OpenSCAD Lesson one

Code Lesson

Intro

In this Tutorial, you will learn how to make the 3 basic shapes of OpenScad, a 3D imaging program that converts code into to shapes.

Vocab

Cube: Creates a cube in the first octant
Cylinder: Creates a cylinder or cone centered about the z axis
Sphere: Creates a sphere at the origin of the coordinate system
Center: determines where the shape is positioned

- When the Boolean value of center is 'true', the cube is centered at the origin (0, 0).
- When the Boolean value of center is 'false', the cubes corner will start at the origin (0, 0).

Syntax

Cube

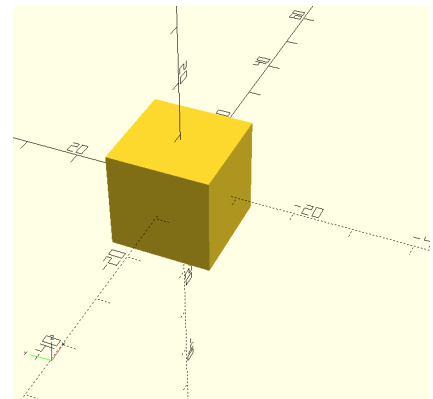cube(size = [x,y,z], center = true/false);

```
cube([18,18,18], center = true);
```

Explanation

Start with the function initiator 'cube',

This has two arguments:

- size - an array of arguments for the dimensions X, Y, and Z
- center - determines where the shape is positioned

Cylinder

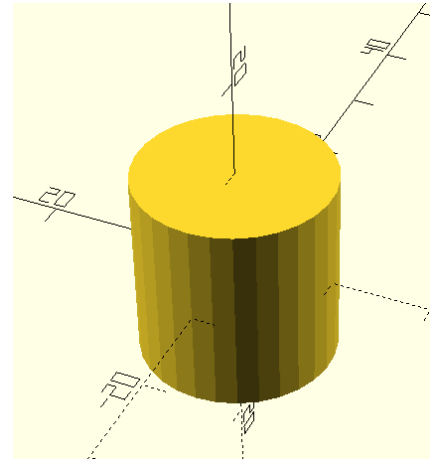cylinder(h = height, r1 = bottomRadius, r2 = topRadius, center = true/false);

```
cylinder(h, r, center = true or false);
```

Explanation:

Start with the function initiator 'cylinder',

This three arguments:

- h - height of the cylinder
- d - diameter of the cylinder
- center - determines where the shape is positioned

Sphere

sphere(r or d, center= true/false)

```
sphere(r = 50, center = true);
```

Explanation:

Start with the function initiator 'sphere',

This has 2 arguments:

- r / d - radius / diameter of the sphere
- center - determines where the shape is positioned