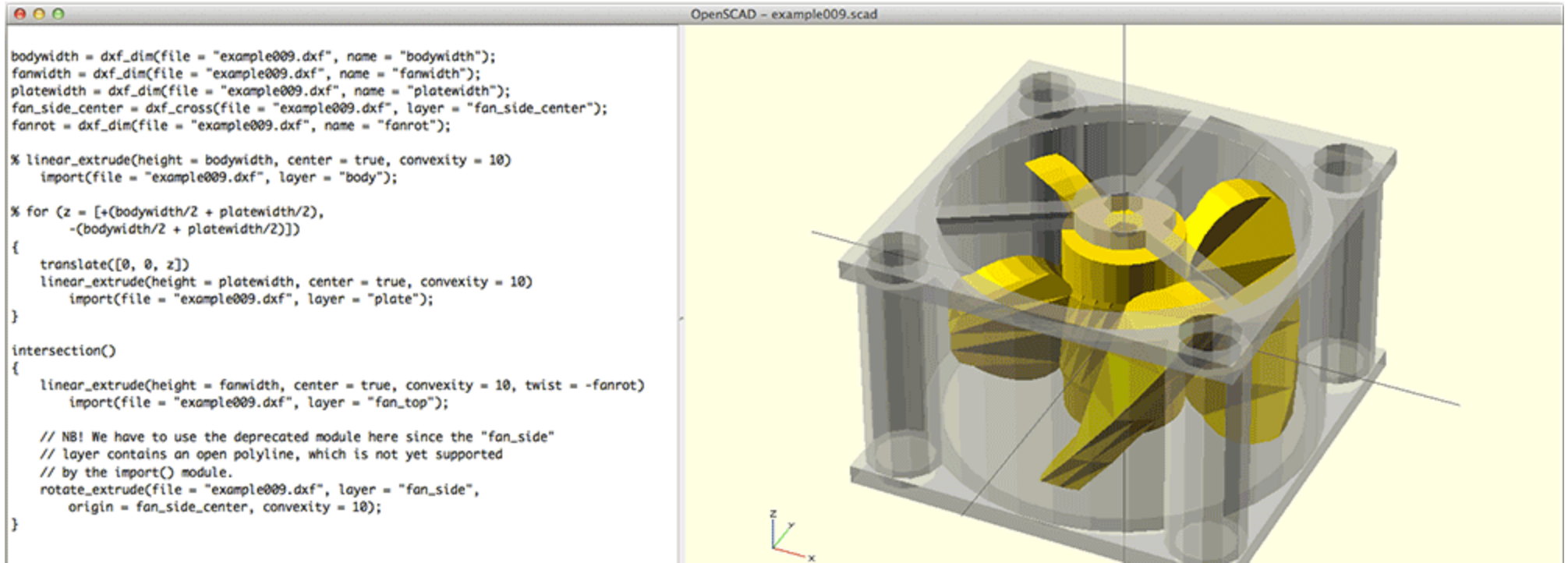
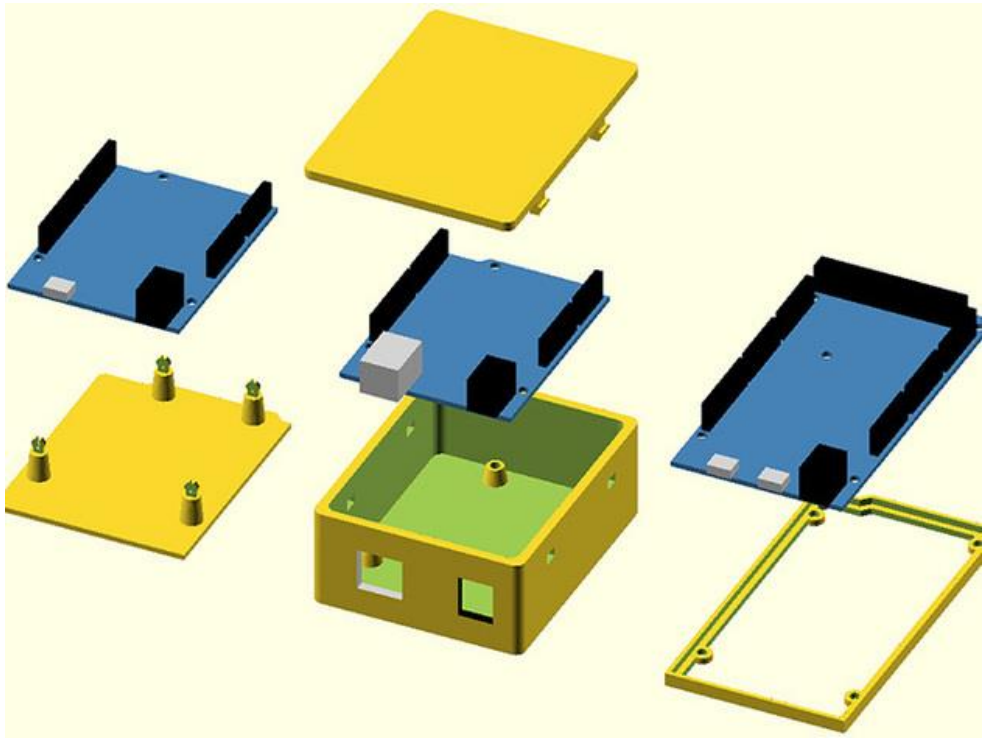
OpenScad


OpenSCAD – example009.scad
bodywidth = dxf_dim(file = "example009.dxf", name = "bodywidth");
fanwidth = dxf_dim(file = "example009.dxf", name = "fanwidth");
platewidth = dxf_dim(file = "example009.dxf", name = "platewidth");
fan_side_center = dxf_cross(file = "example009.dxf", layer = "fan_side_center");
fanrot = dxf_dim(file = "example009.dxf", name = "fanrot");
% linear_extrude(height = bodywidth, center = true, convexity = 10)
import(file = "example009.dxf", layer = "body");
% for (z = [+(bodywidth/2 + platewidth/2),
-(bodywidth/2 + platewidth/2)])
{
translate([0, 0, z])
linear_extrude(height = platewidth, center = true, convexity = 10)
import(file = "example009.dxf", layer = "plate");
}
intersection()
{
linear_extrude(height = fanwidth, center = true, convexity = 10, twist = -fanrot)
import(file = "example009.dxf", layer = "fan_top");
// NB! We have to use the deprecated module here since the "fan_side"
// layer contains an open polyline, which is not yet supported
// by the import() module.
rotate_extrude(file = "example009.dxf", layer = "fan_side",
origin = fan_side_center, convexity = 10);
}
z
y
x

- Why use it?
  - Free!
  - Large community: http://forum.openscad.org/
  - Easier to get started than other softwares;

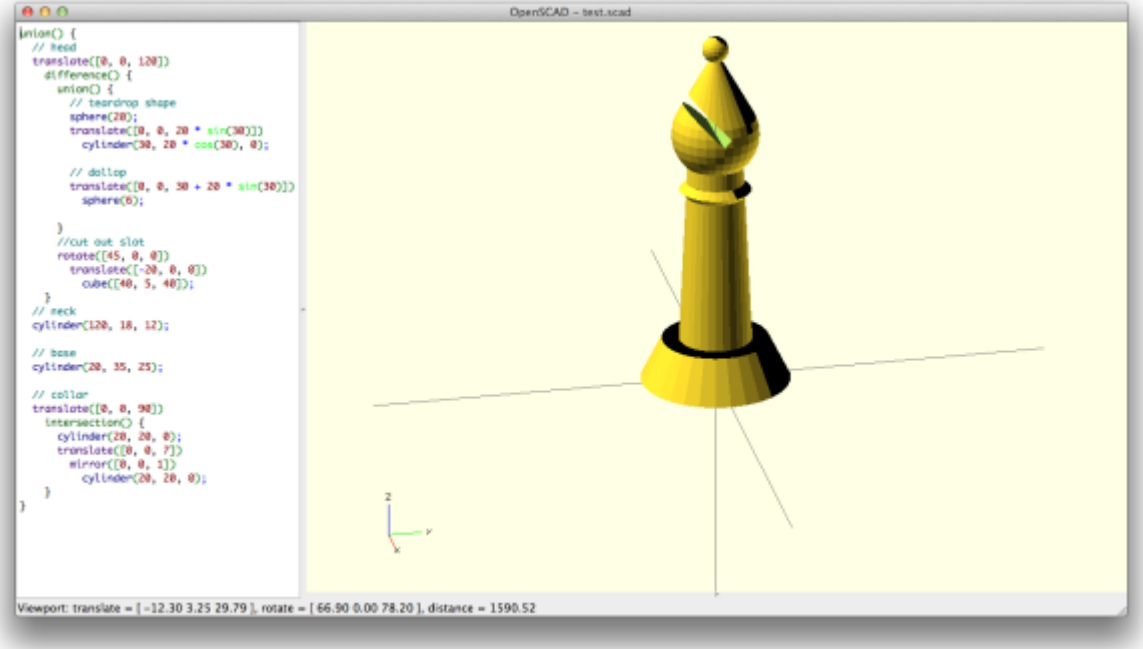


The Basics

- Geometric Figures
  - Cube
  - Sphere
  - Cylinder
- Transformations
  - Translating
  - Scaling
  - Rotating
  - Mirroring

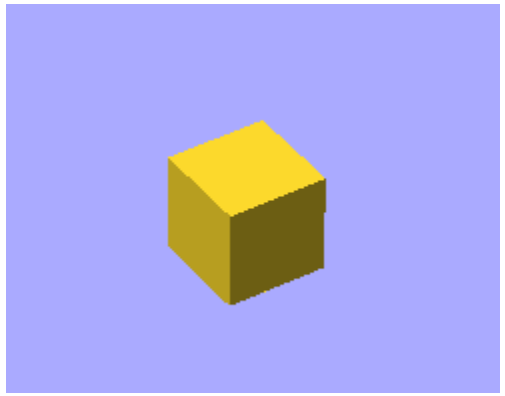
CSG

- Constructive Solid Geometry
  - Union
  - Difference
  - Intersection

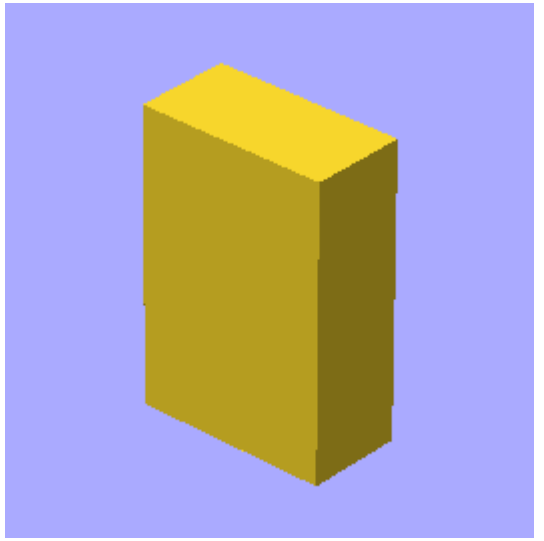


The Basics

- Cube
  - $ cube([x,y,z]);
  - Cube([10,10,10]);

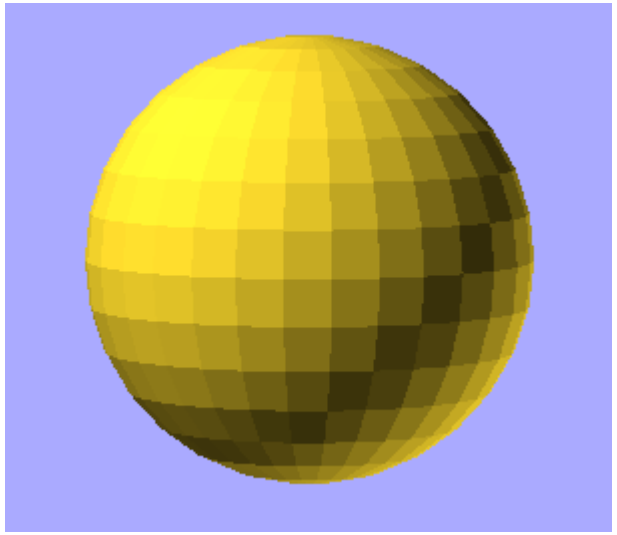


cube([10,20,30], center = true);

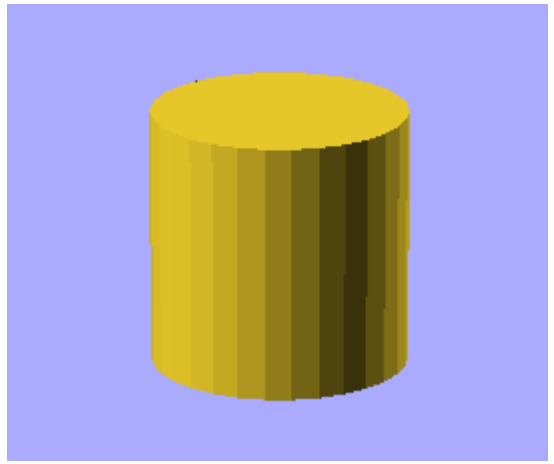


The Basics

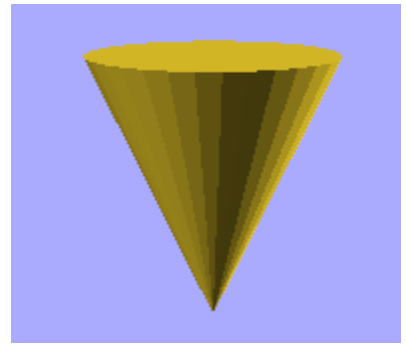
- Sphere
  - $ sphere(r);
  - sphere(r=20); // already centered
  -



The Basics

- Cylinder
  - $ cylinder(h,r1,r2);
  - Cylinder(h=20,r=10);
  - 

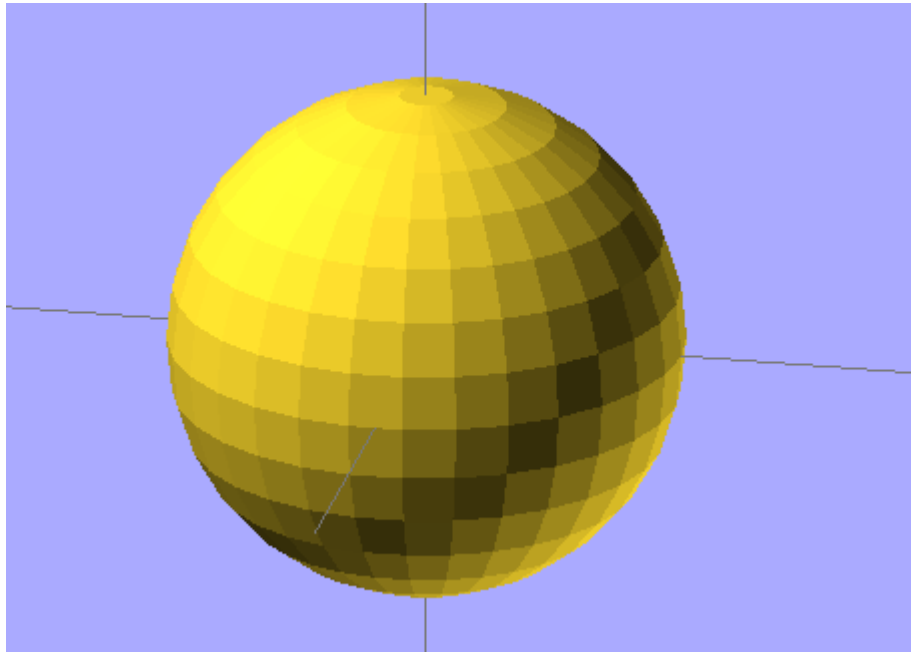
Cylinder(h=20,r=10, r2=0, center = true);



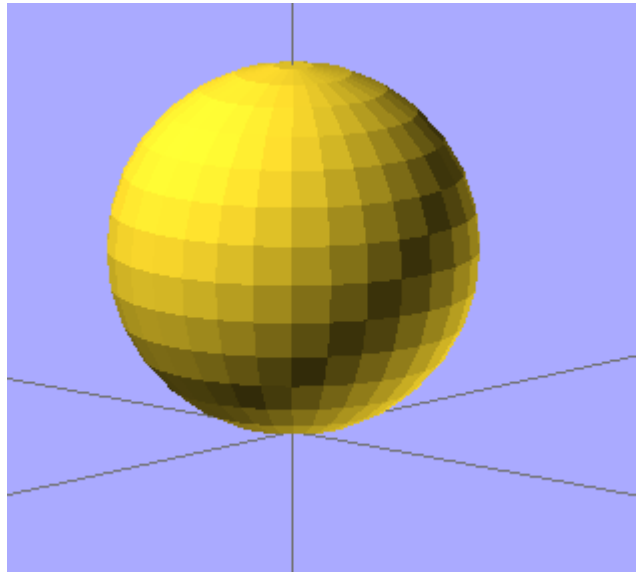
Transformations

- Translation => Move to somewhere else;

- Sphere(r=20);



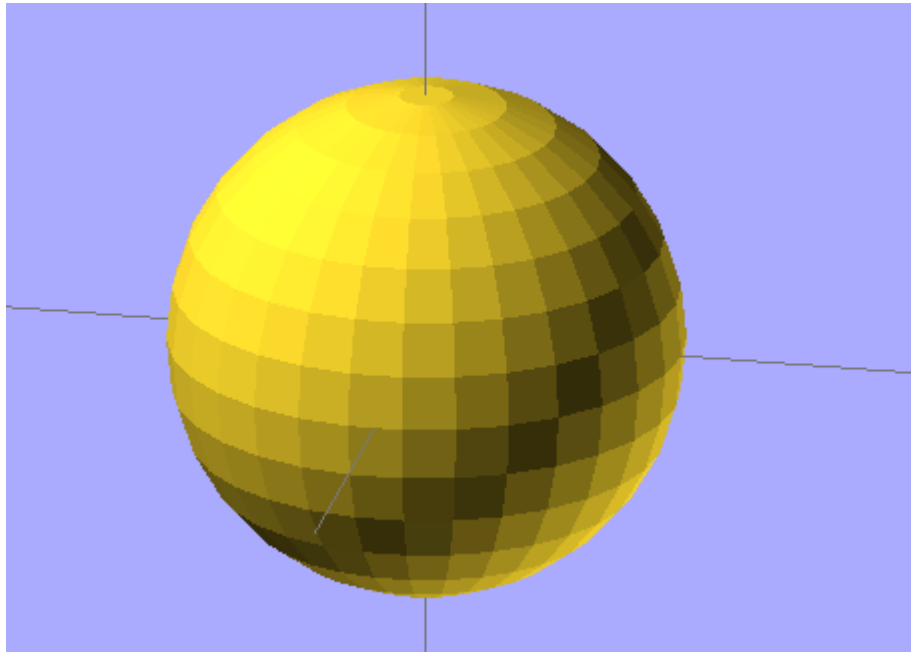
translate([0,0,20] Sphere(r=20);



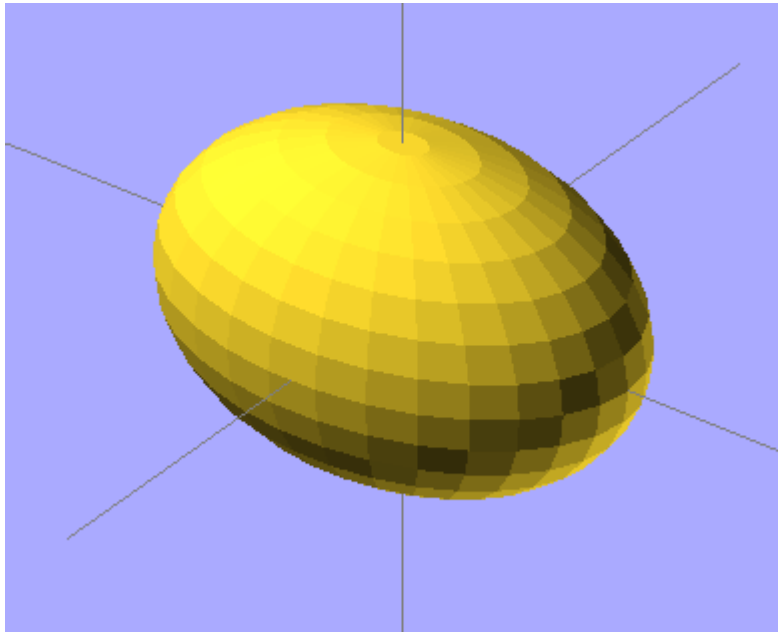
Transformation

- Scaling=> multiply for some value;

- Sphere(r=20);



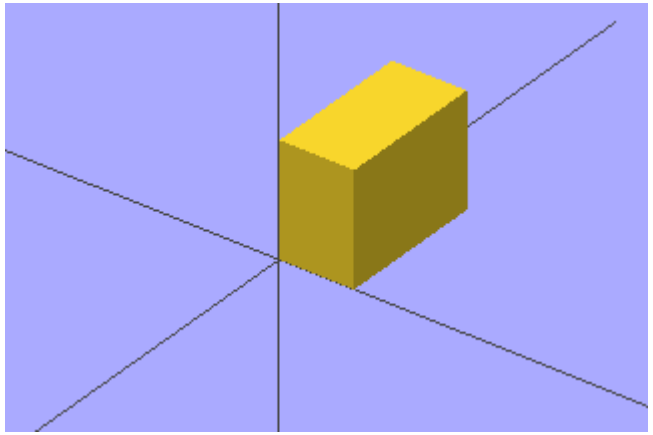
scale([1.5,1,1] Sphere(r=20);



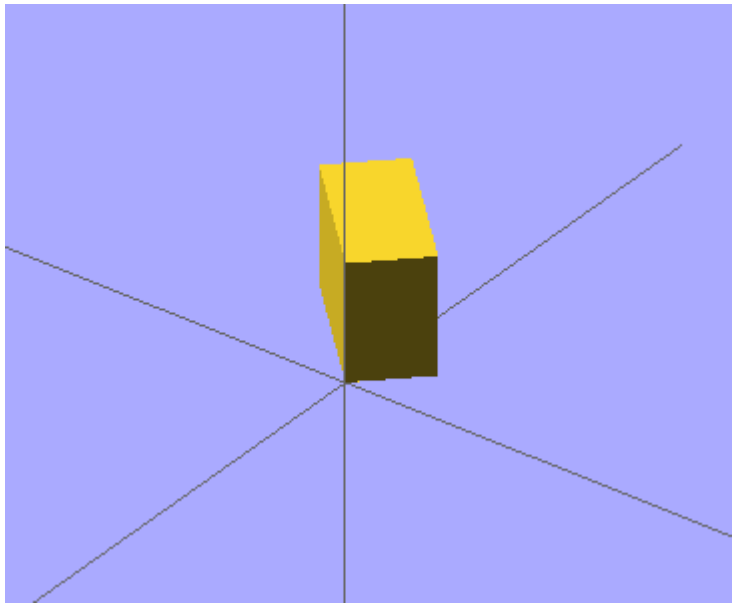
Transformation

- Rotate=> change position in relation to an axis;

- cube([10, 20, 15]);



rotate([0,0,45] cube([10, 20, 15]);

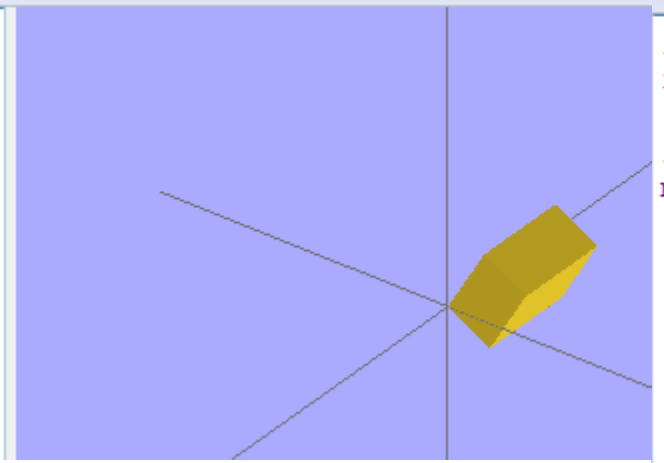


Transformation

- Mirror=> reflect in relation to an axis;

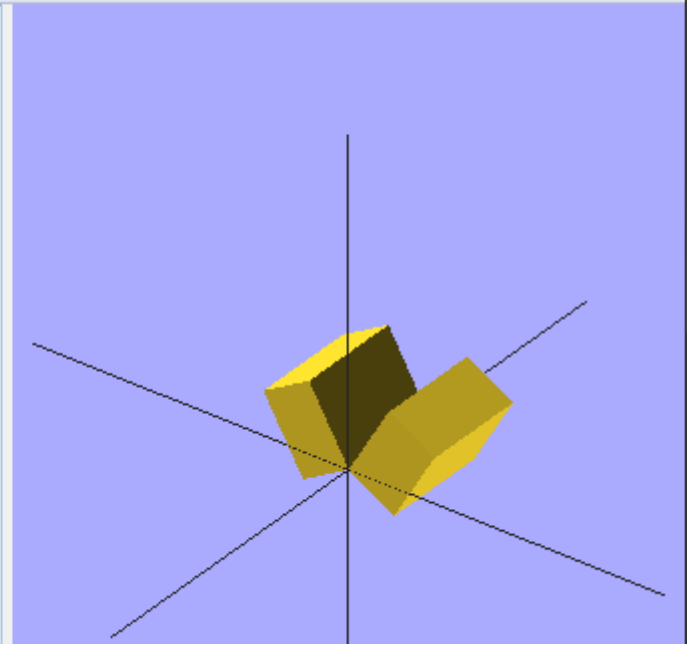
File Edit Design View Help

```
// rotated cube
rotate([0, 30, 0])
  cube([10, 20, 15]);
```



File Edit Design View Help

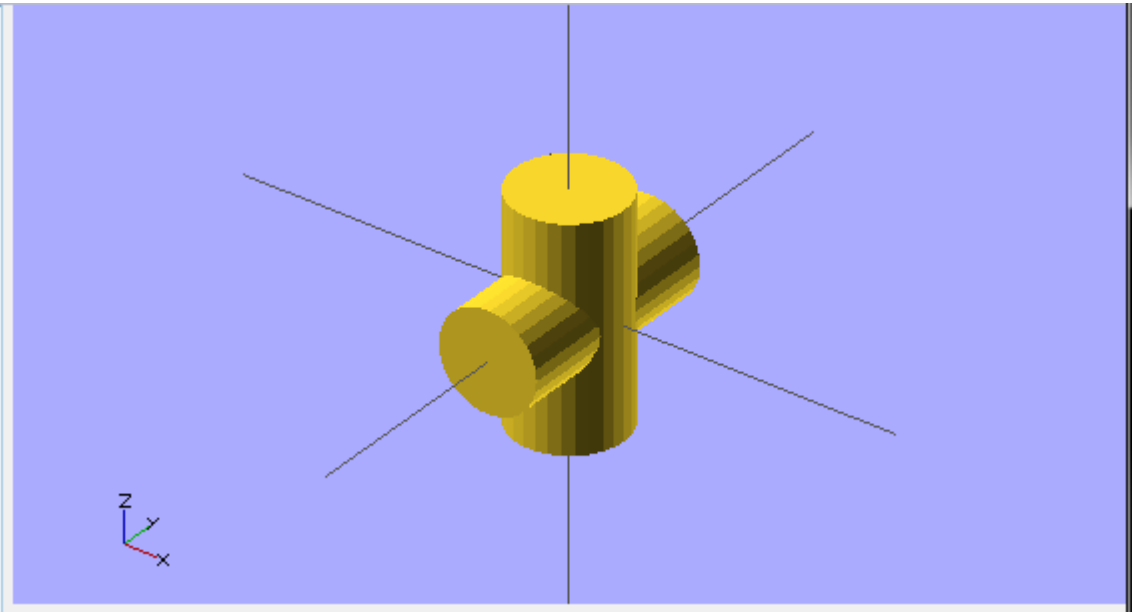
```
// rotated cube
rotate([0, 30, 0])
  cube([10, 20, 15]);
// mirrored across the yz-plane
mirror([1, 0, 0])
  rotate([0, 30, 0])
    cube([10, 20, 15]);
```



Constructive Solid Geometry Operations

- Union

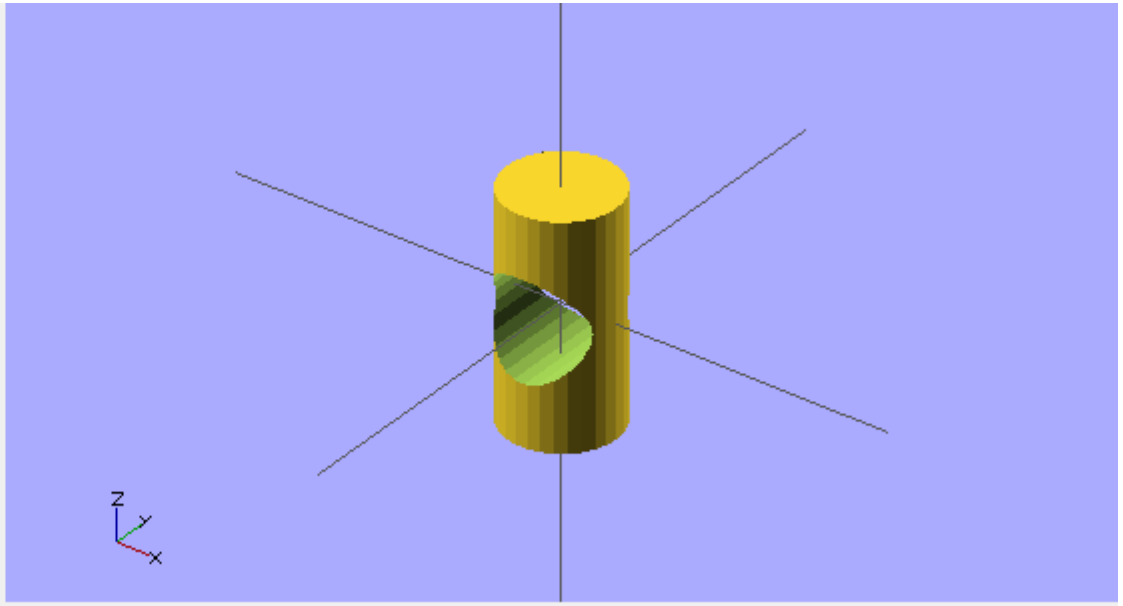
```
union() {
  cylinder(h = 40, r = 10, center = true);
  rotate([90, 0, 0])
    cylinder (h = 40, r = 9, center = true);
}
```



Constructive Solid Geometry Operations

- Difference

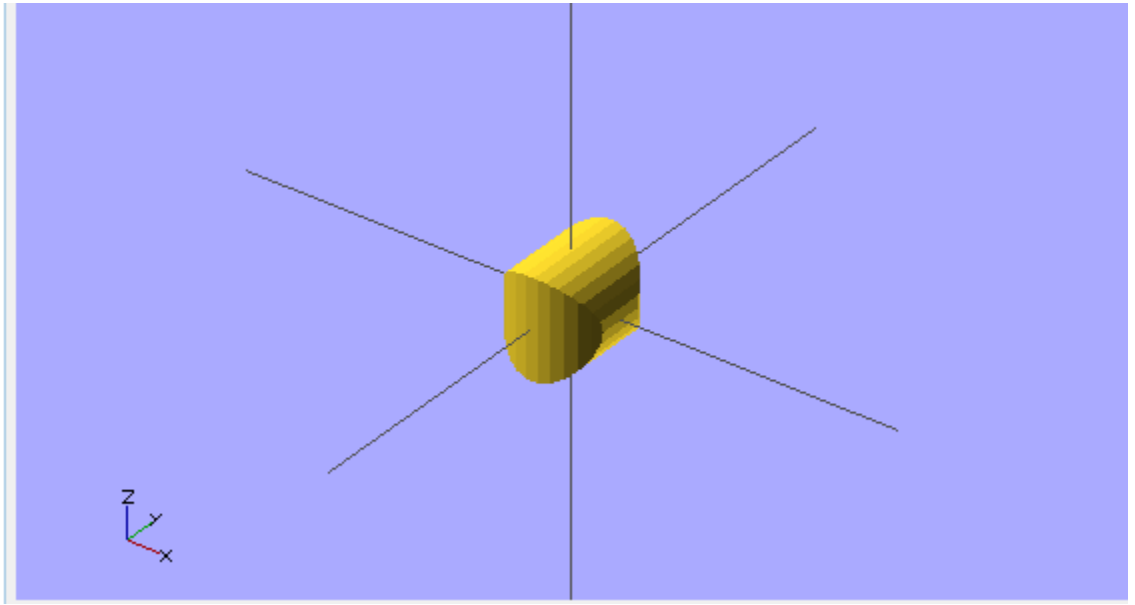
```
difference() {
  cylinder(h = 40, r = 10, center = true);
  rotate([90, 0, 0])
    cylinder(40, r = 9, center = true);
}
```



Constructive Solid Geometry Operations

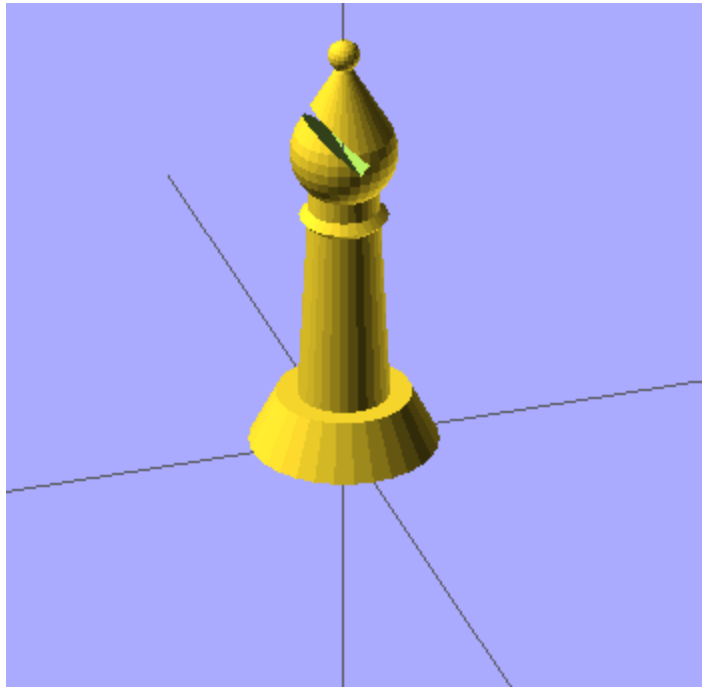
- Intersection => What will overlap?

```
intersection() {
  cylinder(h = 40, r = 10, center = true);
  rotate([90, 0, 0])
    cylinder(40, r = 9, center = true);
}
```



Got the basics?

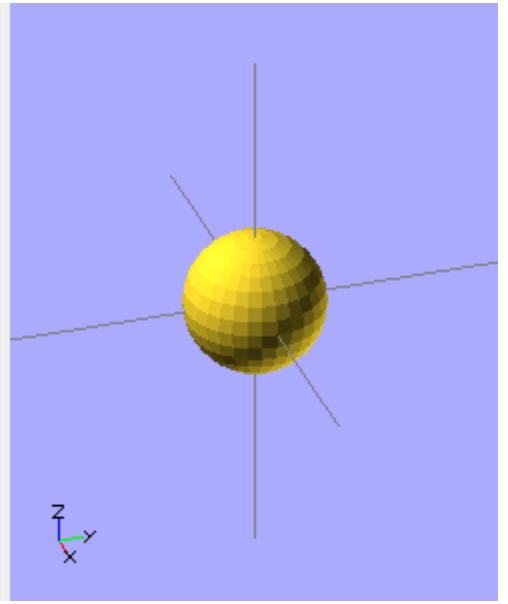
- Practice, practice, practice!
  - And one example



Bishop

- Start with the “teardrop” shape;

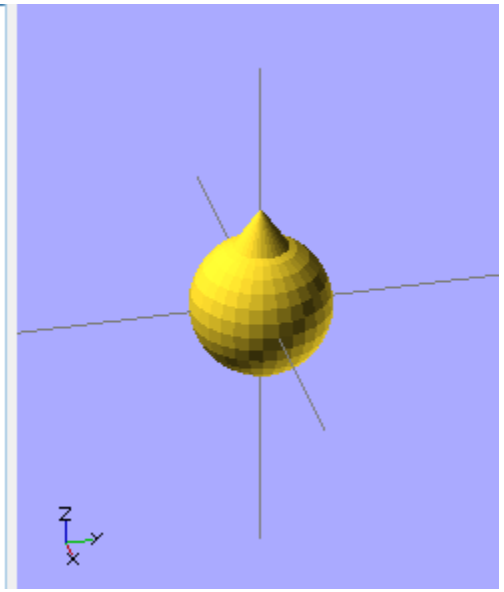
```
sphere(r=20);
```



Bishop

- Start with the “teardrop” shape;
- Add a cone to the sphere;

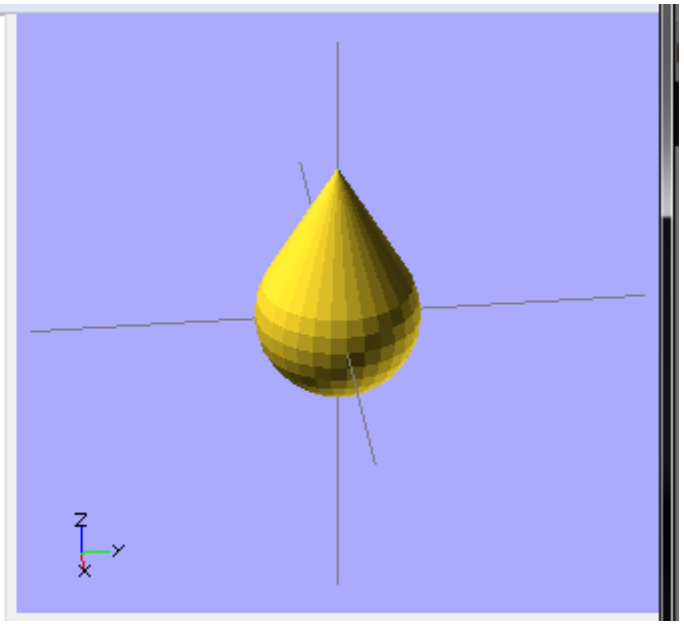
```
union() {
  sphere(r = 20);
  cylinder(h = 30, r1 = 20, r2 = 0);
}
```



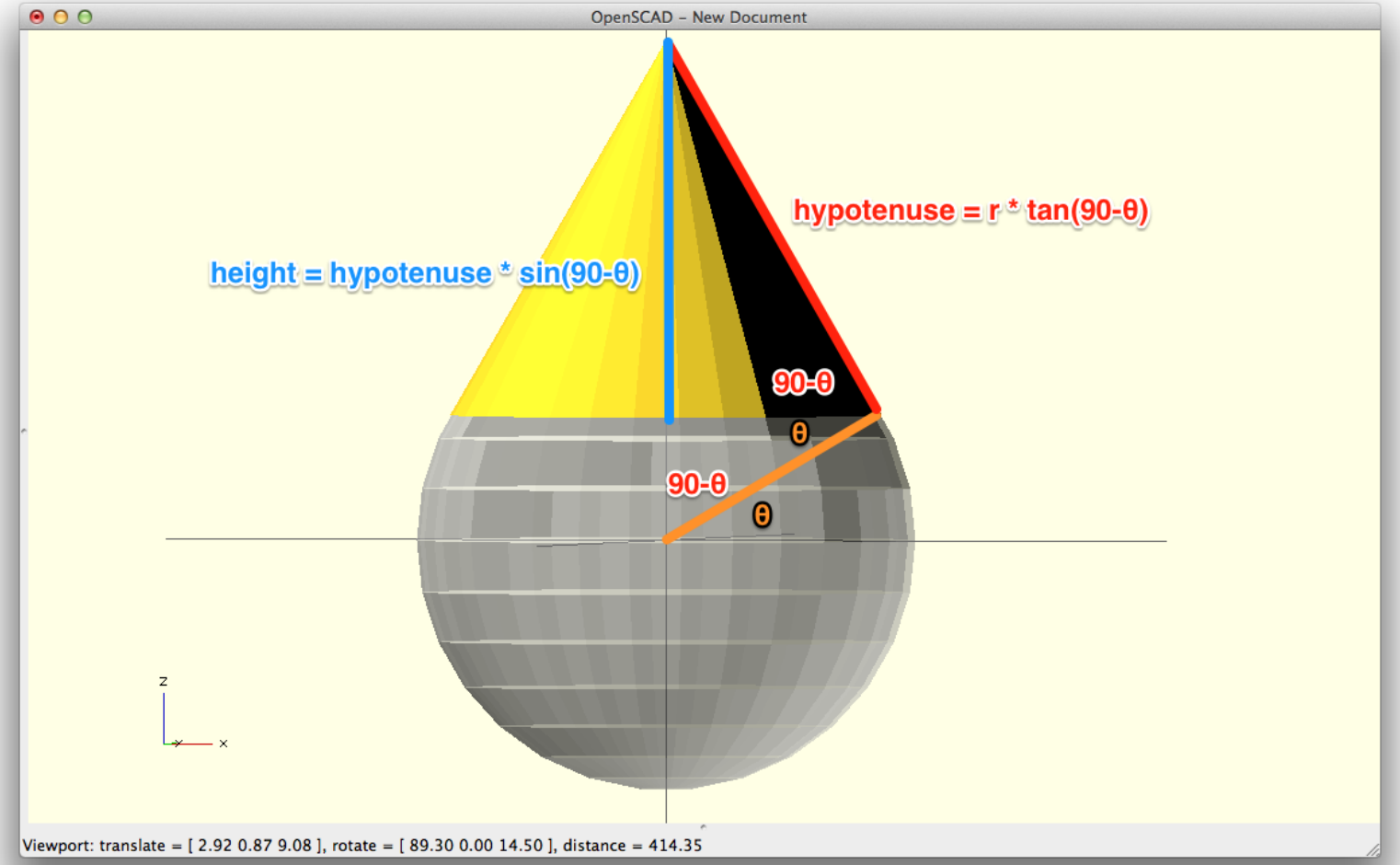
Bishop

- Start with the “teardrop” shape;
- Add a cone to the sphere;
- Match the cone with the sphere

```
union() {
  sphere(r = 20);
  translate([0, 0, 20 * sin(30)])
    cylinder(h = 30, r1 = 20 * cos(30), r2 = 0);
}
```



Why sin(30)???

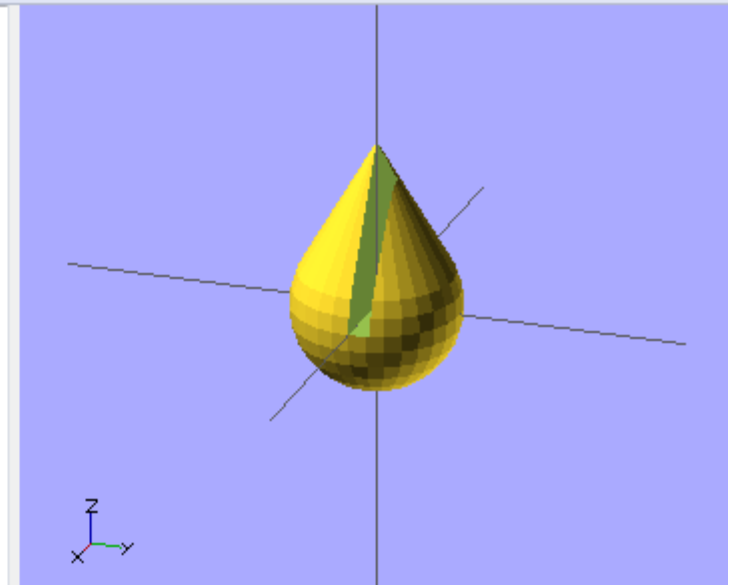

OpenSCAD – New Document
hypotenuse = r * tan(90-θ)
height = hypotenuse * sin(90-θ)
90-θ
θ
90-θ
θ
z
x
Viewport: translate = [2.92 0.87 9.08], rotate = [89.30 0.00 14.50], distance = 414.35

Bishop

- Cut the piece (difference)

File Edit Design View Help

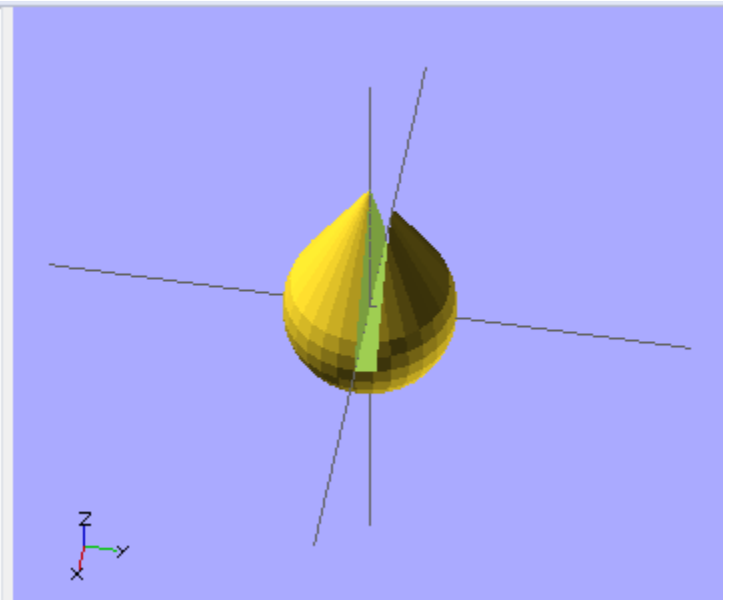
```
difference() {
  union() {
    sphere(r = 20);
    translate([0, 0, 20 * sin(30)])
      cylinder(h = 30, r1 = 20 * cos(30), r2 = 0);
  }
  cube([40, 5, 40]);
}
```



Bishop

- Correct the cut position

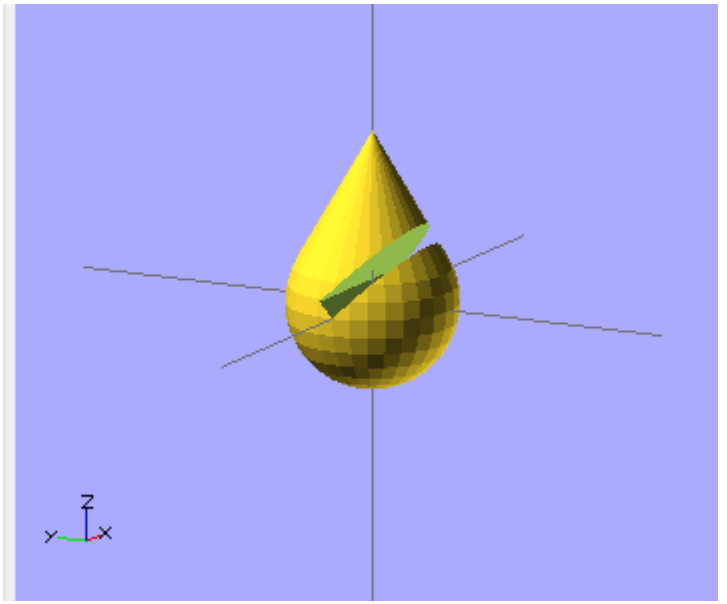
```
difference() {
  union() {
    sphere(r = 20);
    translate([0, 0, 20 * sin(30)])
      cylinder(h = 30, r1 = 20 * cos(30), r2 = 0);
  }
  translate([-20, 0, 0])
    cube([40, 5, 40]);
}
```



Bishop

- Still missing the rotation!

```
difference() {
  union() {
    sphere(r = 20);
    translate([0, 0, 20 * sin(30)])
      cylinder(h = 30, r1 = 20 * cos(30), r2 = 0);
  }
  rotate([45, 0, 0])
    translate([-20, 0, 0])
      cube([40, 5, 40]);
}
```

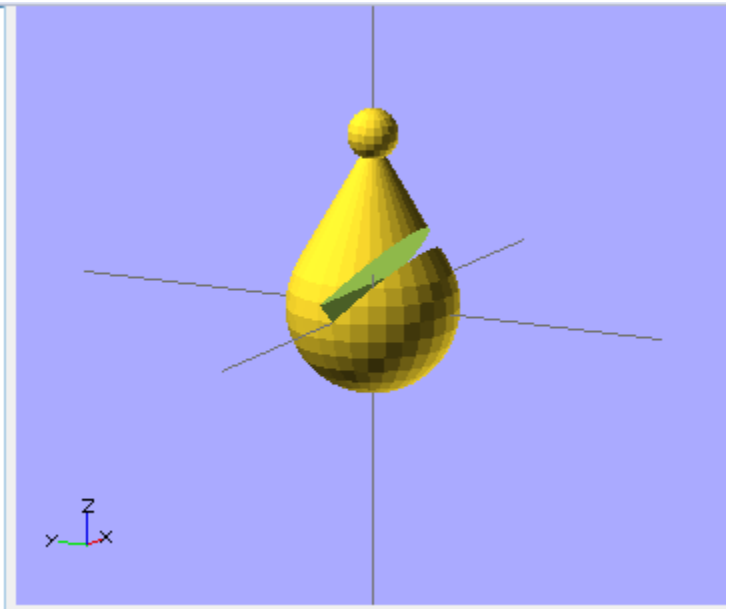


Bishop

- Now, let's add a dollop on top. Since we know the height of the cone is 30, and we moved it up 20 * sin(30), we'll need to translate the dollop 30 + 20 * sin(30). We'll also comment the parts so we don't get confused.

```
difference() {
  union() {
    // teardrop shape
    sphere(r = 20);
    translate([0, 0, 20 * sin(30)])
      cylinder(h = 30, r1 = 20 * cos(30), r2 = 0);

    // dollop
    translate([0, 0, 30 + 20 * sin(30)])
      sphere(r = 6);
  }
  //cut out slot
  rotate([45, 0, 0])
    translate([-20, 0, 0])
      cube([40, 5, 40]);
}
```



Bishop

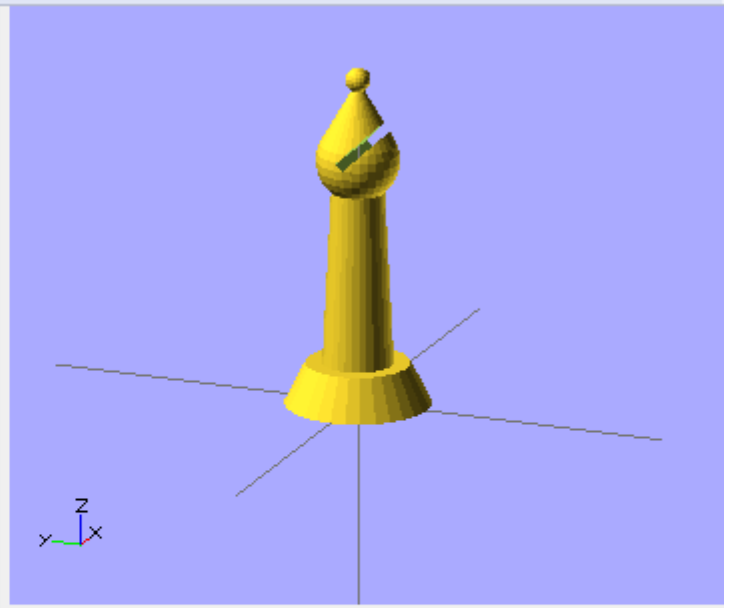
- Add the neck and the base!

File Edit Design View Help

```
  // head
  translate([0, 0, 120])
    difference() {
      union() {
        // teardrop shape
        sphere(r = 20);
        translate([0, 0, 20 * sin(30)])
          cylinder(h = 30, r1 = 20 * cos(30), r2 = 0)
;


        // dollop
        translate([0, 0, 30 + 20 * sin(30)])
          sphere(r = 6);
      }
      //cut out slot
      rotate([45, 0, 0])
        translate([-20, 0, 0])
          cube([40, 5, 40]);
    }
  // neck
  cylinder(h = 120, r1 = 18, r2 = 12);

  // base
  cylinder(h = 20, r1 = 35, r2 = 25);
}
```

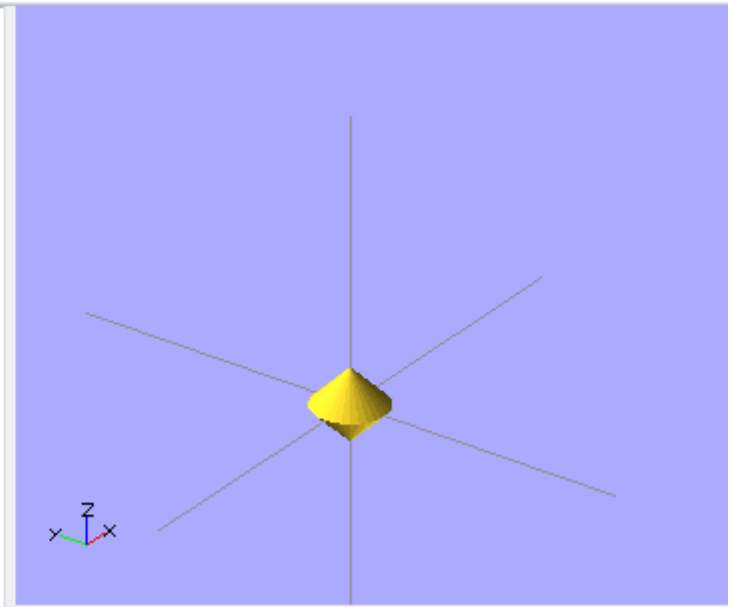



```
CGAL cache size in bytes: 30040624
  Top level object is a 3D object:
  Simple:       yes
  Vertices:     788
  Halfedges:    3368
  Edges:        1684
  Halffacets:   1798
  Facets:       899
  Volumes:      2
Total rendering time: 0 hours, 0 minutes, 2 seconds
Rendering finished.
```

Bishop

- Making a fancy collar

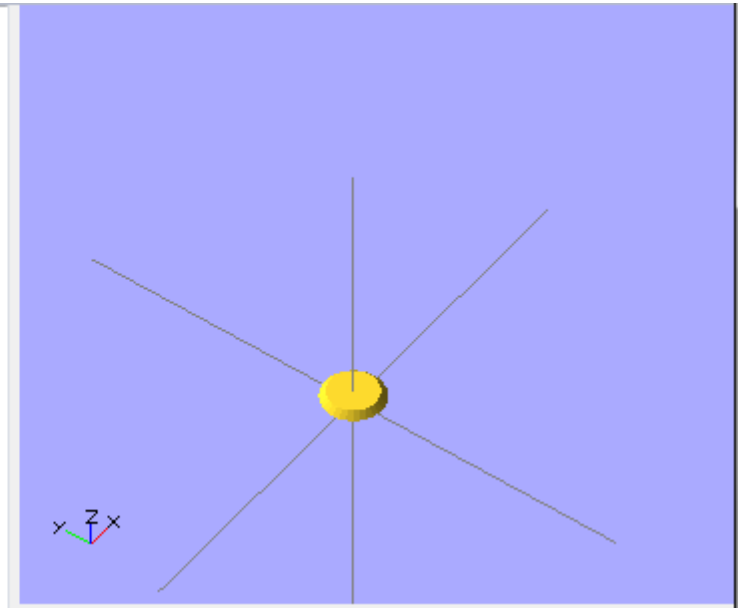
```
cylinder(h = 20, r1 = 20, r2 = 0);
mirror([0, 0, 1])
  cylinder(h = 20, r1 = 20, r2 = 0);
```



Bishop

- Making a fancy collar

```
intersection() {
  cylinder(h = 20, r1 = 20, r2 = 0);
  translate([0, 0, 7])
    mirror([0, 0, 1])
      cylinder(h = 20, r1 = 20, r2 = 0);
}
```



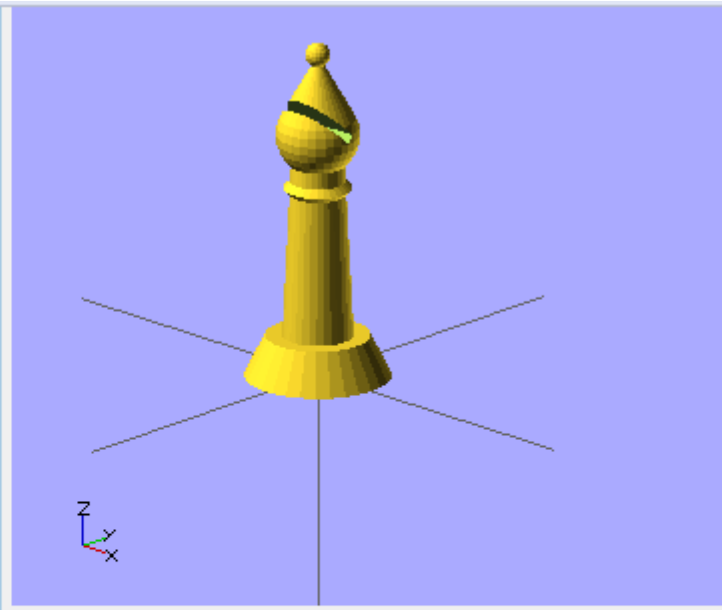
Bishop

- Everything together!

```
        // dollop
        translate([0, 0, 30 + 20 * sin(30)])
          sphere(r = 6);
      }
      //cut out slot
      rotate([45, 0, 0])
        translate([-20, 0, 0])
          cube([40, 5, 40]);
    }
  // neck
  cylinder(h = 120, r1 = 18, r2 = 12);

  // base
  cylinder(h = 20, r1 = 35, r2 = 25);

  // collar
  translate([0, 0, 90])
    intersection() {
      cylinder(h = 20, r1 = 20, r2 = 0);
      translate([0, 0, 7])
        mirror([0, 0, 1])
          cylinder(h = 20, r1 = 20, r2 = 0);
    }
}
```




```
Saved backup file: E:/Victor/My
Documents/OpenSCAD/backups/unsaved-backup-hgqH1736.scad
Compiling design (CSG Tree generation)...
Rendering Polygon Mesh using CGAL...
PolySets in cache: 4
PolySet cache size in bytes: 87936
CGAL Polyhedrons in cache: 72
CGAL cache size in bytes: 31935080
  Top level object is a 3D object:
  Simple:       yes
  Vertices:     990
  Halfedges:    4008
```