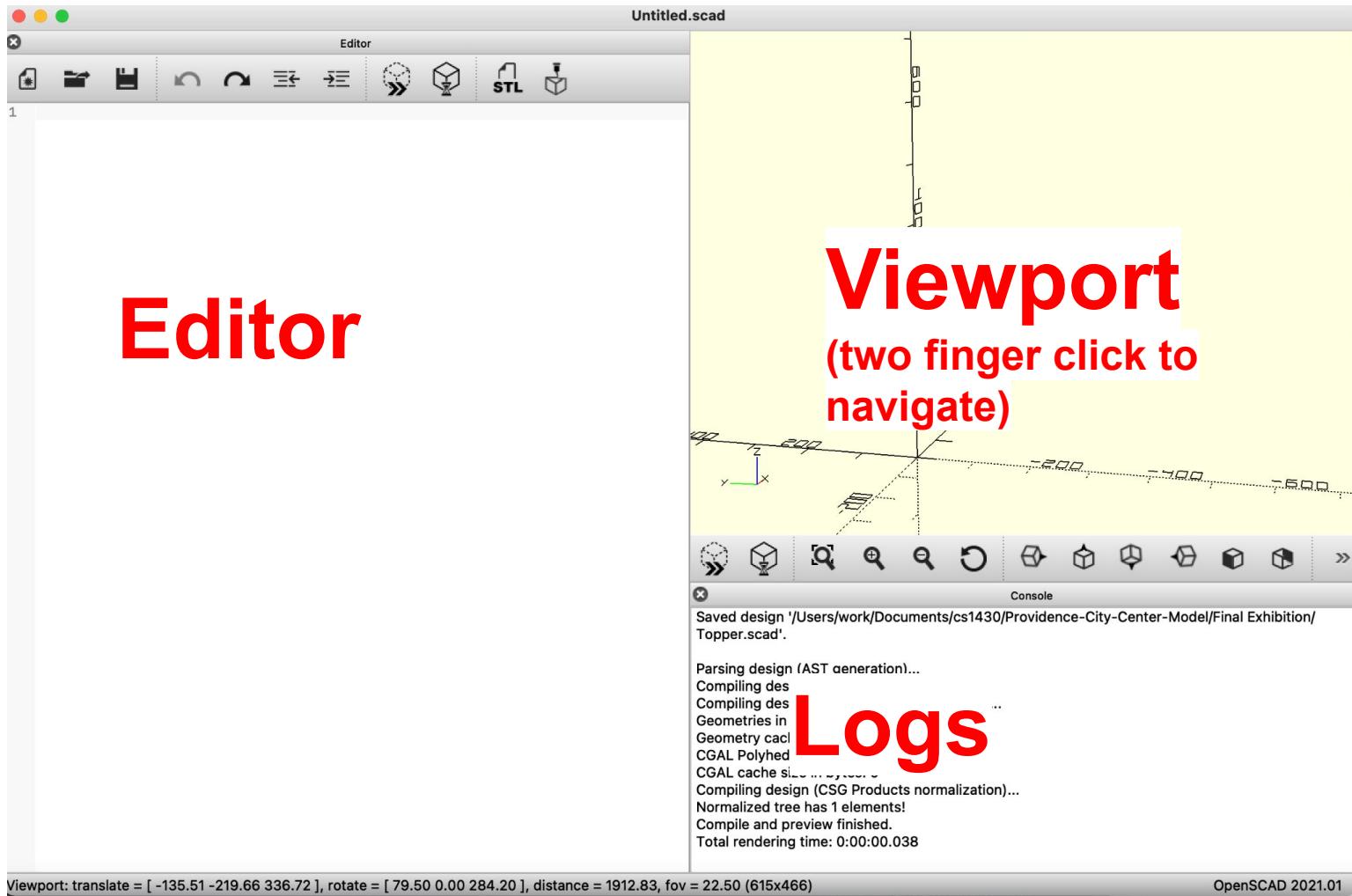
Interactive part!

- download OpenSCAD (~27 MB)
 - <http://openscad.org/downloads.html>

OR

- brew install openscad
- the official docs (linked on their site) are *incredible*

Editor

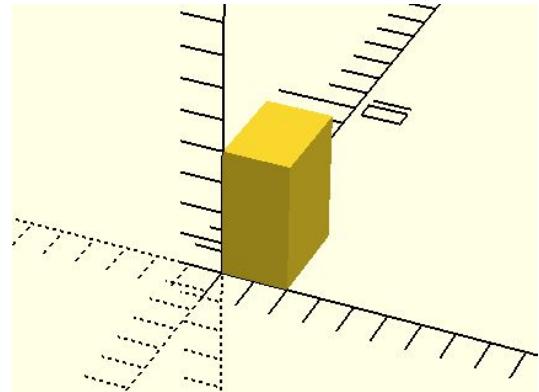


information dump / crash course

(and reminder to self to not speed through this)

```
cube([2, 3, 4]);
```

F5 to render



click  to Reset View if you can't see

cylinder($r = 3$, $h = 5$);

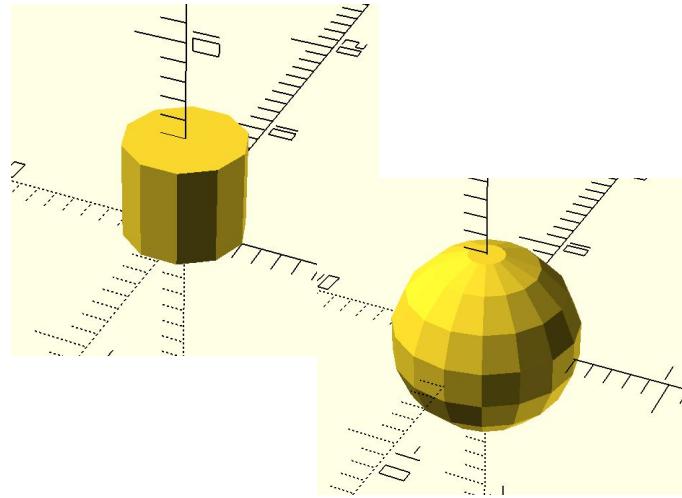
sphere($r = 5$)

object primitives:

cube([width, depth, height]);

cylinder($r = \text{radius}$, $h = \text{height}$)

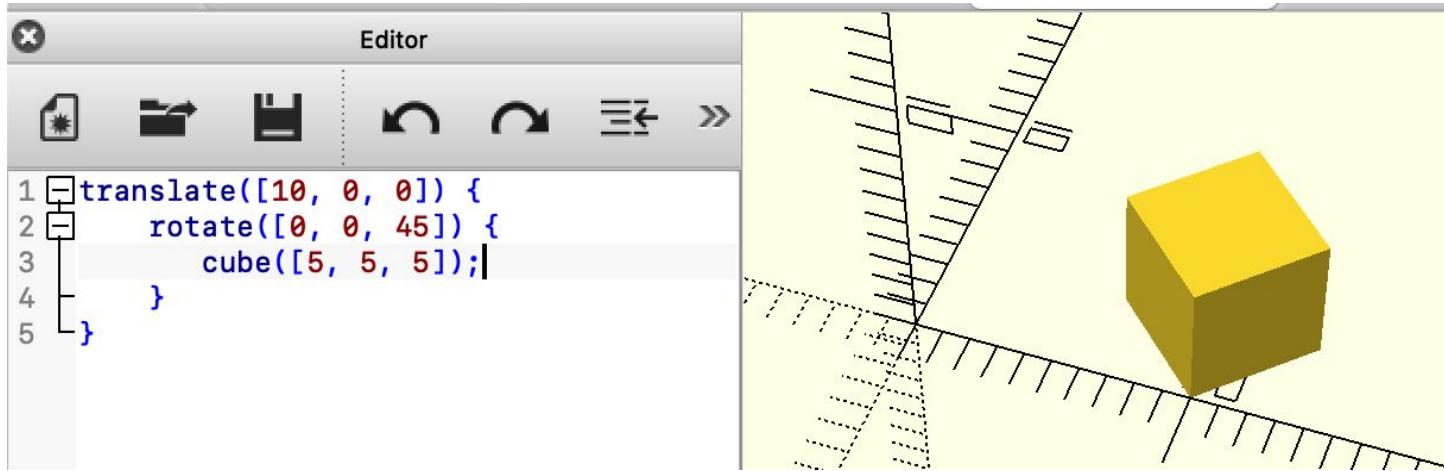
sphere($r = \text{radius}$)



transforms

- `translate([x, y, z]) { }`
- `rotate([x, y, z]) { } (degrees)`
- `scale ([x, y, z]) { } (default: [1, 1, 1])`

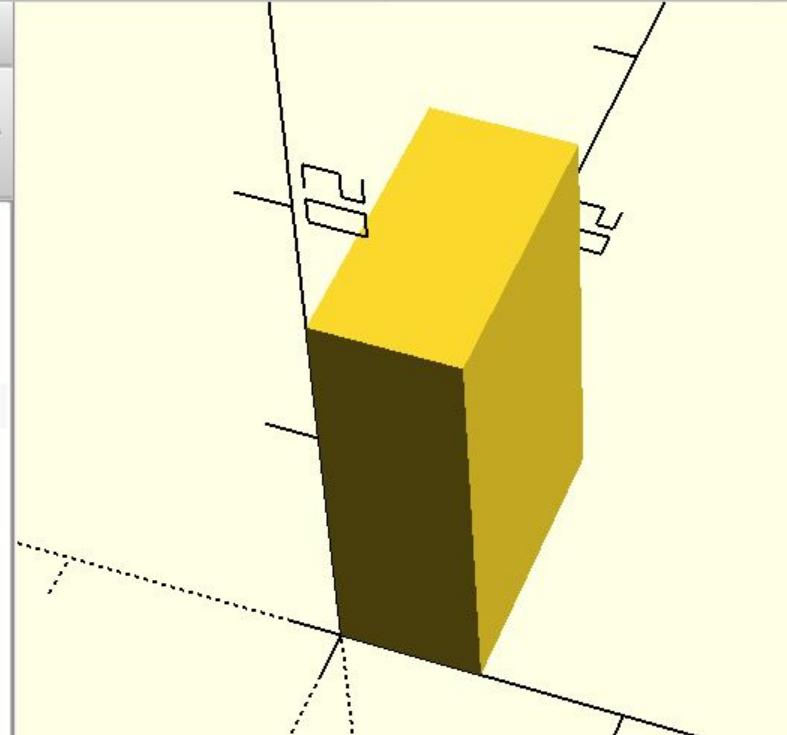
transforms can be nested!



This would

- *create a 5x5x5 cube*
- *rotate it 45 degrees about the Z axis*
- *translate it 10 units across the X axis*
- NOTE: would get a different result if we did in a different order!

variables never change and need no declaration



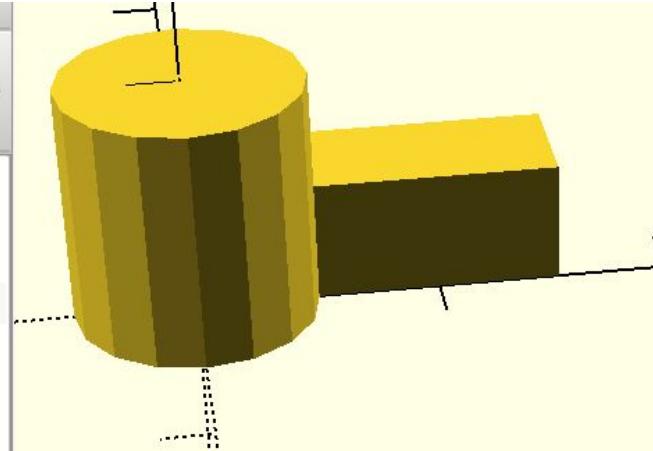
The image shows a 3D rendering of a cube in a coordinate space. The cube is oriented diagonally, with its top face colored yellow and the bottom face colored dark brown/black. It is positioned in the center of a 3D grid. The background is a plain light color.

Editor

File Save Undo Redo Align Fit

```
1 boxWidth = 5;
2 boxDepth = 10;
3 boxHeight = 15;
4
5 cube([boxWidth, boxDepth, boxHeight]);|
```

union()



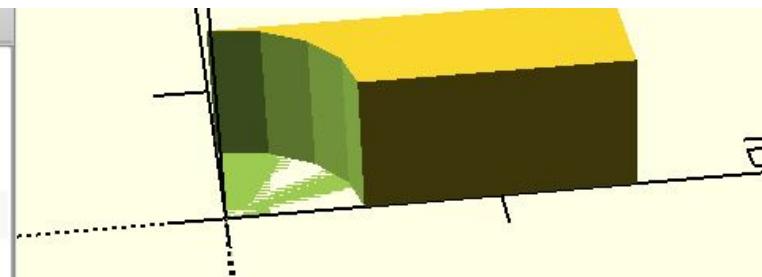
The image shows a 3D rendering of a yellow cylinder and a black cube joined together at their top edges, demonstrating the union() function.

EDITOR

```
union() {
    cube([15, 5, 5]);
    cylinder(r = 5, h = 10);
}
```

difference()

```
1 difference() {  
2     cube([15, 5, 5]);  
3     cylinder(r = 5, h = 10);  
4 }
```



intersection()



The image shows a 3D modeling application's interface. The top bar contains standard file and tool icons. Below the toolbar is a code editor window displaying the following script:

```
1 intersection() {
2     cube([15, 5, 5]);
3     cylinder(r = 5, h = 10);
4 }
```

The preview window on the right shows a 3D scene. It features a large gray cube positioned in the center-left. To its right is a smaller gray cylinder. The two objects overlap, illustrating the result of the intersection operation. The background of the preview area is yellow.

“debugging” with % and

The image shows a CAD software interface with a code editor on the left and a 3D preview on the right.

Code Editor:

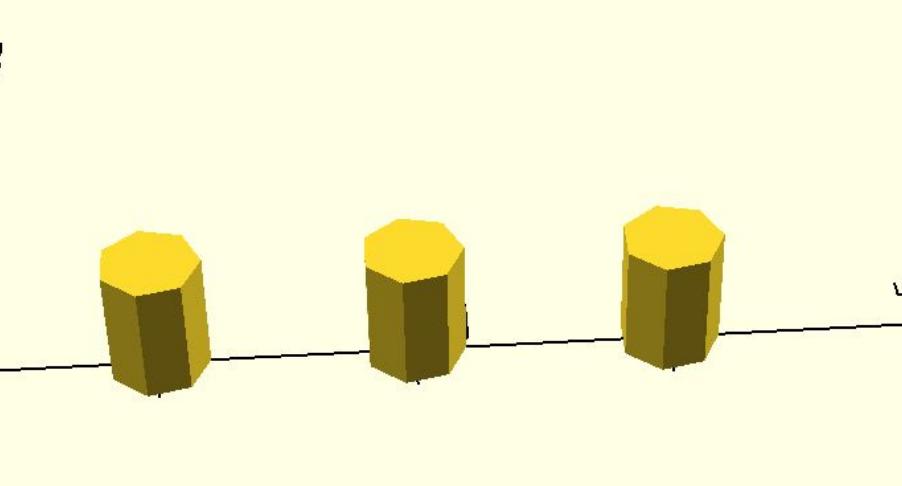
```
1 difference() {  
2     #cube([15, 5, 5]);  
3     %cylinder(r = 5, h = 10);  
4 }
```

3D Preview:

The 3D preview shows a yellow cube and a brown cylinder. The cylinder is positioned such that it appears to be inside or intersecting the cube. The background is a light yellow color.

for() and repetition

EDITOR



```
1 for(i = [1: 3]) {
2     translate([10 * i, 0, 0]) {
3         cylinder(r = 2, h = 5);
4     }
5 }
```

miscellany: \$fn

The image displays three vertically stacked screenshots of a 3D modeling application's interface. Each screenshot shows a 3D view of a yellow cylinder on a coordinate system with dashed axes.

- Screenshot 1:** Shows a cylinder with a relatively smooth surface. The code in the editor is:

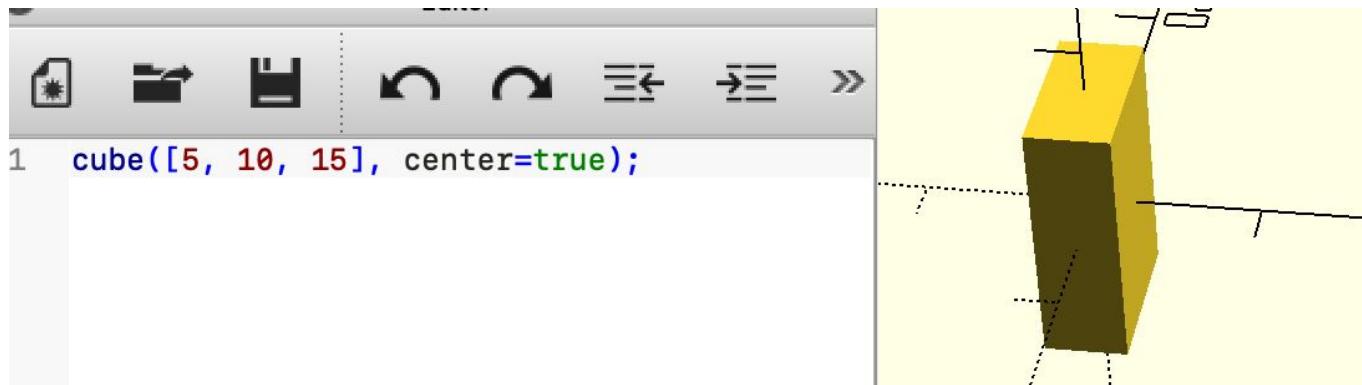
```
1 cylinder(r = 2, h = 2, $fn = 30);
```
- Screenshot 2:** Shows a cylinder with a more faceted, polygonal appearance. The code in the editor is:

```
1 cylinder(r = 2, h = 2, $fn = 10);
```
- Screenshot 3:** Shows a cylinder with a very low polygon count, appearing almost triangular. The code in the editor is:

```
1 cylinder(r = 2, h = 2, $fn = 3);
```

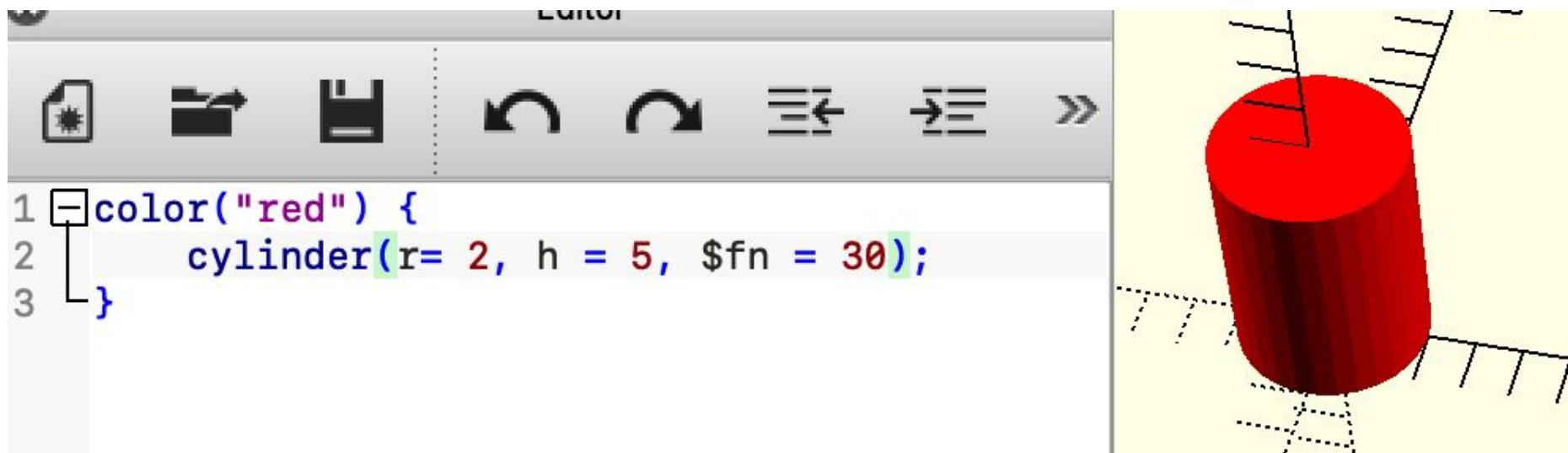
The interface includes a toolbar with icons for file operations (New, Open, Save, etc.) and navigation (Undo, Redo, Zoom). The code editor window has a line number and a syntax-highlighted line.

miscellany: center=true



miscellany: color("colorname")

Editor



```
1 color("red") {  
2     cylinder(r= 2, h = 5, $fn = 30);  
3 }
```

fun bonus: minkowski()

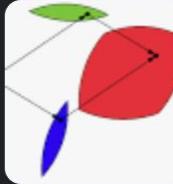
About 1,220,000 results (0.43 seconds)

<https://www.wikiwand.com> › Minkowski_addition ...

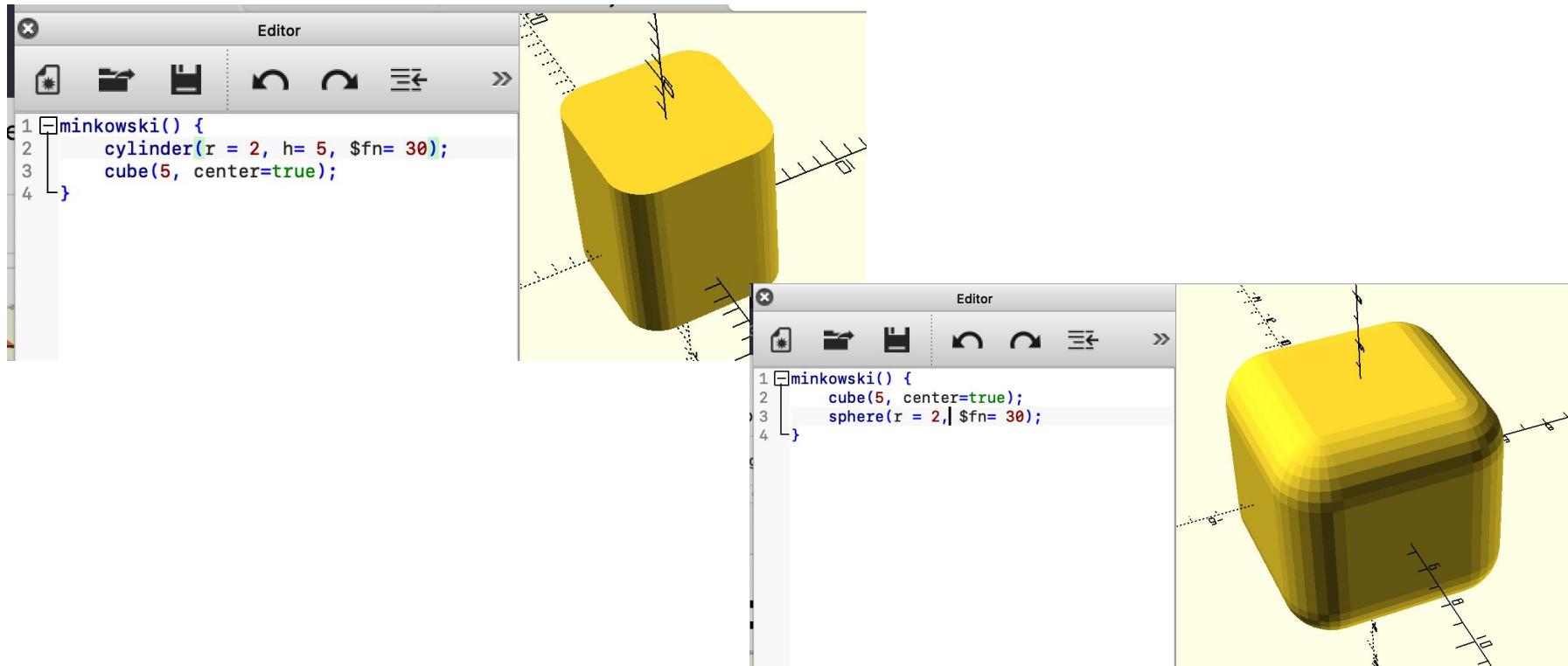
Minkowski addition - Wikiwand

In geometry, the **Minkowski sum** (also known as dilation) of two sets of position vectors A and B in Euclidean space is formed by adding each vector in A to each ...

Example · Convex hulls of Minkowski sums · Algorithms for computing...



fun bonus: minkowski()



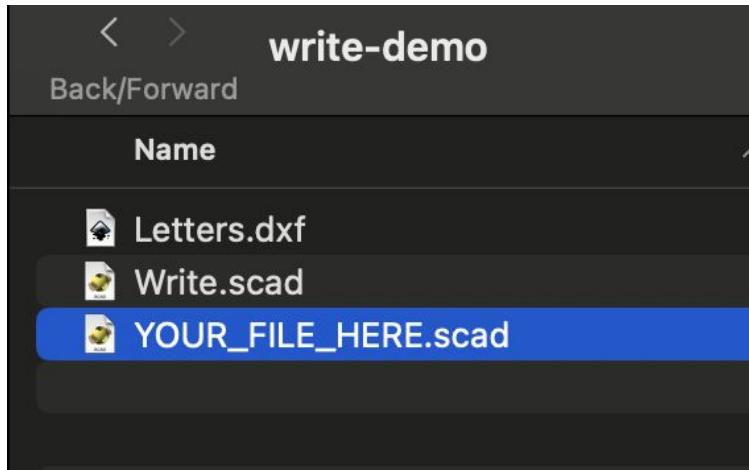
fun bonus: Write.scad

this setup is annoying; there's a ZIP in Slack!

The screenshot shows a CAD software interface. On the left is an 'Editor' window titled 'YOUR_FILE_HERE.scad'. It contains the following SCAD code:

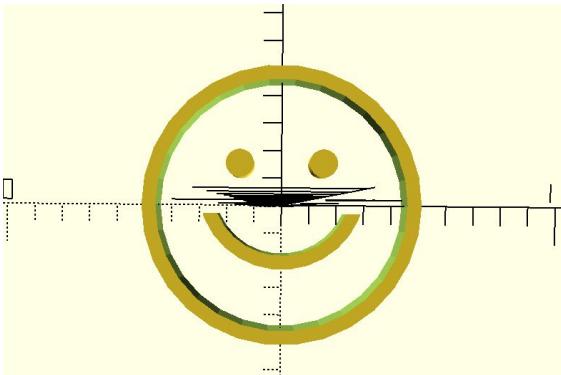
```
use <write.scad>
translate([0,30,0])
write("abcdefghijklmn",t=10.5,h=10,center=true);
```

A message at the top of the editor window states: "Compilation generated 3 warnings. For details see the [error log](#) and [console window](#)". To the right is a preview window showing a 3D rendering of the letters 'abcdefghijklmn' in yellow, centered at [0,30,0]. A coordinate system with X, Y, and Z axes is visible at the bottom.

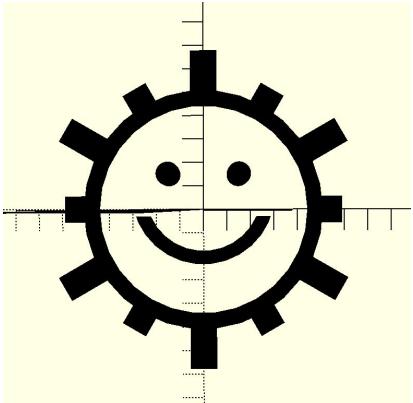


Your Challenge(s)

v1



v2



v3



<https://openscad.org/cheatsheet/>

OpenSCAD v2021.01

Syntax

```
var = value;  
var = cond ? value_if_true : value_if_false;  
var = function (x) x + x;  
module name(...) { ... }  
name();  
function name(...) = ...  
name();  
include <...>.scad>  
use <...>.scad>
```

Modifier Characters

*	disable
_	show only
#	highlight / debug
%	transparent / background

2D

```
circle(radius | d=diameter)  
square(size,center)  
square([width,height],center)  
polygon([points])  
polygon([points],[paths])  
text(t, size, font,  
     halign, valign, spacing,  
     direction, language, script)  
import("...ext", convavity)  
projection(cut)
```

Constants

```
undef undefined value  
PI mathematical constant π (~3.14159)
```

Operators

```
n + m Addition  
n - m Subtraction  
n * m Multiplication  
n / m Division  
n % m Modulo
```

3D

```
sphere(radius | d=diameter)  
cube(size, center)
```

(spoiler ahead!)

smiley-1.scad

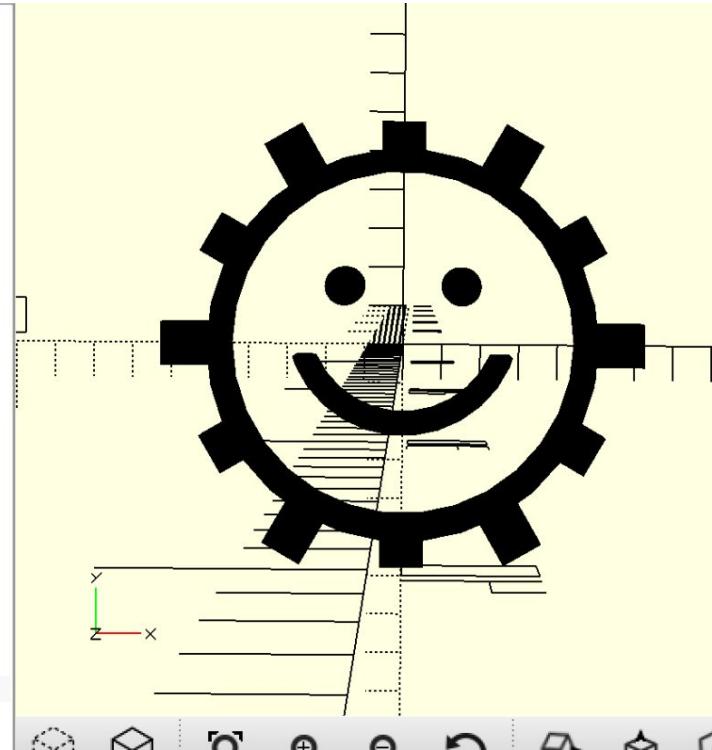
Editor

```
1 sfn=30;
2
3
4
5
6 difference() {
7     cylinder(r=5, h=2, $fn=sfn, center=true);
8     cylinder(r=4.5, h=5, $fn=sfn, center=true);
9 }
10
11 for(i = [-1, 1]) {
12     translate([1.5*i,1.5,0])cylinder(r=0.5,h=2
13     , $fn=sfn, center=true);
14 }
15
16 translate([0,0.7,0]) difference() {
17     cylinder(r=3, h=2, $fn=sfn, center=true);
18     cylinder(r=2.5, h=5, $fn=sfn, center=true);
19     translate([0,4,0]) cube([10,10,5], center=
20     true);
```

Normalized tree has 7 elements!
Compile and preview finished.

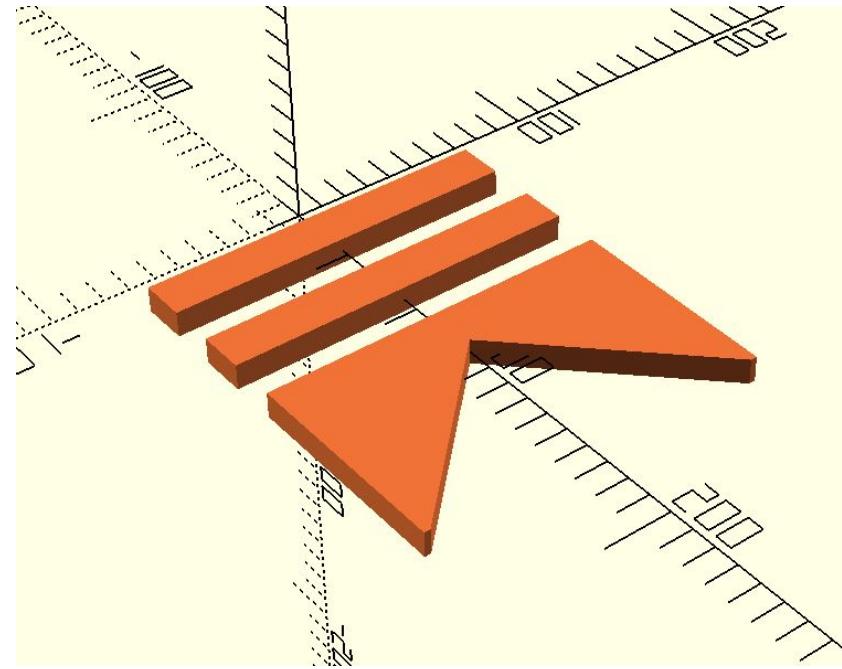
smiley-2.scad

```
1  sfn=30;  
2  
3  color("black", 1.0) {  
4      difference() {  
5          cylinder(r=5, h=2, $fn=sfn, center=true);  
6          cylinder(r=4.5, h=5, $fn=sfn, center=true);  
7      }  
8  
9      for(i = [-1, 1]) {  
0          translate([1.5*i,1.5,0])cylinder(r=0.5,h=2, $fn=sfn, center=true);  
1      }  
2  
3  
4      translate([0,0.7,0]) difference() {  
5          cylinder(r=3, h=2, $fn=sfn, center=true);  
6          cylinder(r=2.5, h=5, $fn=sfn, center=true);  
7          translate([0,4,0]) cube([10,10,5], center=true);  
8      }  
9      for(i=[0:6]) {  
0          rotate([0,0,i*60 + 30])  
1              translate([0,5.35, 0])  
2                  cube([1.12, 1.75, 1], center=true);  
3          rotate([0,0,i*60])  
4              translate([0,5.1,0])  
5                  cube([1.12, 1.25, 1], center=true);  
6      }  
7  }
```



slides from when i did this as a lunch 'n learn

Your Challenge



<https://openscad.org/cheatsheet/>

OpenSCAD v2021.01

Syntax

```
var = value;
var = cond _ value_if_true ± value_if_false;
var = function (x) x + x;
module name(...) { ... }
name();
function name(...) = ...
name();
include <...>.scad>
use <..>.scad>
```

Modifier Characters

<u>*</u>	disable
<u>!</u>	show only
<u>#</u>	highlight / debug
<u>%</u>	transparent / background

2D

```
circle(radius | d=diameter)
square(size,center)
square([width,height],center)
polygon([points])
polygon([points],[paths])
text(t, size, font,
      halign, valign, spacing,
      direction, language, script)
import("...ext", convexity)
projection(cut)
```

Constants

```
undefined undefined value
PI mathematical constant π (~3.14159)
```

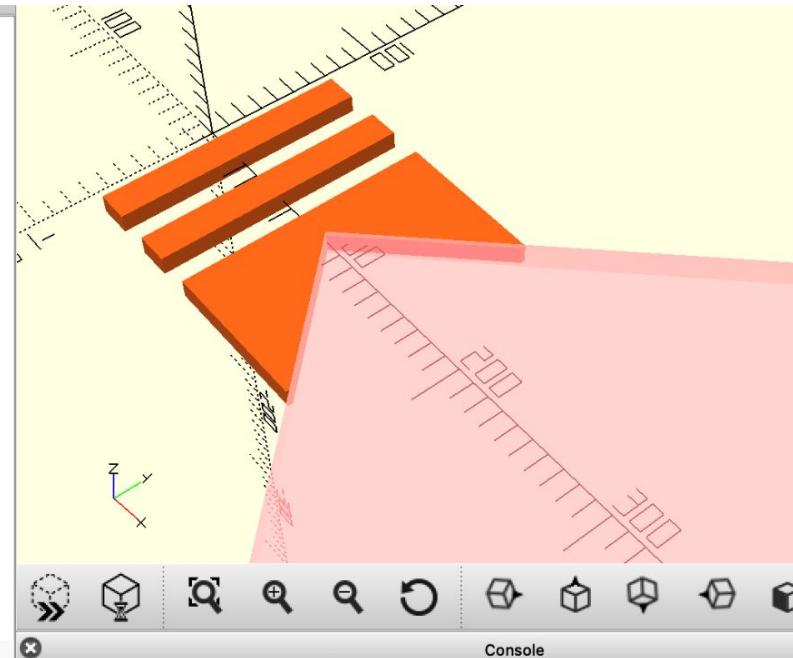
Operators

<u>n + m</u>	Addition
<u>n - m</u>	Subtraction
<u>n * m</u>	Multiplication
<u>n / m</u>	Division
<u>n % m</u>	Modulo

3D

```
sphere(radius | d=diameter)
cube(size, center)
```

```
1  logoWidth = 136;
2  barThickness = 18;
3  depth = 10;
4  bottomLength = 84;
5
6 color("#ed7135") difference() {
7   union() { // group first before subtracting stuff
8     //the two top things
9     for(i = [1, 3]) { //renders two; one at i = 3, one at i = 3
10       translate([ barThickness*i, 0, 0]) {
11         cube([barThickness, logoWidth, depth], center = true); // bars
12       }
13     }
14   translate([barThickness*4.5 + bottomLength/2, 0, 0]) {
15     // bottom thick rectangle we cut the notch out of
16     cube([bottomLength,logoWidth, depth], center=true);
17   }
18 }
19 // this is what we are cutting out of the top
20 translate([barThickness*4.5 + bottomLength*1.7 + logoWidth/2, 0,0]) {
21   rotate([0,0,45]) { // rotate 45 so we get the corner
22     // this is the notch
23     #cube([logoWidth*2, logoWidth*2, depth+5], center=true);
24   }
25 }
26 }
27 }
```



[BACK TO CUSTOMIZABLE GARDEN SIGN](#)

Now Using: Customizer

Queue

▼ let-s-grow-something-awesome

🔗 <https://www.thingiverse.com/ap>

Copy

[View Source](#)

[Create Thing](#)

Label Text What are we planting?

Substack

Font

Letters



Build Plate Selector For display only, not part of final model

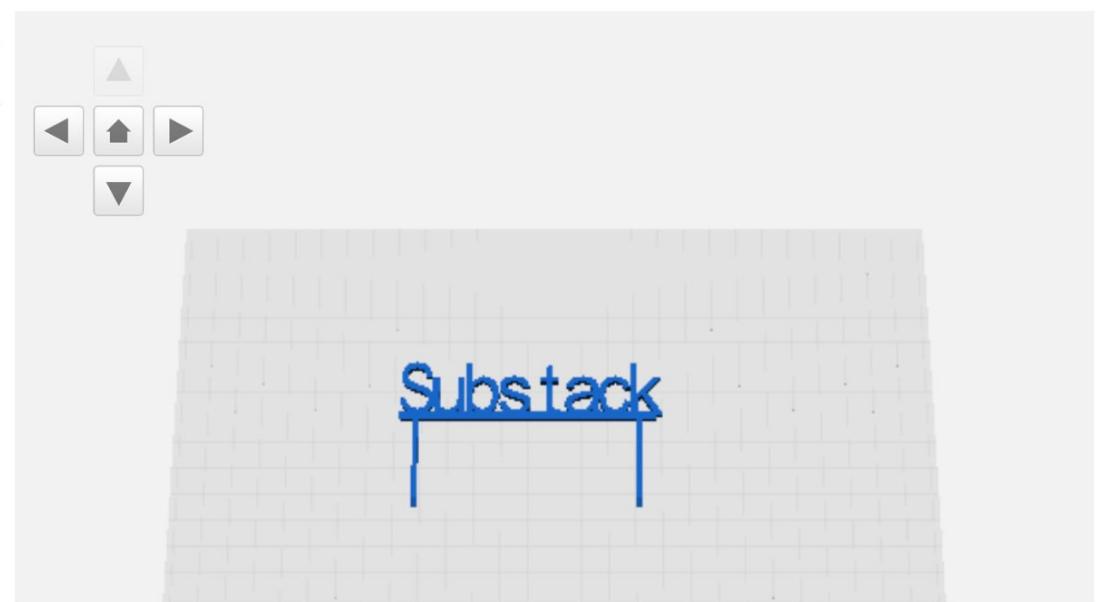
Replicator 2/2X

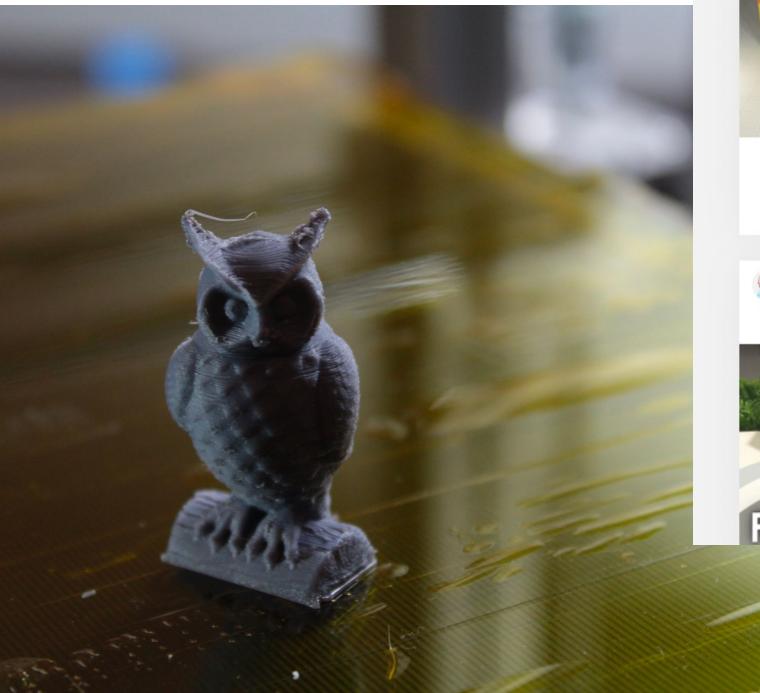
Build Plate Manual X "Manual" build plate X dimension

10

Build Plate Manual Y "Manual" build plate Y dimension

10





MakerBot Thingiverse

Popular Last 30 Days

All Things

Filter By

Stackable Storage Caddies

No supports required! Fully Customizable

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Filament Machine II - Petamentor2

PETamentor²

FILAMENT MACHINE II

Vase #650

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