OpenSCAD Lesson one

Code Lesson

Code Color Guide

Keyword Number Boolean

# 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