# OpenSCAD

https://lamm.space/
3D Design Day 2017

@idnorton

I'm not an
OpenSCAD expert... :)

# Why OpenSCAD?

- Absolute positioning and sizing

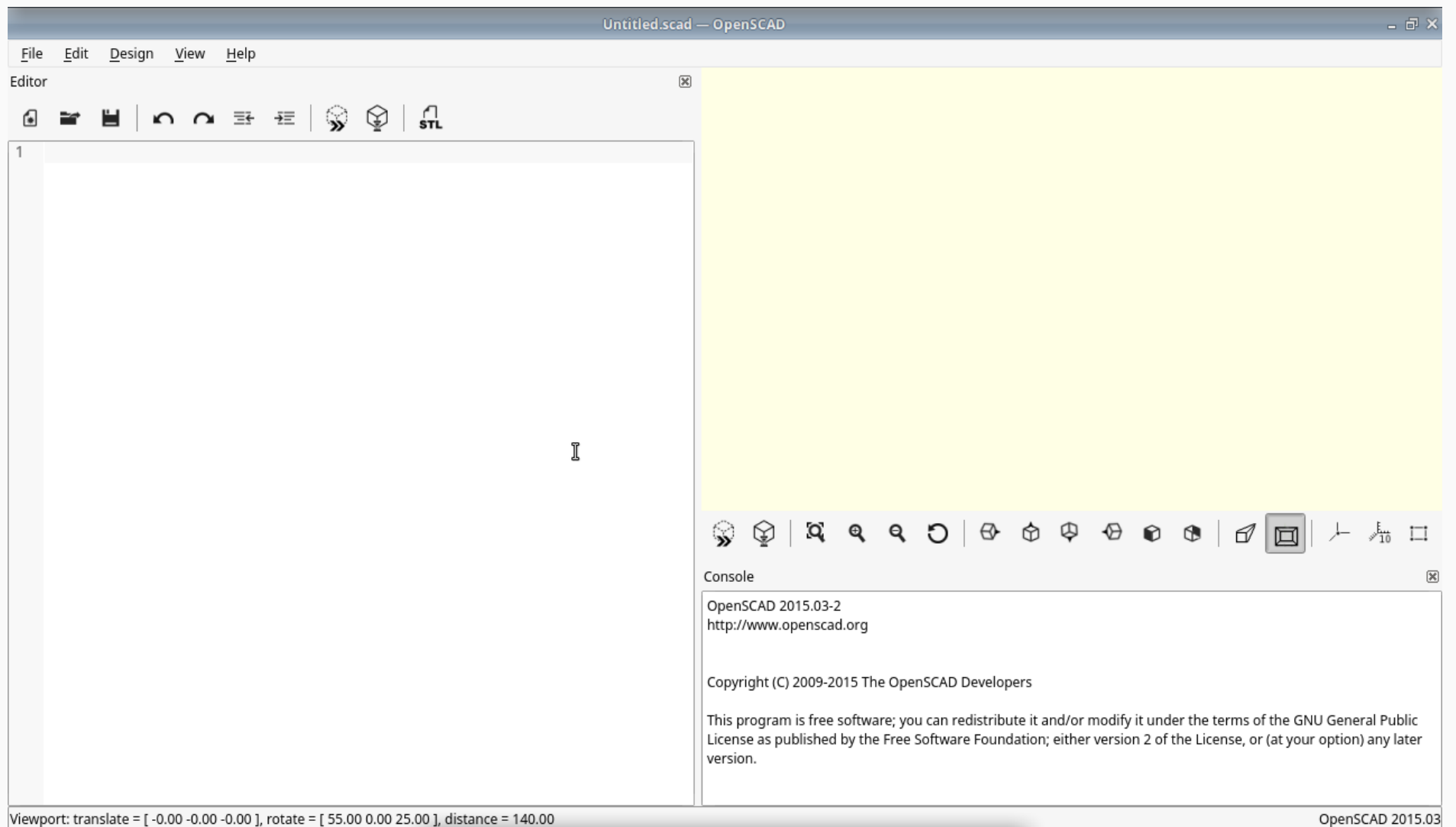- Programmatic

- Works on most OS

# Why OpenSCAD?

- Absolute positioning and sizing

- Programmatic

- Works on most OS
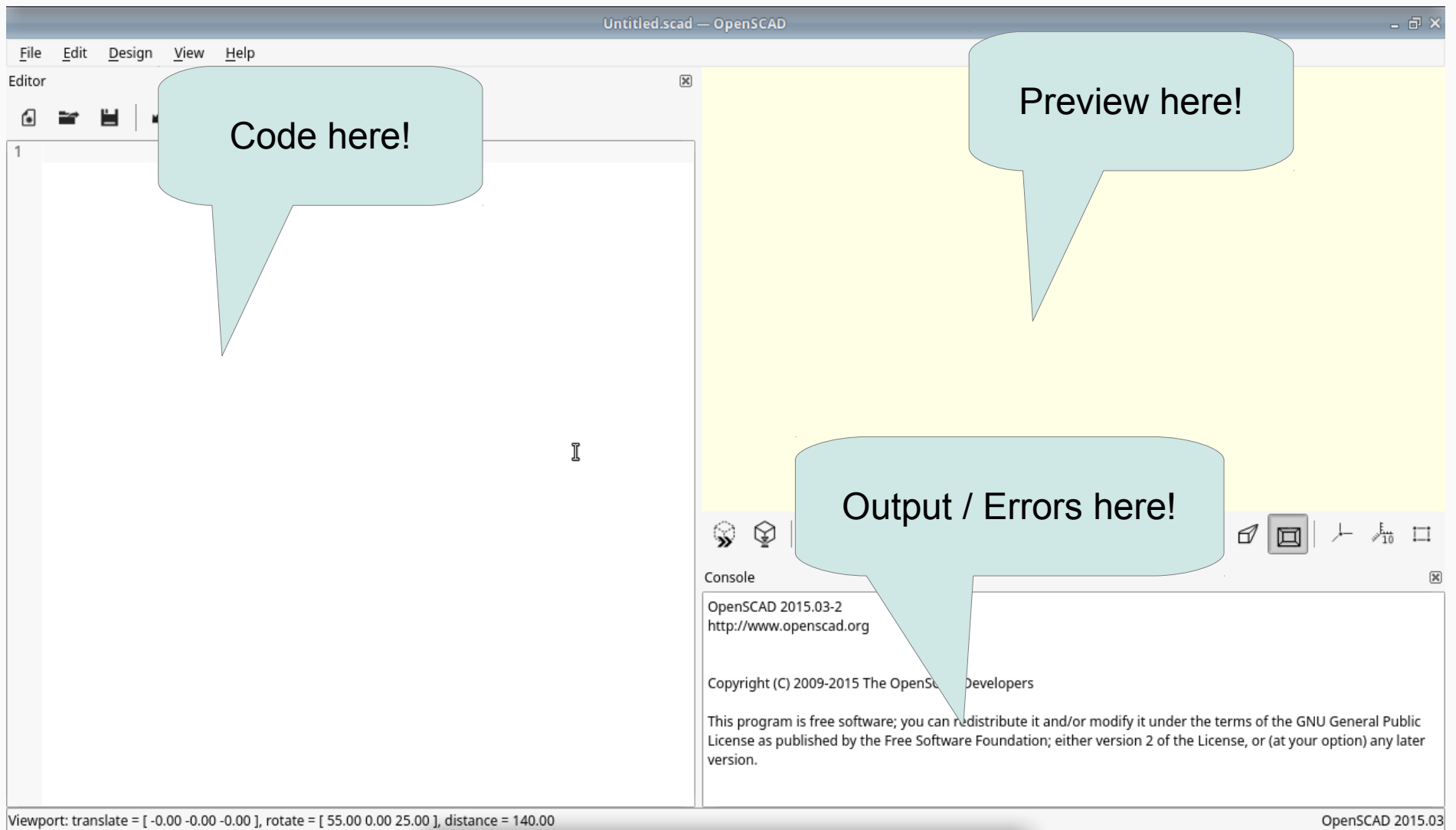
- I really hate graphical tooling :/

# Which version?

# Which version?

>= 2015.03

Adds support for text

File   Edit   Design   View   Help

Editor

```
1
```

Console

Viewport: translate = [ -0.00 -0.00 -0.00 ], rotate = [ 55.00 0.00 25.00 ], distance = 140.00                OpenSCAD 2015.03

# Simple Example

# Simple Example

- Cube

```
cube(size = 10, center = false);
```

File    Edit    Design    View    Help

Editor

```
1    cube(size = 10, center = false);
```

Console

```
Saved backup file: /home/iann/.local/share/OpenSCAD/backups/unsaved-backup-eoBu3010.scad
Compiling design (CSG Tree generation)...
Compiling design (CSG Products generation)...
Geometries in cache: 46
Geometry cache size in bytes: 29896
CGAL Polyhedrons in cache: 2
CGAL cache size in bytes: 401136
Compiling design (CSG Products normalization)...
Normalized CSG tree has 1 elements
Compile and preview finished.
Total rendering time: 0 hours, 0 minutes, 0 seconds
```

Viewport: translate = [ -0.00 -0.00 -0.00 ], rotate = [ 55.00 0.00 25.00 ], distance = 140.00                OpenSCAD 2015.03

Preview pane

# Navigating preview

- Left mouse button
  - Move in 3d

# Navigating preview

- ## Left mouse button

  - ### Move in 3d


- ## Right mouse button

  - ### Move in 2d

# Navigating preview

- Left mouse button
  - Move in 3d


- Right mouse button
  - Move in 2d


- Scroll wheel to zoom in and out

# 2D Shapes

# Square

```
square(size = [x, y], center = true/false);
```

```
square(size = [10, 10], center = false);
```

# Circle

```
circle(r=radius | d=diameter);
```
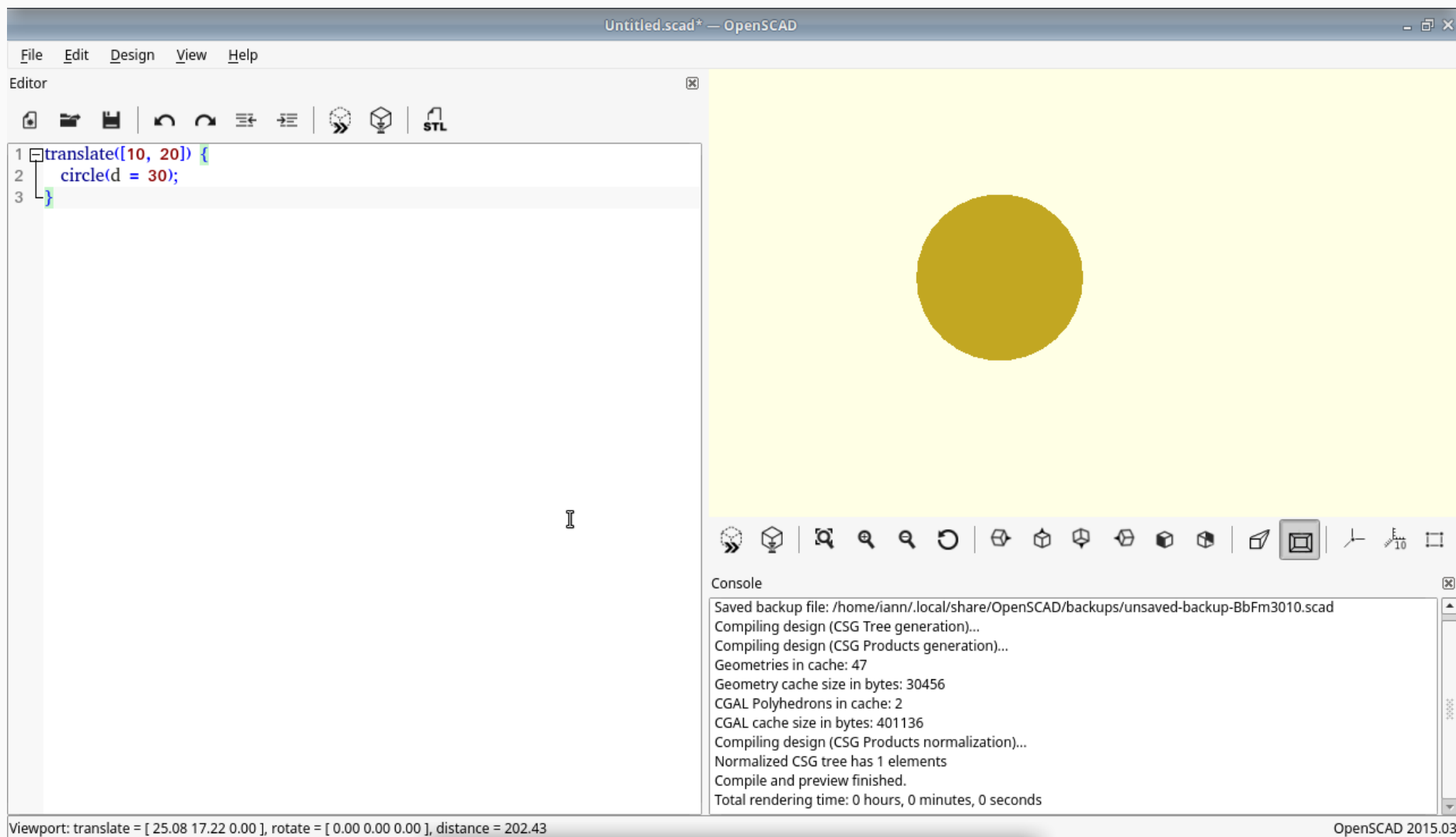
```
circle(d = 10);
```

# Making 2D thing

# Positioning

```
translate([x, y]) {
  // Add your thing here
}
```

- Circle at x 10, y 20, with diameter 30

```
translate([10, 20]) {
  circle(d = 30);
}
```
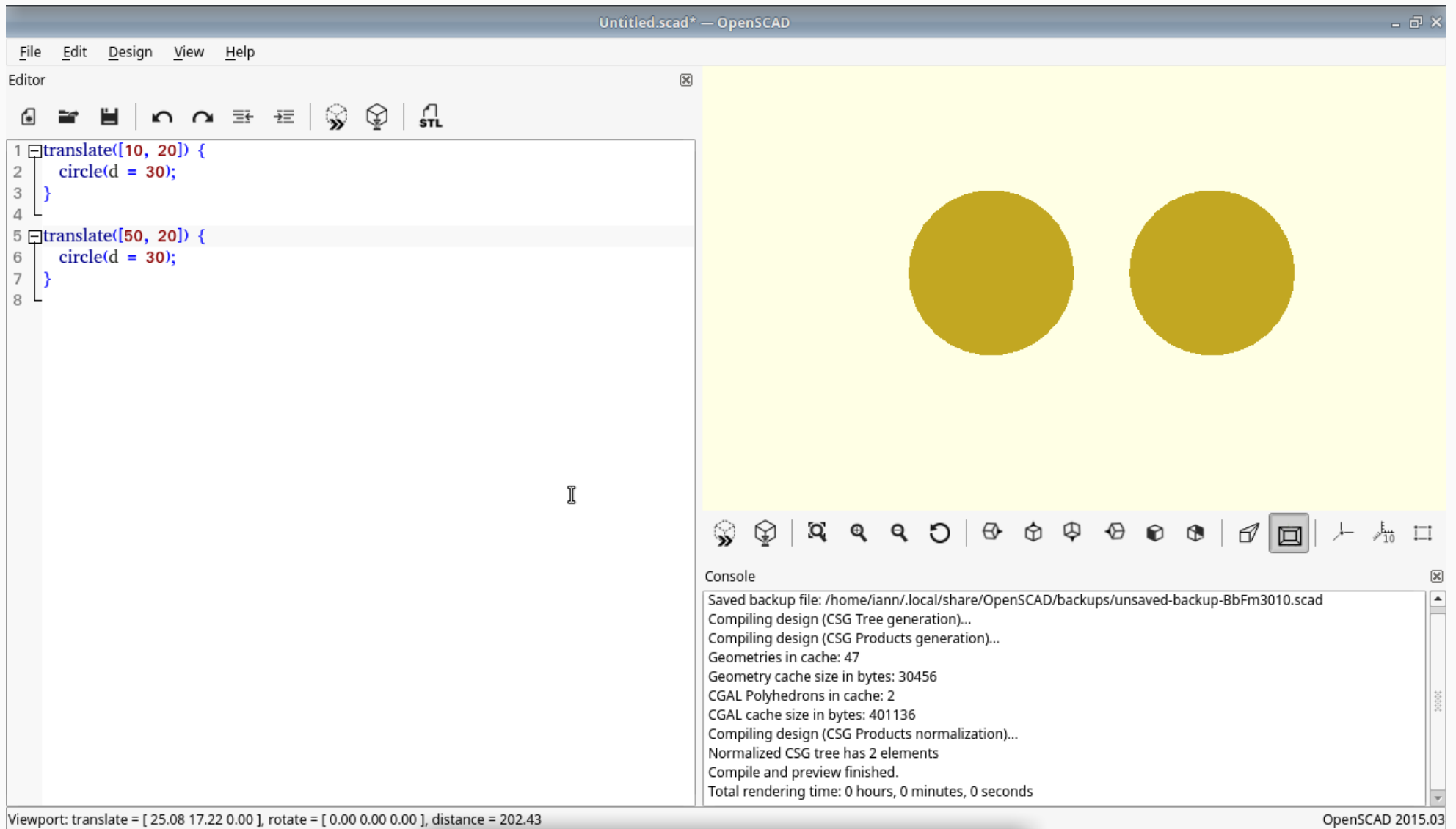
File    Edit    Design    View    Help

Editor

```
1  translate([10, 20]) {
2      circle(d = 30);
3  }
```

Console

Compiling design (CSG Tree generation)...
Compiling design (CSG Products generation)...
Geometries in cache: 47
Geometry cache size in bytes: 30456
CGAL Polyhedrons in cache: 2
CGAL cache size in bytes: 401136
Compiling design (CSG Products normalization)...
Normalized CSG tree has 1 elements
Compile and preview finished.
Total rendering time: 0 hours, 0 minutes, 0 seconds

Viewport: translate = [ 25.08 17.22 0.00 ], rotate = [ 0.00 0.00 0.00 ], distance = 202.43                                    OpenSCAD 2015.03

# Positioning

- More than one circle

```
translate([10, 20]) {
  circle(d = 30);
}

translate([50, 20]) {
  circle(d = 30);
}
```

File   Edit   Design   View   Help

Editor

```
1  translate([10, 20]) {
2      circle(d = 30);
3  }
4
5  translate([50, 20]) {
6      circle(d = 30);
7  }
8
```

Console

Viewport: translate = [ 25.08 17.22 0.00 ], rotate = [ 0.00 0.00 0.00 ], distance = 202.43          OpenSCAD 2015.03

# Combining objects

# difference

```
difference() {
  // Outer circle 50mm diameter
  circle(d = 50);

  // Hole through the middle 20mm diameter
  circle(d = 20);
}
```

File    Edit    Design    View    Help
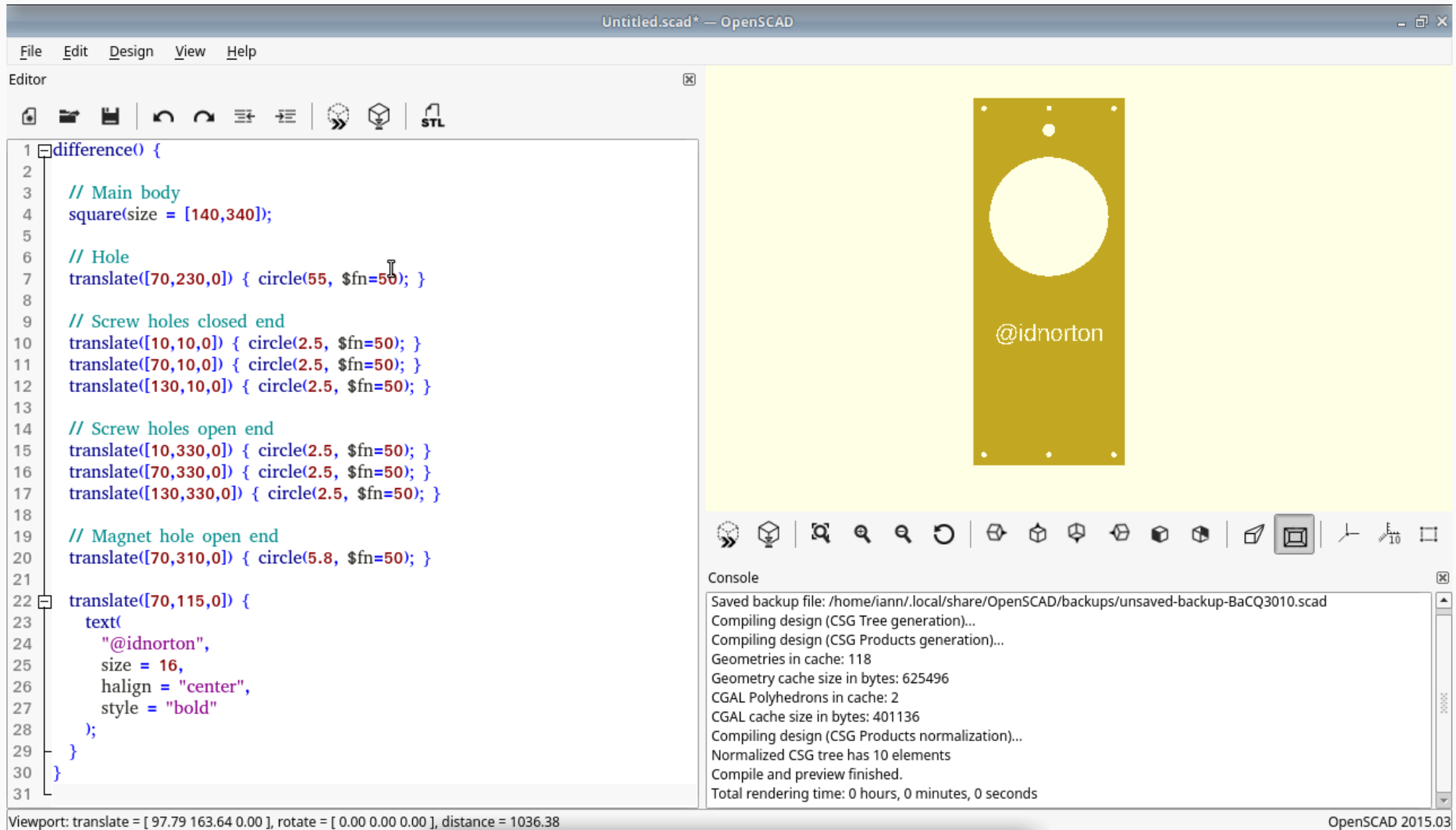
Editor ⊠

```
1  difference() {
2      // Outer circle 50mm diameter
3      circle(d = 50);
4
5      // Hole through the middle 20mm diameter
6      circle(d = 20);
7  }
```
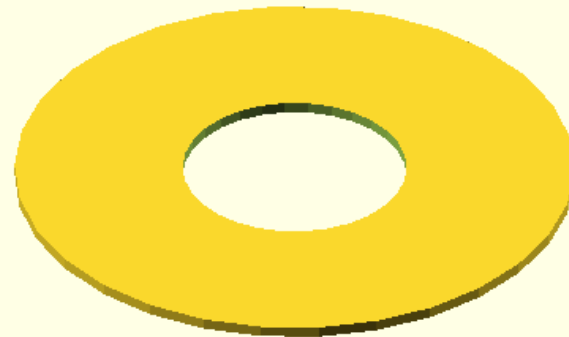
Console ⊠

```
Saved backup file: /home/iann/.local/share/OpenSCAD/backups/unsaved-backup-AnVB3010.scad
Compiling design (CSG Tree generation)...
Compiling design (CSG Products generation)...
Geometries in cache: 48
Geometry cache size in bytes: 31016
CGAL Polyhedrons in cache: 2
CGAL cache size in bytes: 401136
Compiling design (CSG Products normalization)...
Normalized CSG tree has 2 elements
Compile and preview finished.
Total rendering time: 0 hours, 0 minutes, 0 seconds
```

Viewport: translate = [ -0.00 -0.00 -0.00 ], rotate = [ 55.00 0.00 25.00 ], distance = 140.00          OpenSCAD 2015.03

# More complex example

# Other combinations

- union
- difference
- intersection
- render

# 3D Shapes

# Cube

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

cube(size = 10, center = false);
```

# Cylinder

```
Cylinder(
    h = height,
    r1 = BottomRadius,
    r2 = TopRadius,
    center = true/false
);
```

# Further reading

- http://www.openscad.org/
- https://en.wikibooks.org/wiki/OpenSCAD_User_Manual

- https://github.com/lammspace/3d-printer-shelves
- https://github.com/lammspace/Laser-Supply-Storage
- https://github.com/idnorton/hardware-designs

# Questions

# &

# Answers

# Thanks for listening!