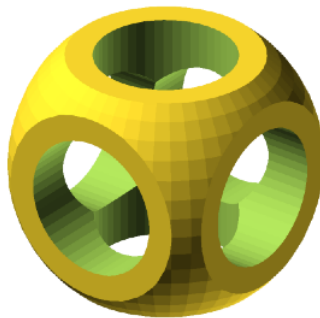# Transforming Shapes - OpenSCAD Syntax Errors

Circle the differences between the two different versions of the same design. One design has multiple syntax errors, while the other is error-free.

Version 1

```
$fn=50;

difference(){

    intersection(){
        cube([10, 10, 10], center=true);
        sphere(d=13);
    }

    union(){
        cylinder(h=12, r=3, center=true);
        rotate([0,90,0]) cylinder(h=12, r=3, center=true);
        rotate([90,0,0]) cylinder(h=12, r=3, center=true);
    }

}
```

Version 2 - 10 syntax errors

```
fn=50;

difference()

    intersect(){
        cube[10, 10, 10], center=true);
        sphere(d 13);
    }

    Union ){
        cylinder(h=12, r=3, center=true)
        rotate(0,90,0) cylidner(h=12, r=3, center=true);
        rotate([90,0,0]) cylinder(h=12, r=3, centr=true);
    }


}
```