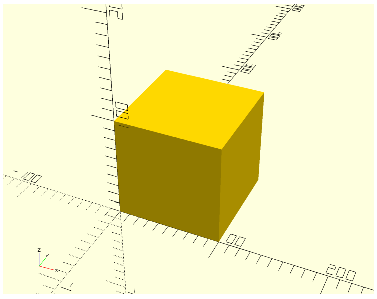
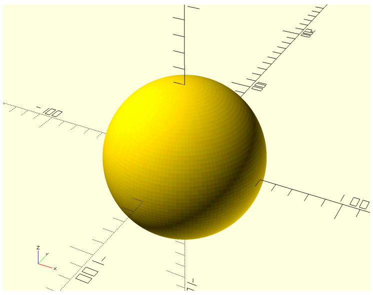
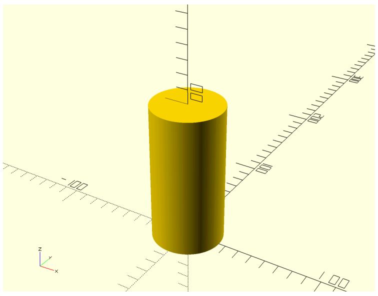
Shapes

There are only three basic 3D shapes you start with, these are the cube, the sphere, and the cylinder.







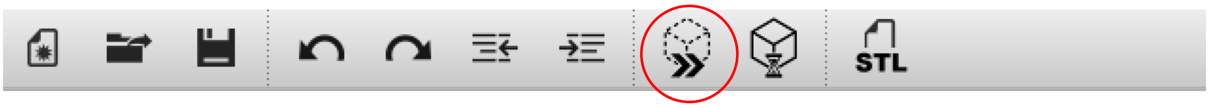
1) Cube

The cube is pretty simple. You declare it with:

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

Don’t forget the semicolon! Now, there’s a cube with length 10, width 20, and height. Each corresponding to each of the x, y, and z axis directions.

To actually draw the object, click the >> symbol in the top ribbon.



2) Sphere

The sphere is pretty simple as well. You only have to declare the radius.

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

3) Cylinder

The cylinder is a bit more complicated, but not by much.

```
cylinder(h = 40, r1 = 20, r2 = 20);
```

This gives us a cylinder of height 40, and a radius of 20. Notice that unlike the cube, the parameters aren’t in a vector, because each of the numbers don’t correspond to the x, y, and z axis. Why is 20 repeated twice? It’s because there’s the radius of the top and bottom circles.

You will see your cylinder is not as smooth as this one. You need to tell OpenSCAD how many ‘sections’ to use for curved objects. Add the following as your first line of code.

```
$fn = 64;
```

Now it should look a lot smoother.

Transformations

Transformations are the way you can manipulate objects in OpenSCAD. Here are a few to get you started.

4) Translate

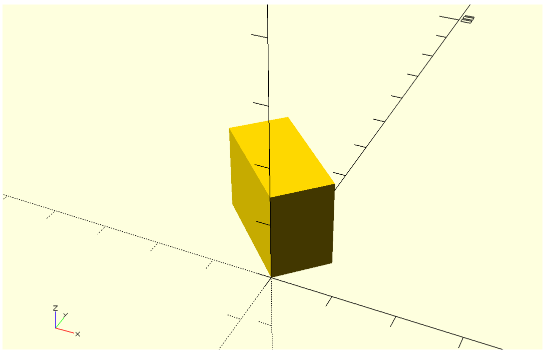
Translation means moving an object by some amount along the x, y, and z axis. To translate (i.e. move) an object along the z-axis by its radius we would do the following. Note that translate is NOT followed by a semi-colon!

```
translate([0, 0, 20])

       sphere(r = 20);
```

5) Rotation

Rotation can get tricky, but easy if handled in steps. Try starting with a cube and rotate it 45 degrees counter-clockwise around the z-axis.

```
rotate([0, 0, 45])

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



6) Union

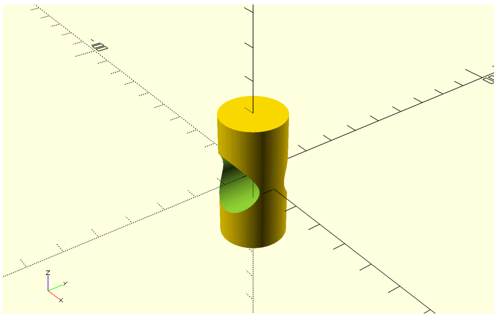
The simplest of the three set operations is union. Union is the sum of all shapes between its brackets.

```
union() {

  cylinder(h = 40, r = 10, center = true);

  rotate([90, 0, 0])

    cylinder (h = 40, r = 9, center = true);

}
```

7) Difference

Difference is using the second shape (and all subsequent shapes in the bracket) to cut out from the first shape. Note that, in this case, only one ‘*r*’ is used for the cylinder and center is set to *true*.

```
difference() {

  cylinder(h = 40, r = 10, center = true);

  rotate([90, 0, 0])

    cylinder(40, r = 9, center = true);

}
```



8) Scaling

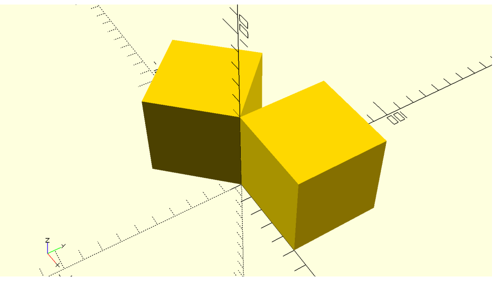
Scaling is just shrinking or stretching a model along an axis. Taking our sphere again, say we want to make an ellipsoid. Try the following...

```
scale([1.5, 1, 1])

  sphere(r = 20);
```

9) Mirror

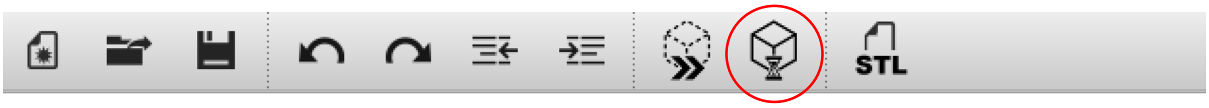
Mirroring can be tricky! It's best to learn this one by experimentation. Here we rotate the cube that we are mirroring so you can see the effect.

```
cube([50, 50, 50]);

rotate(45,0,0)

mirror(1, 0, 0)

cube([50, 50, 50]);
```

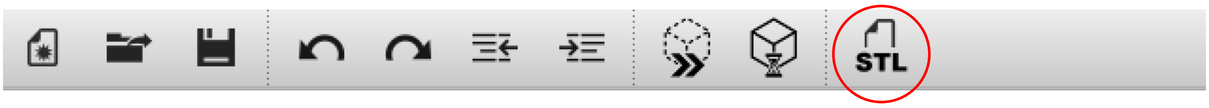


10) Compiling and exporting

Once your design is finished you need to first compile it using the hourglass / cube icon. Depending on the complexity of the design this may take some time!



Finally, you can export your completed stl file ready for slicing.



With the shapes and transformations listed above it's possible to design some pretty cool and complex objects. See what you can come up with! Search online for OpenSCAD examples and tutorials for more information and some more complex tools and techniques.

The content for this workshop is based on 'Know only 10 things to be dangerous in OpenSCAD' by Wil Chung (@iamwil)