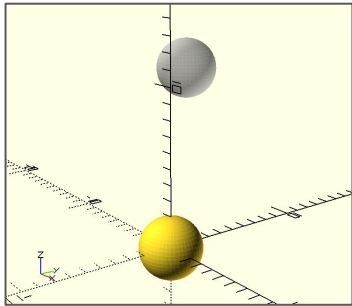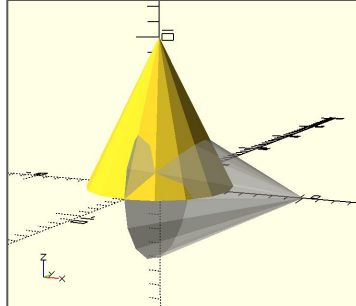## Translate
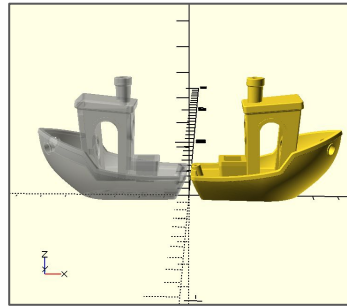
```
translate([-5, 5, 10])
  sphere(r=2, $fn=50);
```
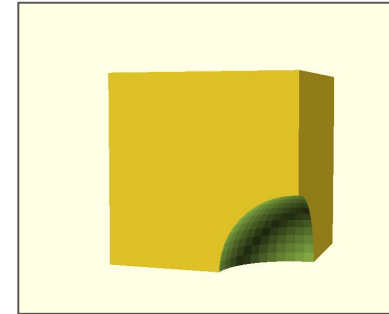
Rotate

```
rotate([0, 90, 0])
  cylinder(h=10, r1=5, r2=0);
```

Mirror
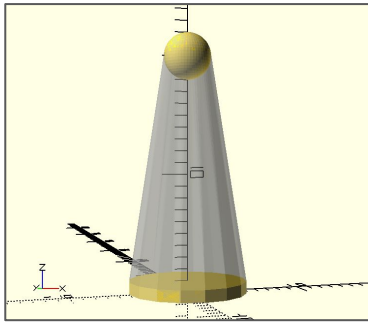
```
mirror([1,0,0])
  translate([30,0,0])
    import("3DBenchy.stl");
```
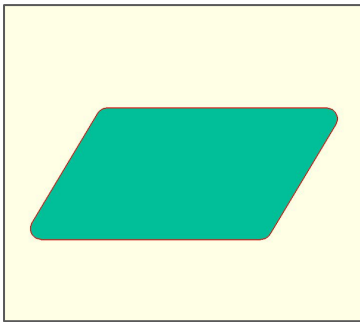
Difference

```
difference(){
  cube([5, 5, 5]);
  sphere(r=2, $fn=50);
}
```
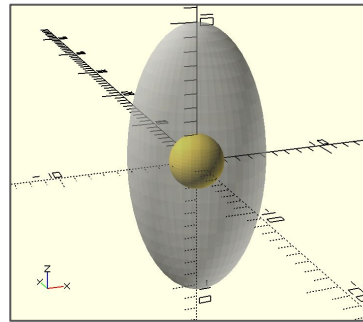
Hull

```
hull(){
  translate([0,0,20])
    sphere(r=2, $fn=50);
  cylinder(h=1, r1=5, r2=5);
}
```
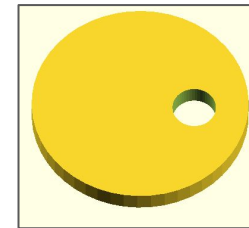
Offset with Fillet

```
offset(r=5) {
  polygon(points=[
    [0,0],[100,0],[130,50],[30,50]
  ]);
}
```
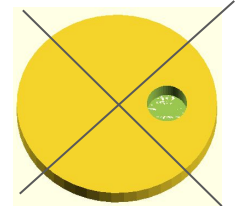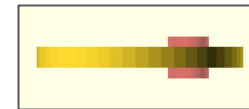
Resize

```
resize([10, 10, 20])
  sphere(r=2, $fn=50);
```
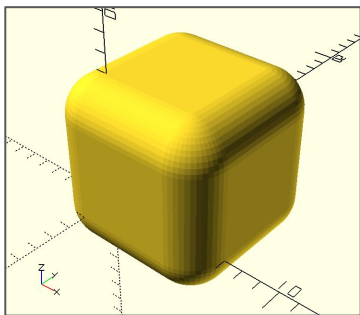
DIfference - Avoid Shimmering Walls

```
difference(){
  cylinder(h=2, r1=10, r2=10, $fn=50);
  translate([5,0,-1])
    cylinder(h=4, r1=2, r2=2, $fn=50);
}
```
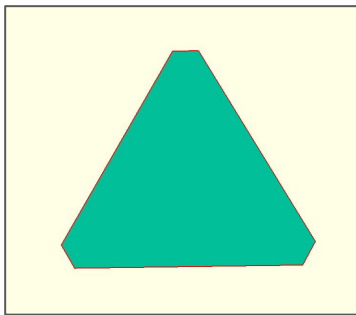
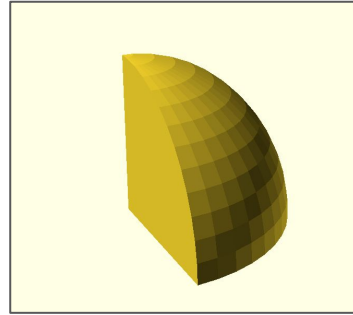## Minkowski

```
minkowski(){
  cube([5, 5, 5]);
  sphere(r=2, $fn=50);
}
```
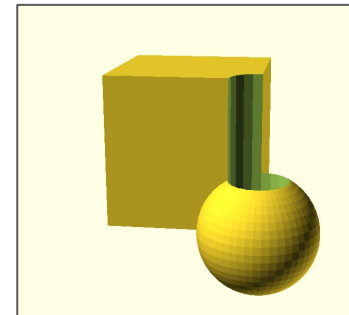
Offset with Chamfer

```
offset(delta=1, chamfer=true) {
  circle(r=5, $fn=3);
}
```

Intersection

```
intersection(){
  sphere(r=2, $fn=50);
  cube([5, 5, 5]);
}
```

Union

```
difference(){
  union(){
    cube([5, 5, 5]);
    sphere(r=2, $fn=50);
  }
  cylinder(h=10, r1=1, r2=1, $fn=20);
}
```

OpenSCAD: Transforming Shapes
Quick Reference