Parametric Model Design for 3D Printing

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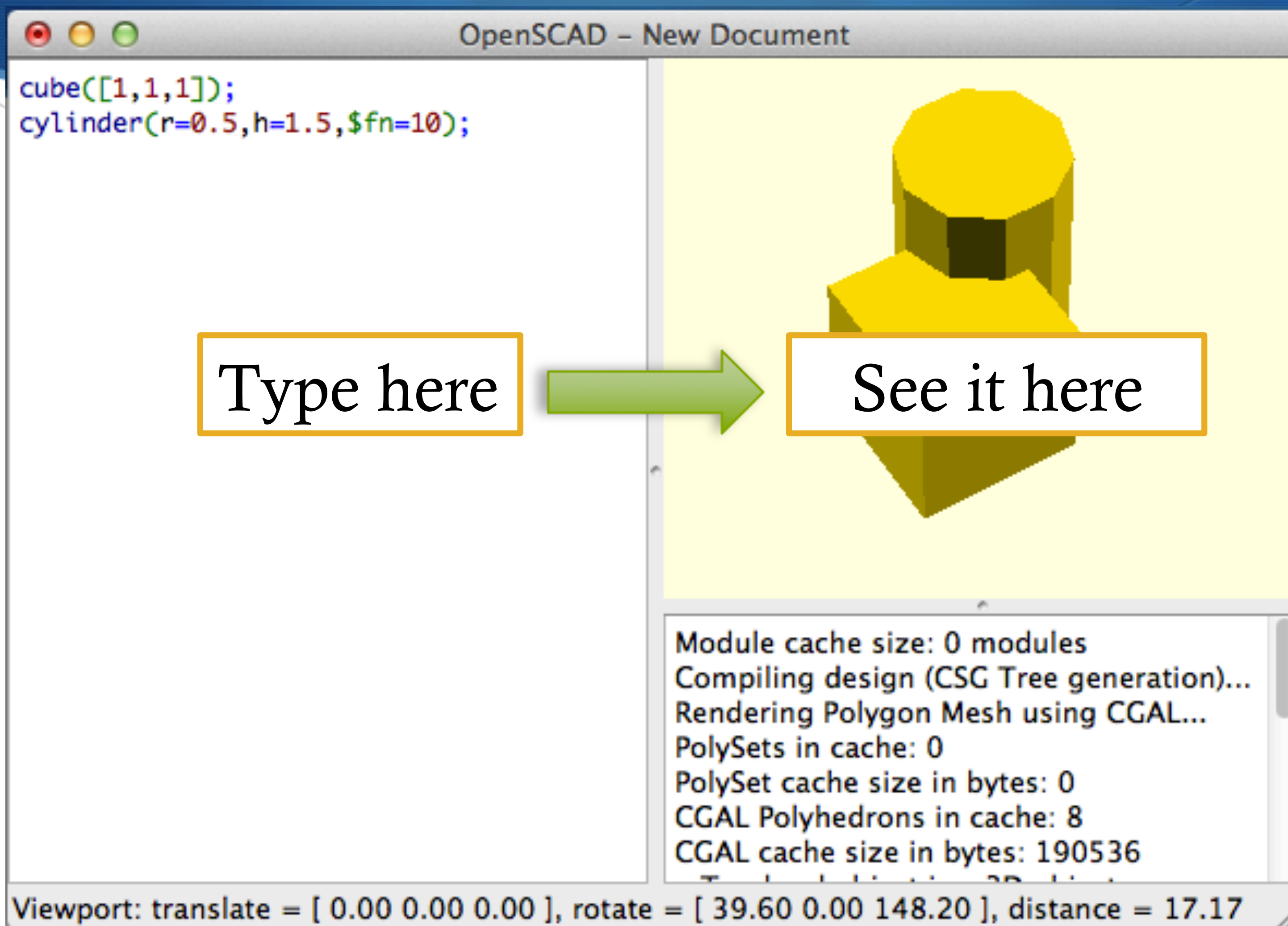
Maker Group

Examples & Presentation

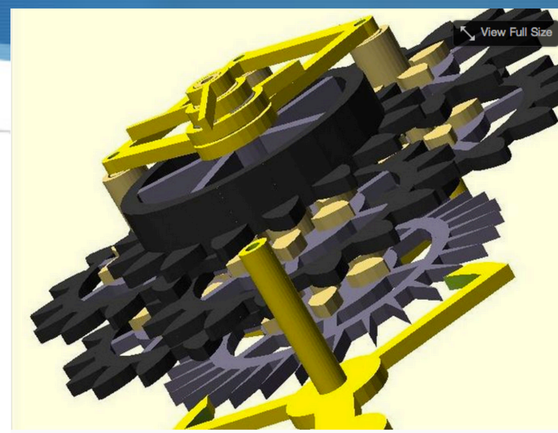
- The examples and this presentation can be found here:
  - http://github.com/celer/3d-things
  - You can download a zip file from here or install git
- Install git http://git-scm.com/downloads
  - Then clone the repo
    - git clone git://github.com/celer/3d-things.git
  - Quick intro to git
    - http://git-scm.com/videos
    - http://git-scm.com/book/en/Getting-Started

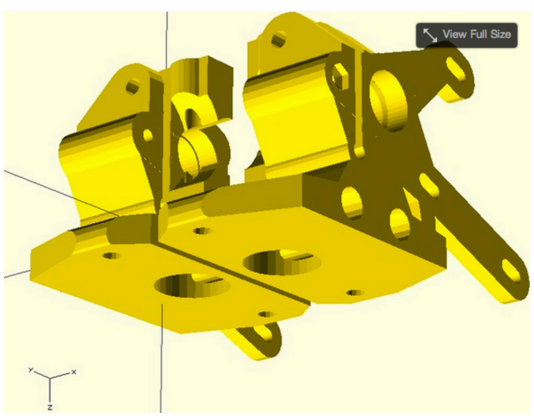
An introduction to OpenSCAD

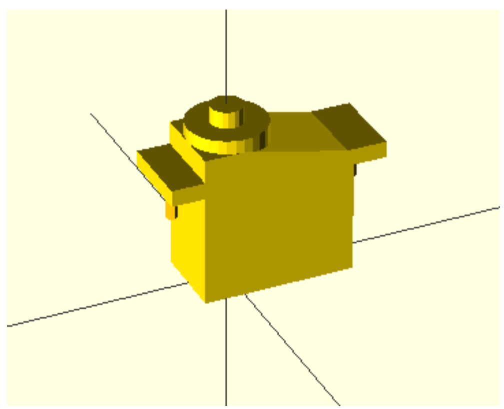


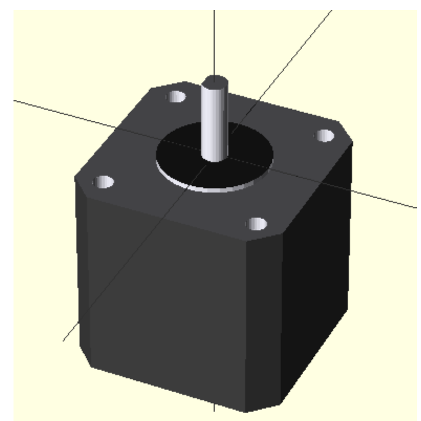
OpenSCAD is amazing!

(These are random things from thingiverse)





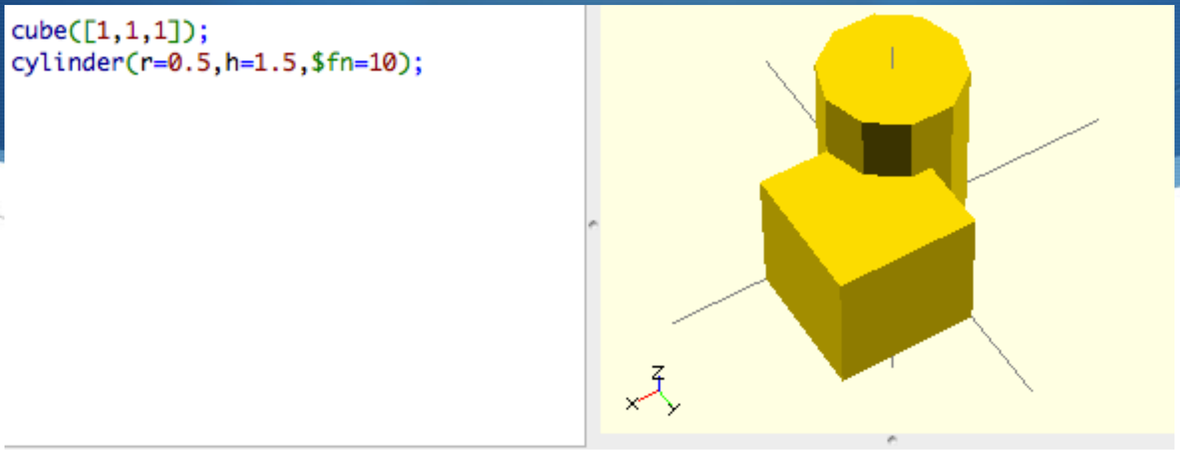




Diving in!

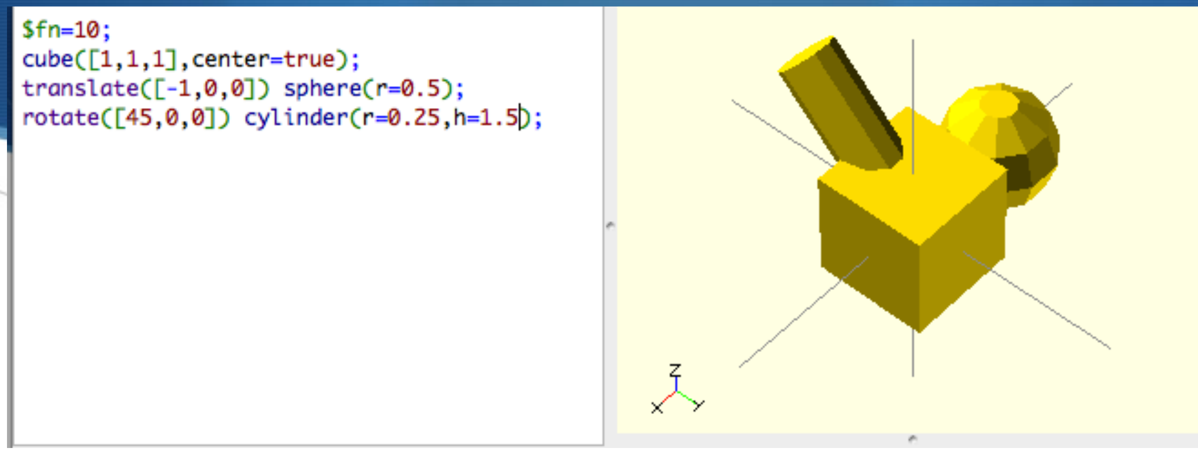
- www.openscad.org
  - It's free!
  - It works on every major platform
  - Produces dimensionally accurate designs (by default all units are MM)
- To use it for 3D Printing
  - Type in your design
  - Render it
  - Save it to a .STL file
  - Use a Slicer to convert the .STL file to a .GCODE File
  - Print!

Shapes



- You type in the shape's you want
  - cube[x-dimension,y-dimension,z-dimension]);
  - cylinder(r=radius,h=height)
  - sphere(r=radius)
- You can modify the shapes by adding:
  - center=true - center the shape
  - $fn=fineness – adjust the fineness of the generated shape
  - etc

Modifying Shapes



- You can modify a shape using these commands
  - translate([moveByX,moveByY,moveByZ])
  - scale(newScale) or scale([scaleX,scaleY,scaleZ])
  - rotate([rotateX,rotateY,rotateZ])
- Modify a shape by placing the command before it or by putting the shapes in braces ‘{}’

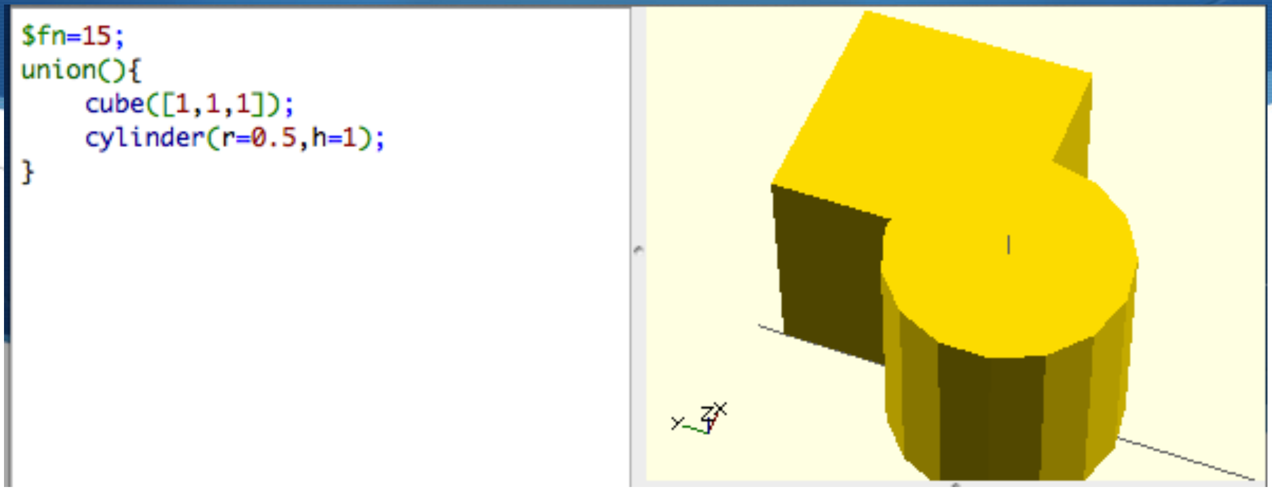
How to select what to modify

- Modify a single shape
  - rotate([45,0,0]) translate([1,0,0]) cube([1,1,1]);
    - You can chain modifications
    - The order matters! (move then rotate != rotate then move)
      - First translate (move)
      - Then rotate
- Modify a collection of stuff

```
rotate([45,0,0]) translate([1,0,0])  {
    cube(1,1,1);
    scale(1.5) sphere(r=2);
}
```

- Operations are applied in reverse order
  - sphere(r=2) is first scaled, then translated and finally rotated

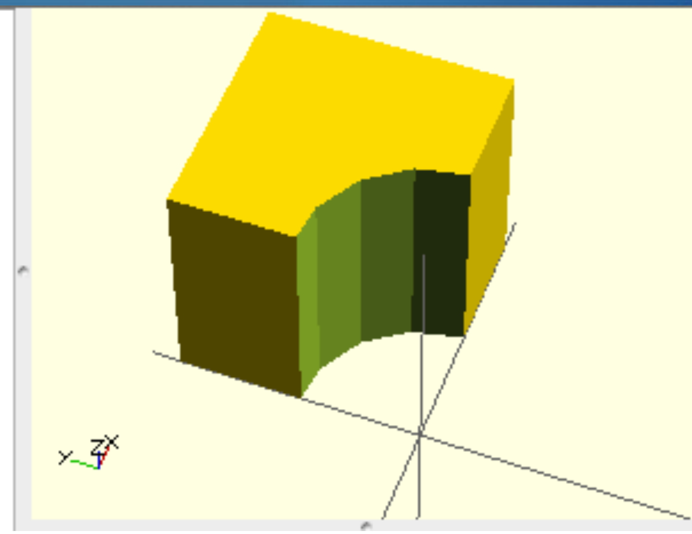
Union



- union(){ }
  - Produce a shape by combining all the shapes

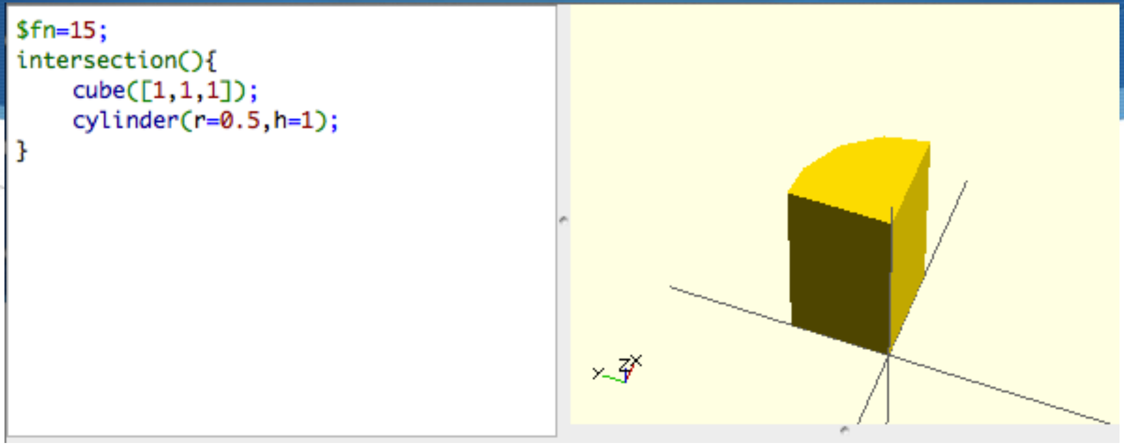
Difference

```
$fn=15;
difference(){
    cube([1,1,1]);
    cylinder(r=0.5,h=1);
}
```



- You can perform operations on shapes to create new shapes
  - difference(){ shapeA(); shapeB(); shapeC(); }
    - Subtract shapes from each other
    - shapeA-(shapeB+shapeC)

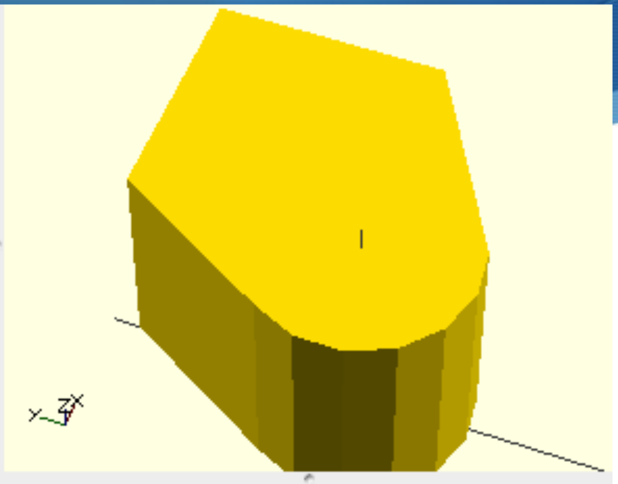
Intersection



- intersection(){ }
  - Product a new shape from the intersection of shapes

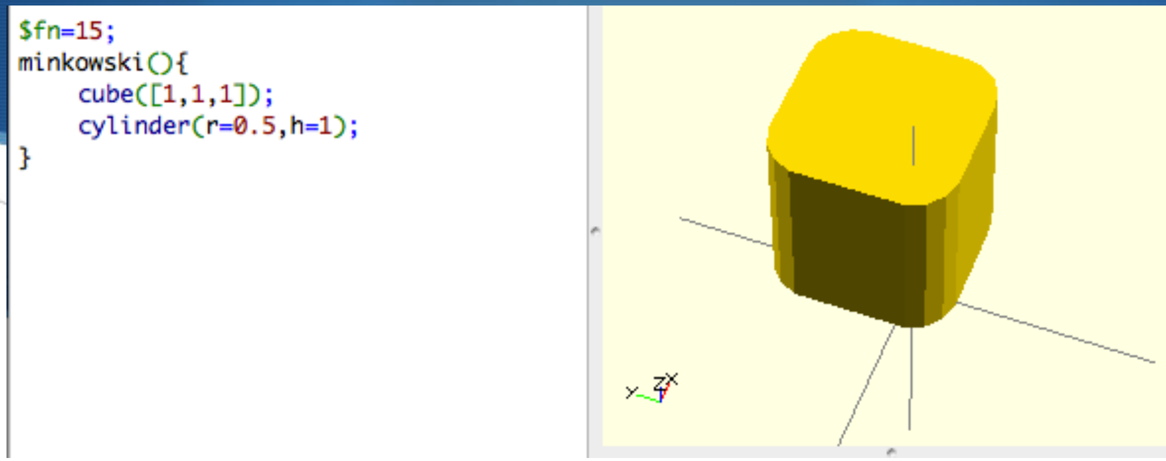
Hull

```
$fn=15;
hull(){
    cube([1,1,1]);
    cylinder(r=0.5,h=1);
}
```



- hull(){ }
  - Produce a shape by combining the profiles of two shapes

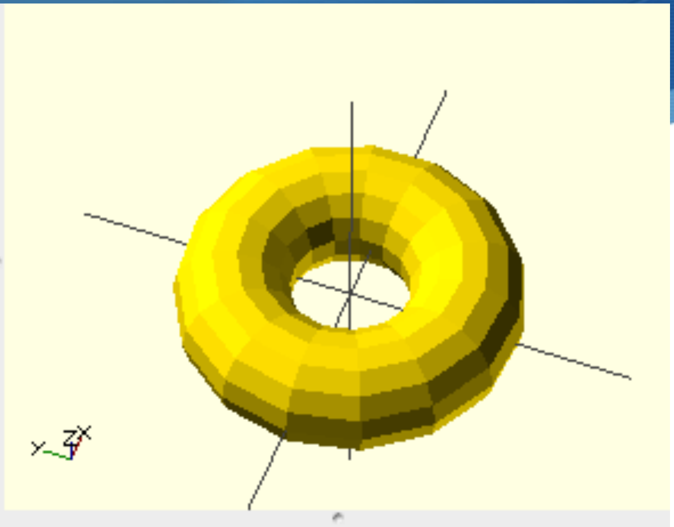
Minkowski



- minkowski(){ }
  - Produce a shape by tracing one shape around another
    - This will trace the cylinder around the cube

Rotate & Extrude

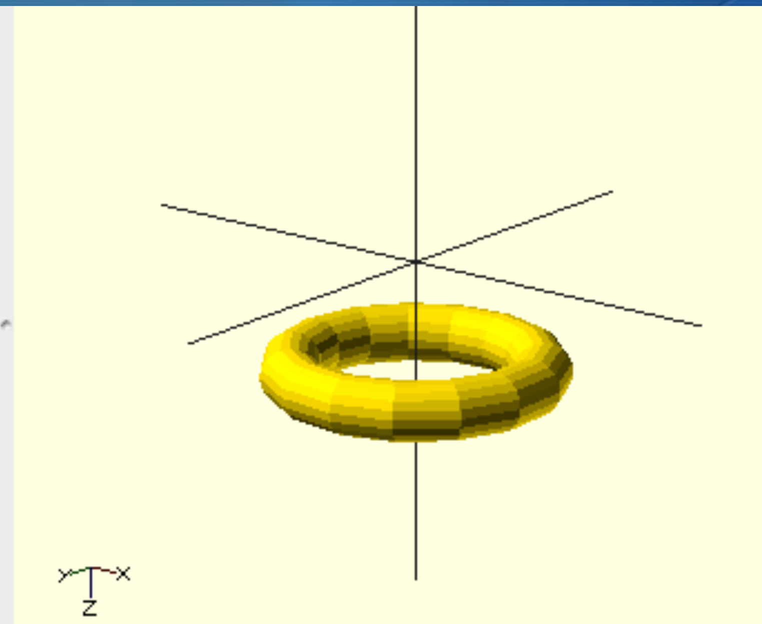
```
$fn=15;
rotate_extrude(convexity = 10)
translate([2, 0, 0])
circle(r = 1);
```



- rotate_extrude(){ }
  - Produce a shape rotate it and extruding it

Modules

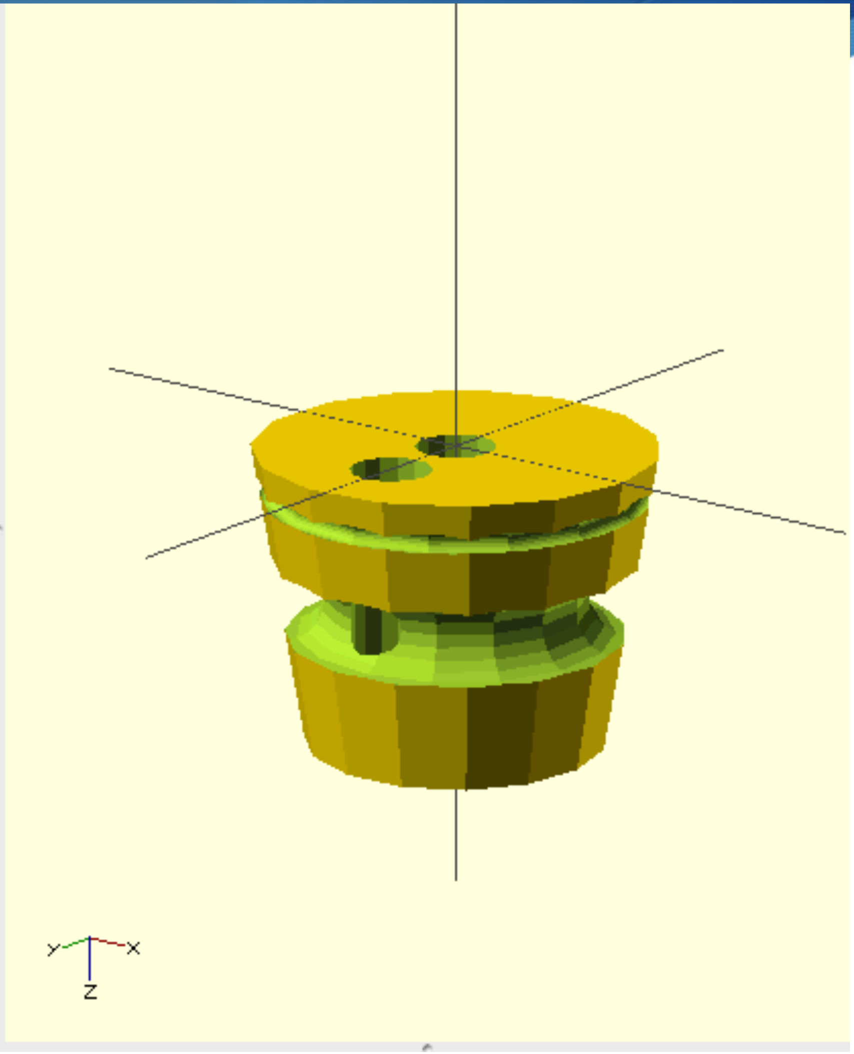
```
$fn=15;

module groove(){
    translate([0,0,15]){
        rotate_extrude(convexity = 10)
        translate([17, 0, 0 ])
        circle(r = 4,$fn=20);
    }
}

groove();
```



- Modules let you combine a bunch of shapes and operations into a single thing so you can re-use.
    - They also let OpenSCAD cache a shape

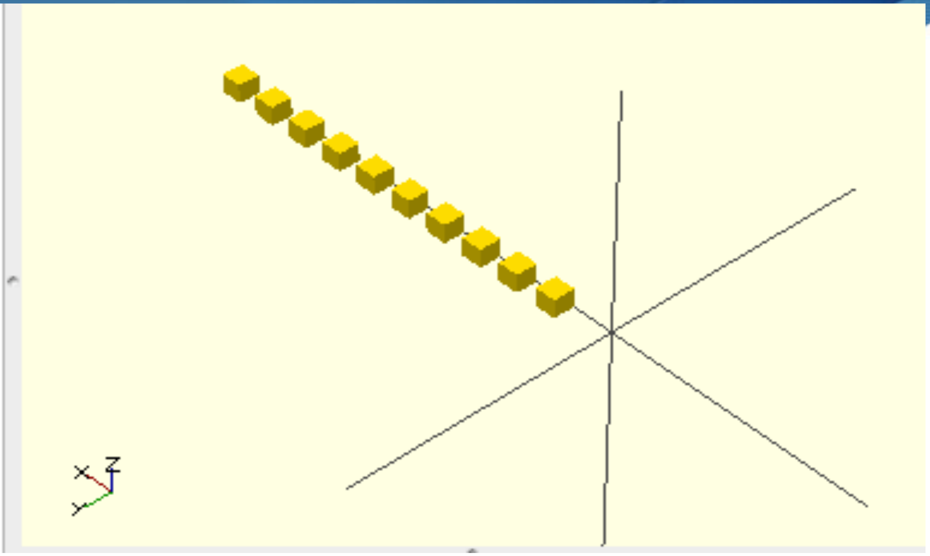
Composite shapes!

```
$fn=15;

module plug(){
    difference(){
        cylinder(r1=20,r2=15,h=30);
        union(){
            groove();
            oring();
            translate([0,10,-1]) cylinder(r=4,h=22);
            translate([0,0,-1]) cylinder(r=4,h=35);
        }
    }
}

module groove(){
    translate([0,0,15]){
        rotate_extrude(convexity = 10)
        translate([17, 0, 0 ])
        circle(r = 4,$fn=20);
    }
}

module oring(){
    translate([0,0,4]){
        rotate_extrude(convexity = 10)
        translate([19, 0, 0 ])
        circle(r = 1,$fn=20);
    }
}

plug();
```



Loop & Iterate

```
$fn=15;

for(i=[1:10]){
    translate([i*2,0,0]) cube([1,1,1]);
}
```



- OpenSCAD has variables
  - i=5; //set I to 5
- You can loop or iterate with openscad
  - for(variable=[start:increment:end]){
  - }

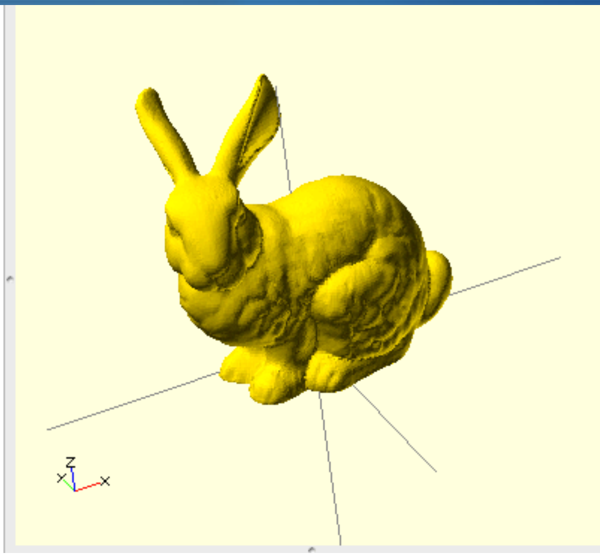
3D IO

The power is in combinations

- Inputs
  - 3D Model (.OBJ, .STL)
    - Thingiverse, Sculptris, Wings3D, 123D Catch, etc
  - 2D Models / Drawings (.DXF, .SVG, PD)
    - Inkscape, Autocad, QCAD, Desktop Scanner, etc
- OpenSCAD can import and use these files
  - import(“kitten.obj”);
- You can then use them like any other shape!

Importing things!

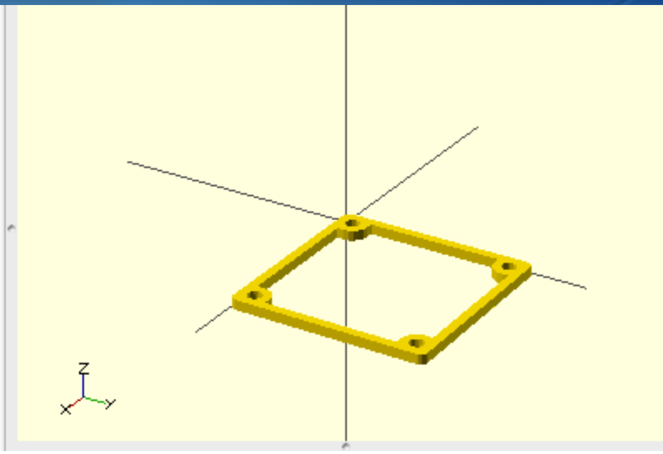
```
import("/Users/dtyree/Downloads/bunny-flatfoot.stl");
```



```
linear_extrude(height=5){
    import("/Users/dtyree/Downloads/smiley.dxf");
}
```

Using libraries

```
//uncomment this for example
fan_mount(size=60,thick=3);

module fan_mount(size=40,thick = 4)
{
if(size == 25)
    {
    _fan_mount(
            fan_size = 25,
            fan_mounting_pitch = 20,
            fan_m_hole_dia = 3,
            holder_thickness = thick
        );
    }
if(size == 30)
    {
    _fan_mount(
```

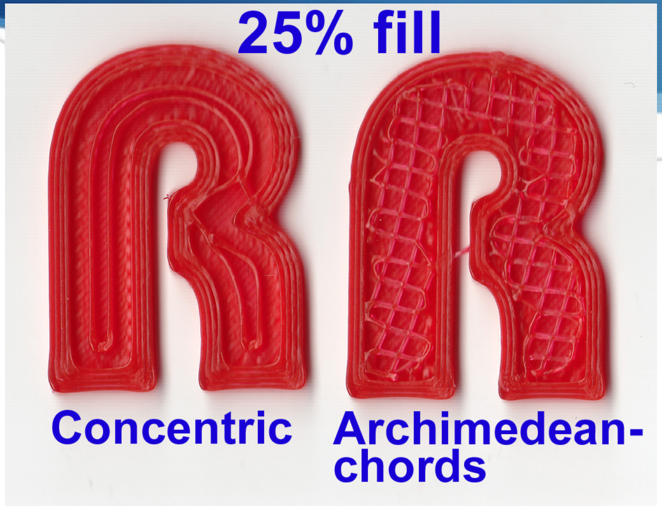


- OpenSCAD has a large number of libraries
  - MCAD (https://github.com/elmom/MCAD)
    - Screws, gears, servos, steppers, motors, bolts, etc
  - Thingiverse
- Download the library and then import it
  - Usually the library will have instructions inside of how to use it

Designing stuff!

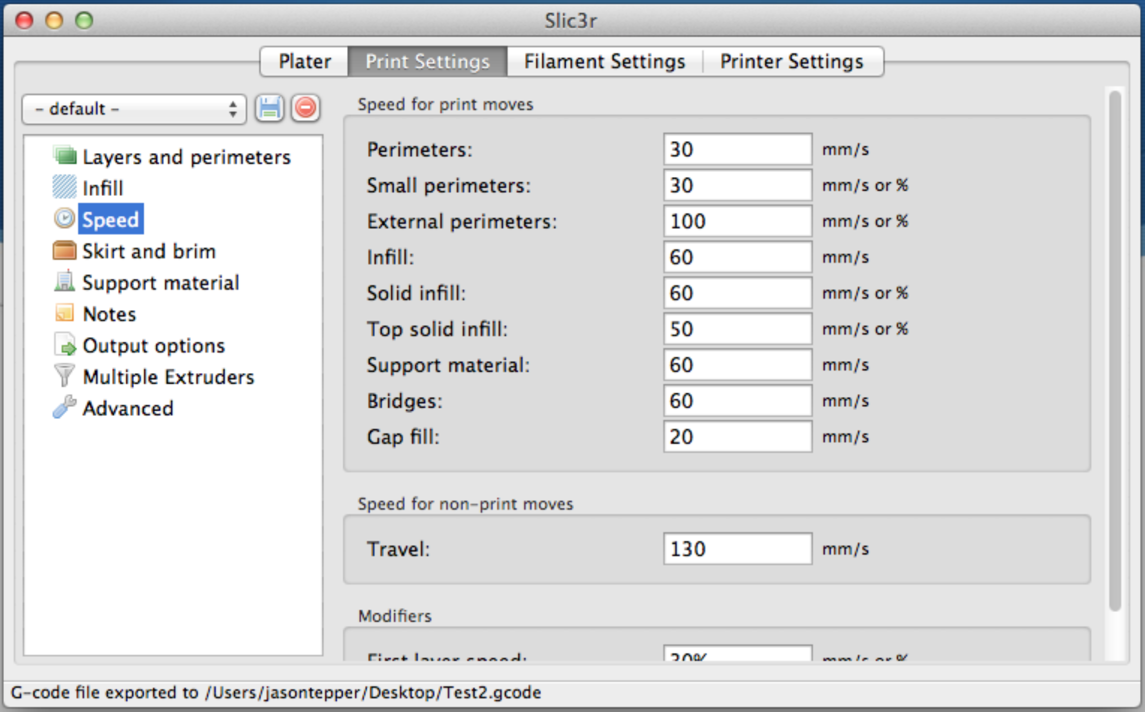
- What requirements?
- What tools / inputs do I have ?
  - Capture from real life
    - Desktop Scanner / 123D Catch
  - Model it
    - Sculptris, Thingiverse, Sketchup
- Can I print it?
  - Does it have big overhangs?
  - How dense does it need to be?
  - How accurately must it be printed to work (0.3mm is what I consider the minimum feature size to be useful)

Slicing



- After creating the thing we choose some print settings

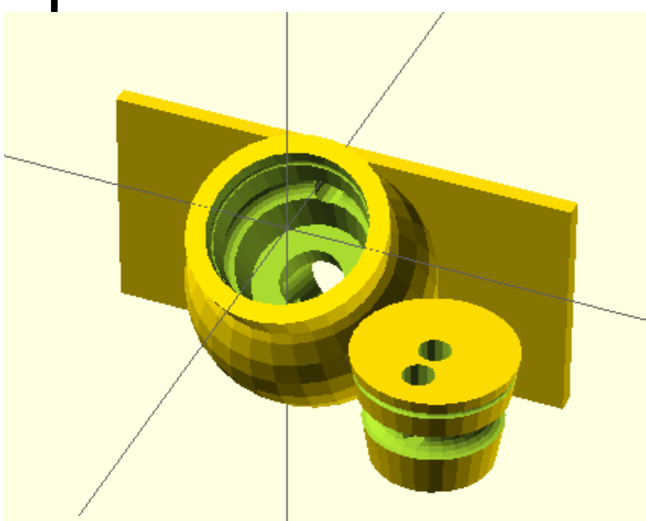
Slicing



- After designing the model it is time to print it! (I use Slic3r)
  - Choose a density and fill pattern
  - Choose number of top / bottom layers
  - Choose number of perimeter
- Print the generated .GCODE file (I use Printrun)

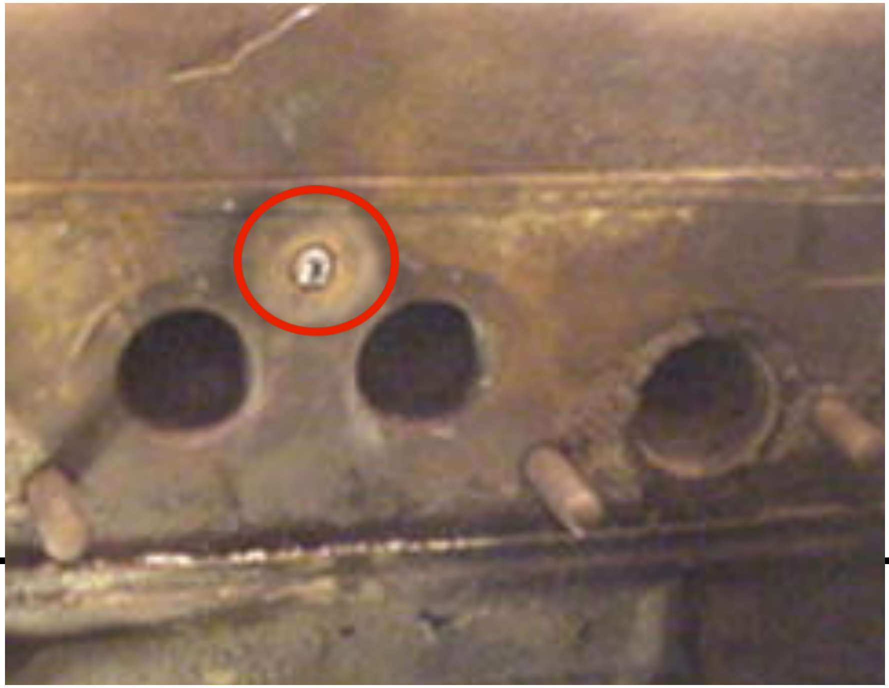
Example: Quick disconnect for cool suit for racing (Drawing it out on paper is a must for this one)

- A hose connector for quickly disconnecting hoses for a cool suit, which can be mounted on the side of a cooler
  - Must disconnect quickly and easily
  - Must plug back in easily
  - Must not let it pump water out of the cooler
  - Must be printable



Example: A custom JIG for drilling out a snapped bolt on a motor

- You want to build a JIG to drill out a snapped exhaust bolt.
  - Use a desktop scanner to scan a new gasket for the exhaust
  - Convert the scanned image to a .SVG/.DXF file
  - Extrude it with OpenSCAD



Example: A Fallout PitBoy

- You want to build a Fallout Pitboy
  - You can use 123D catch to acquire a model of your wrist
  - You can then use a 3D model of the PitBoy and subtract the model of your wrist from it.