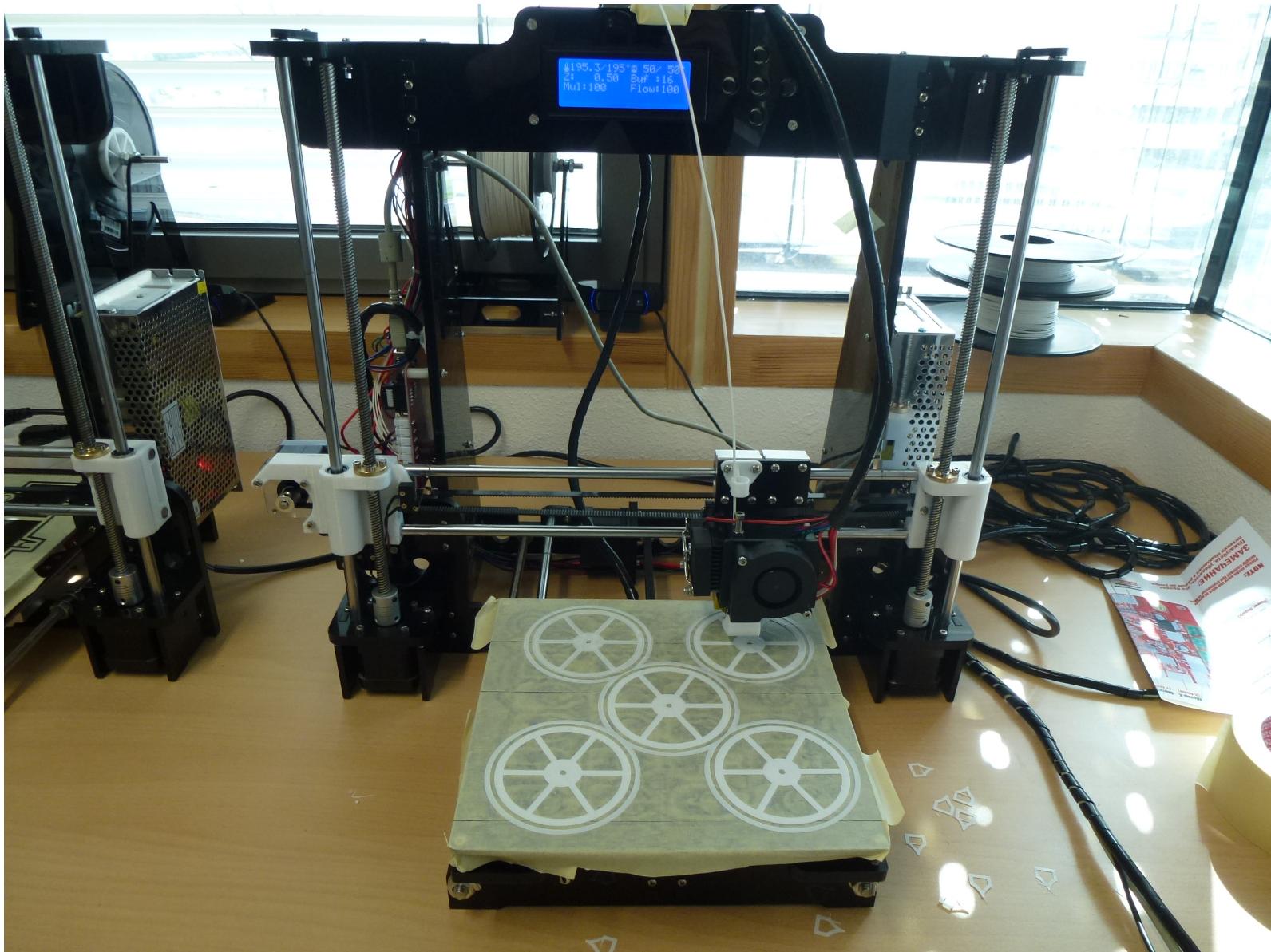


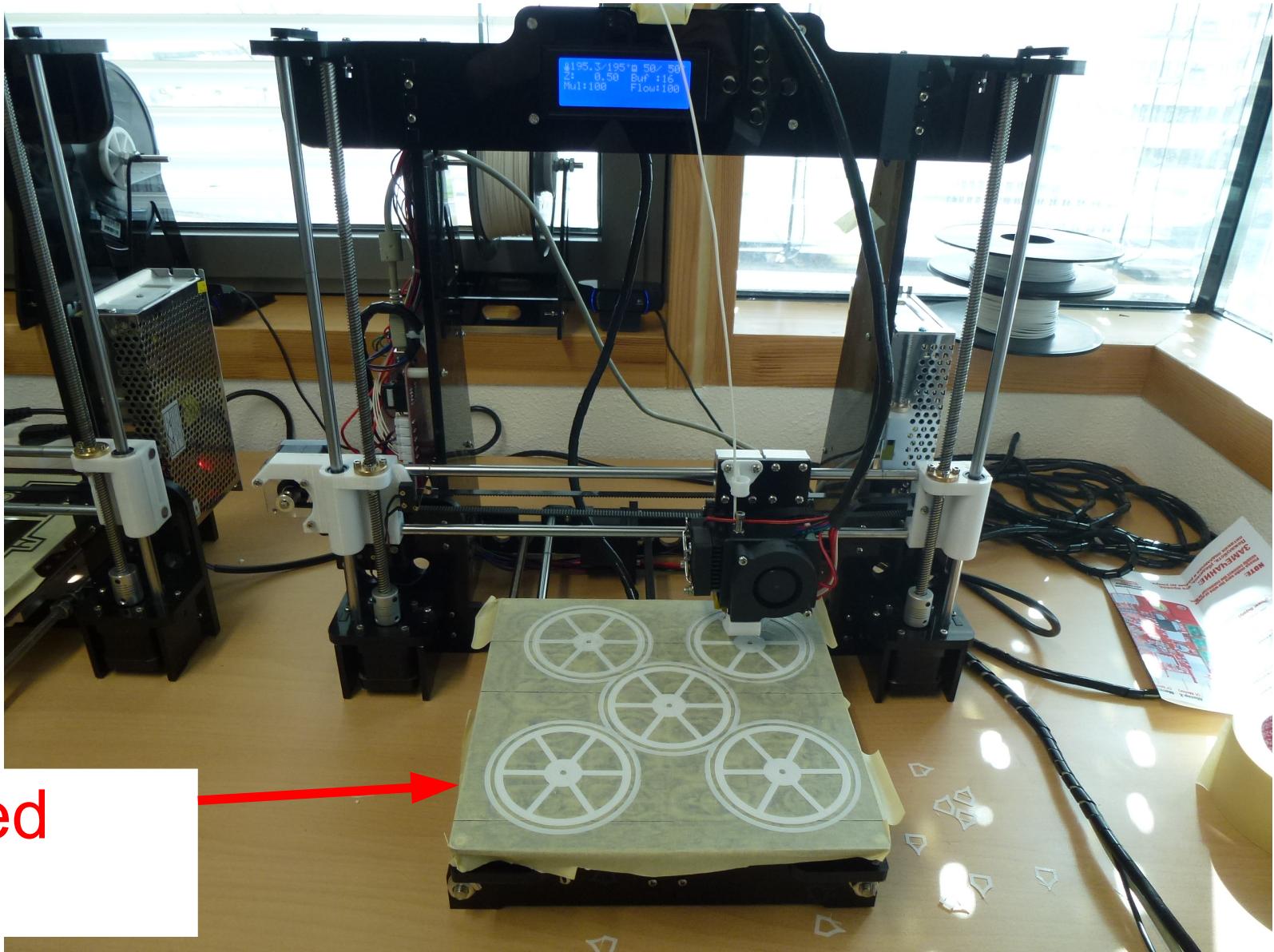
# 3D printing



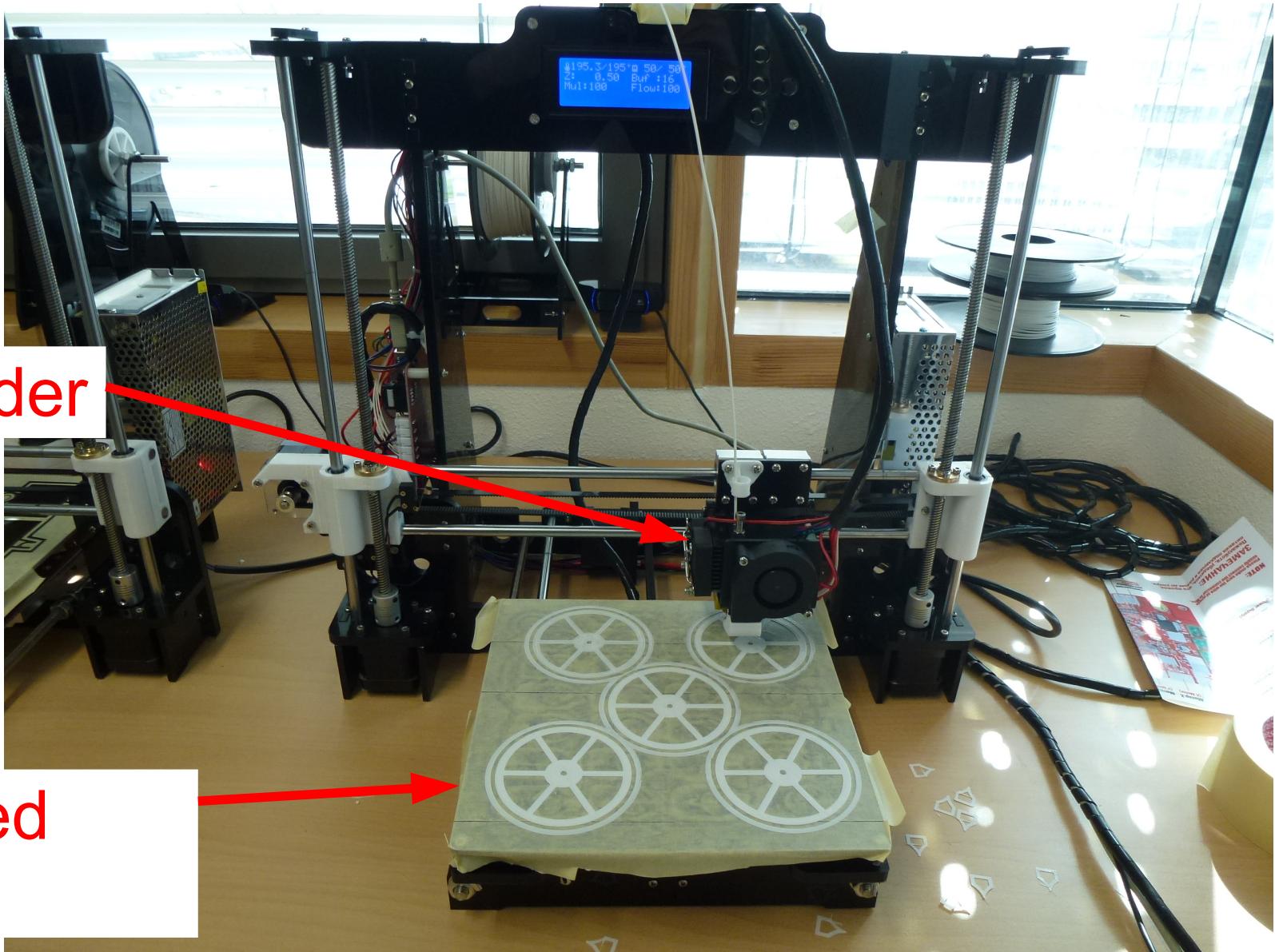
# 3D printers



# 3D printers



# 3D printers

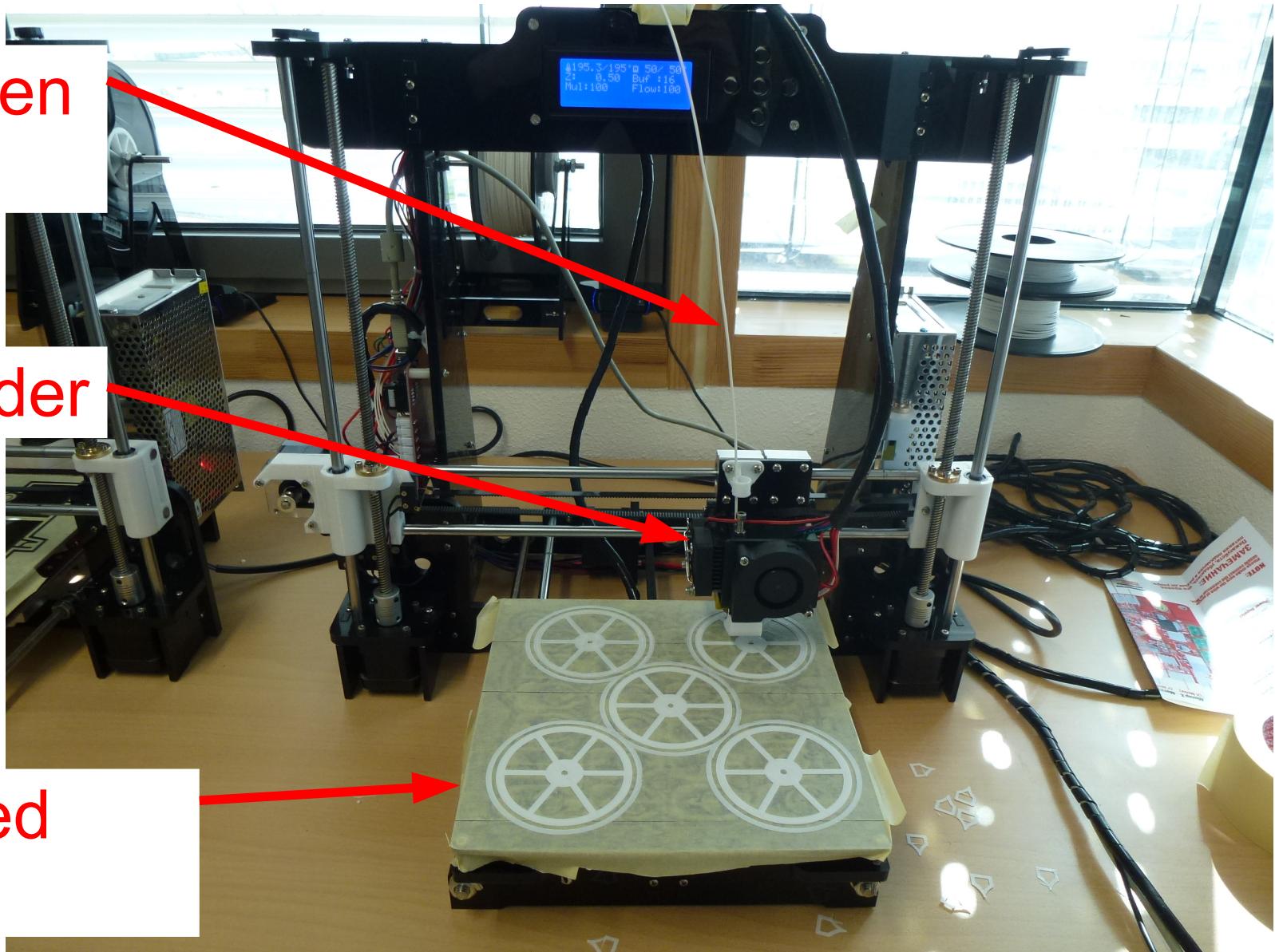


# 3D printers

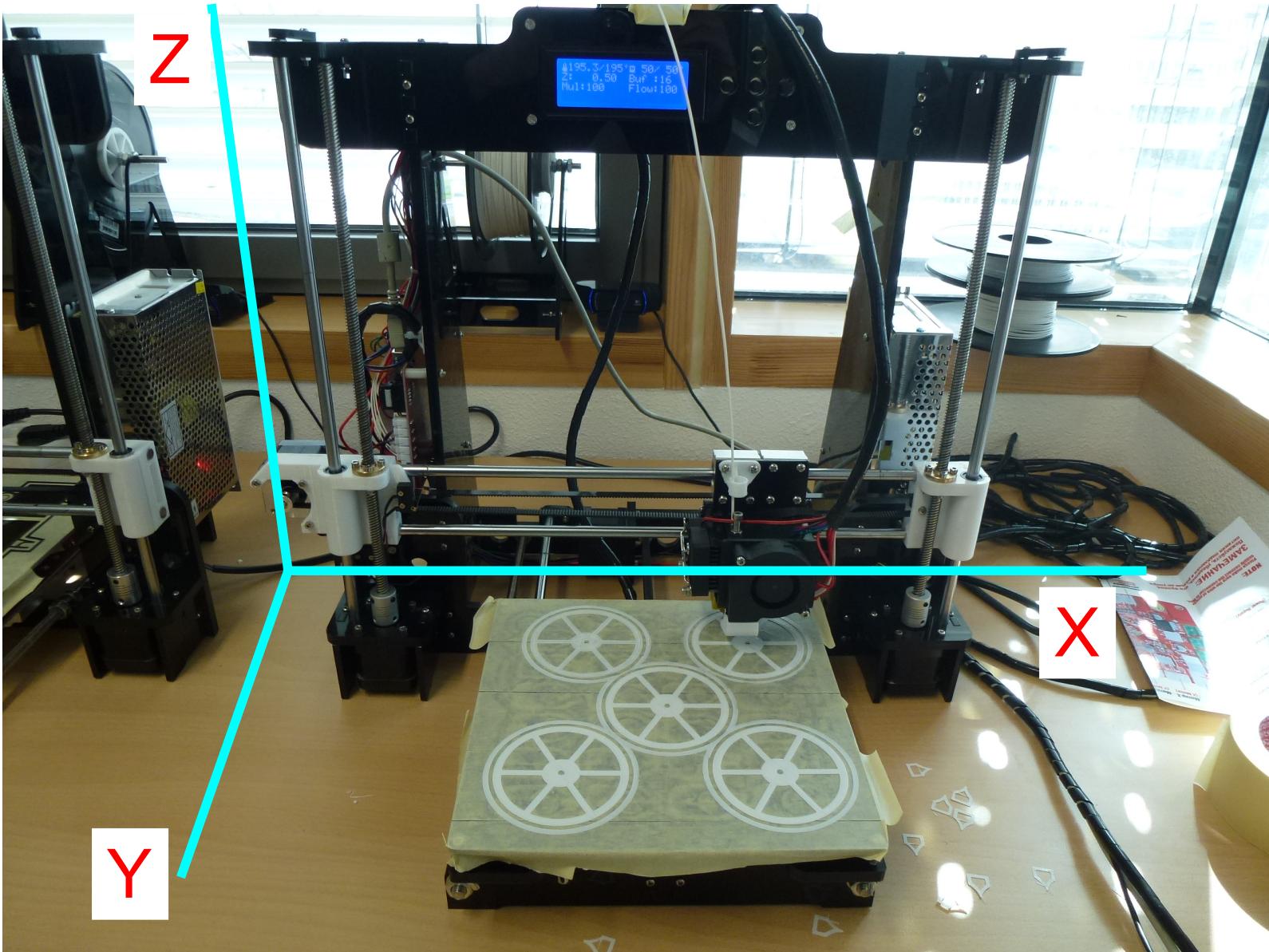
Filamen  
t

Extruder

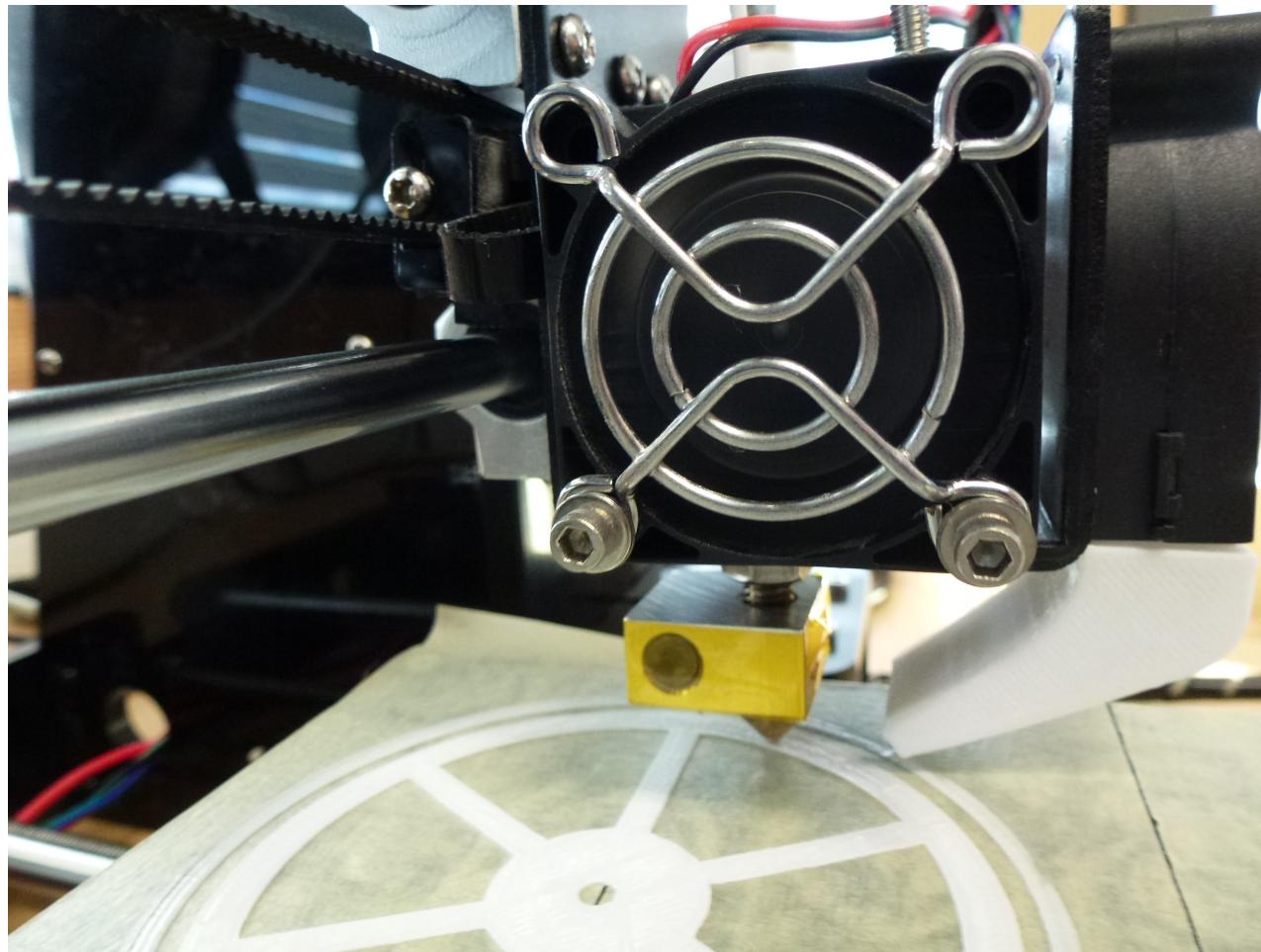
Heated  
bed



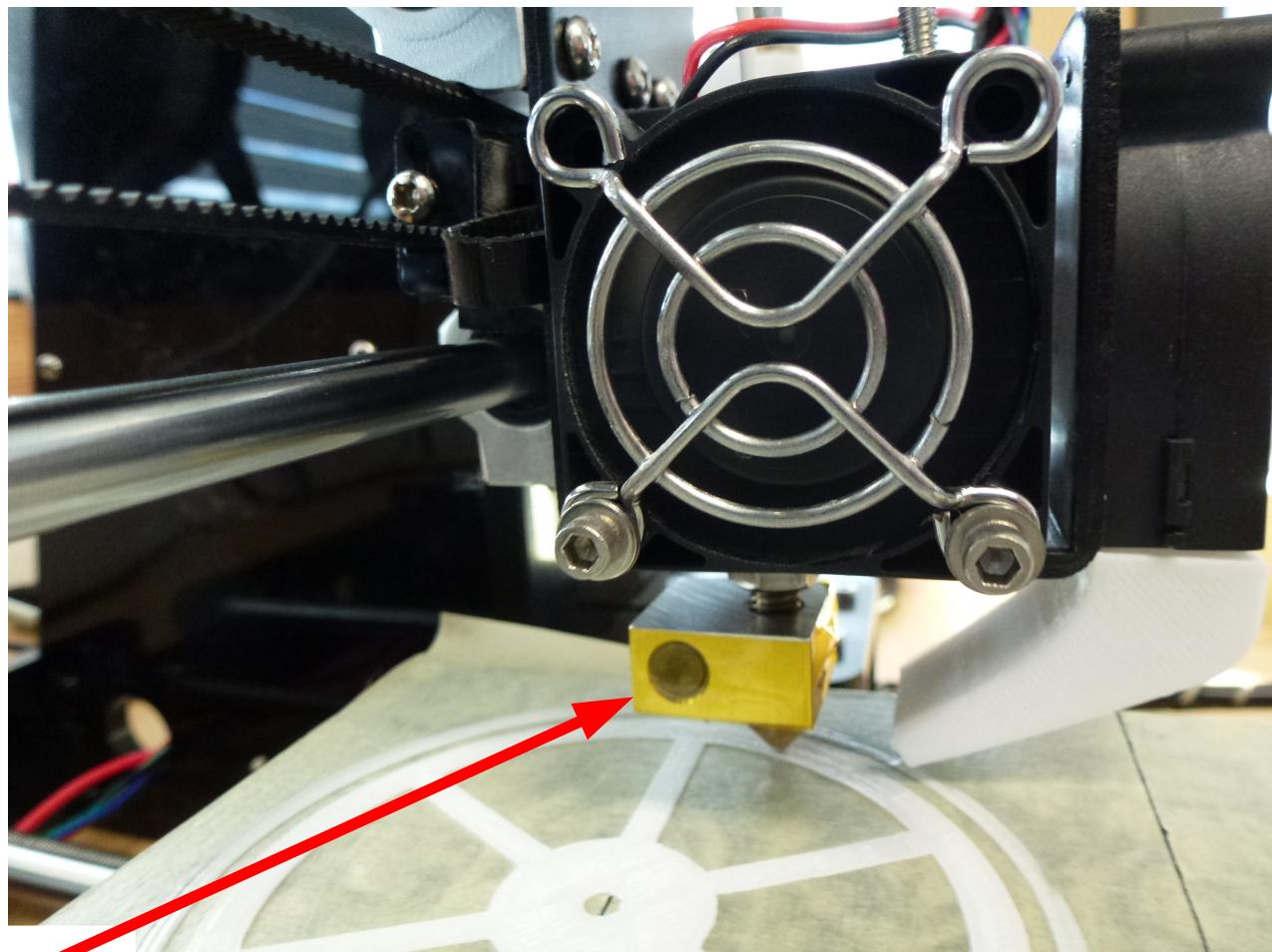
# 3D printers - axes



# 3D printers - extruder

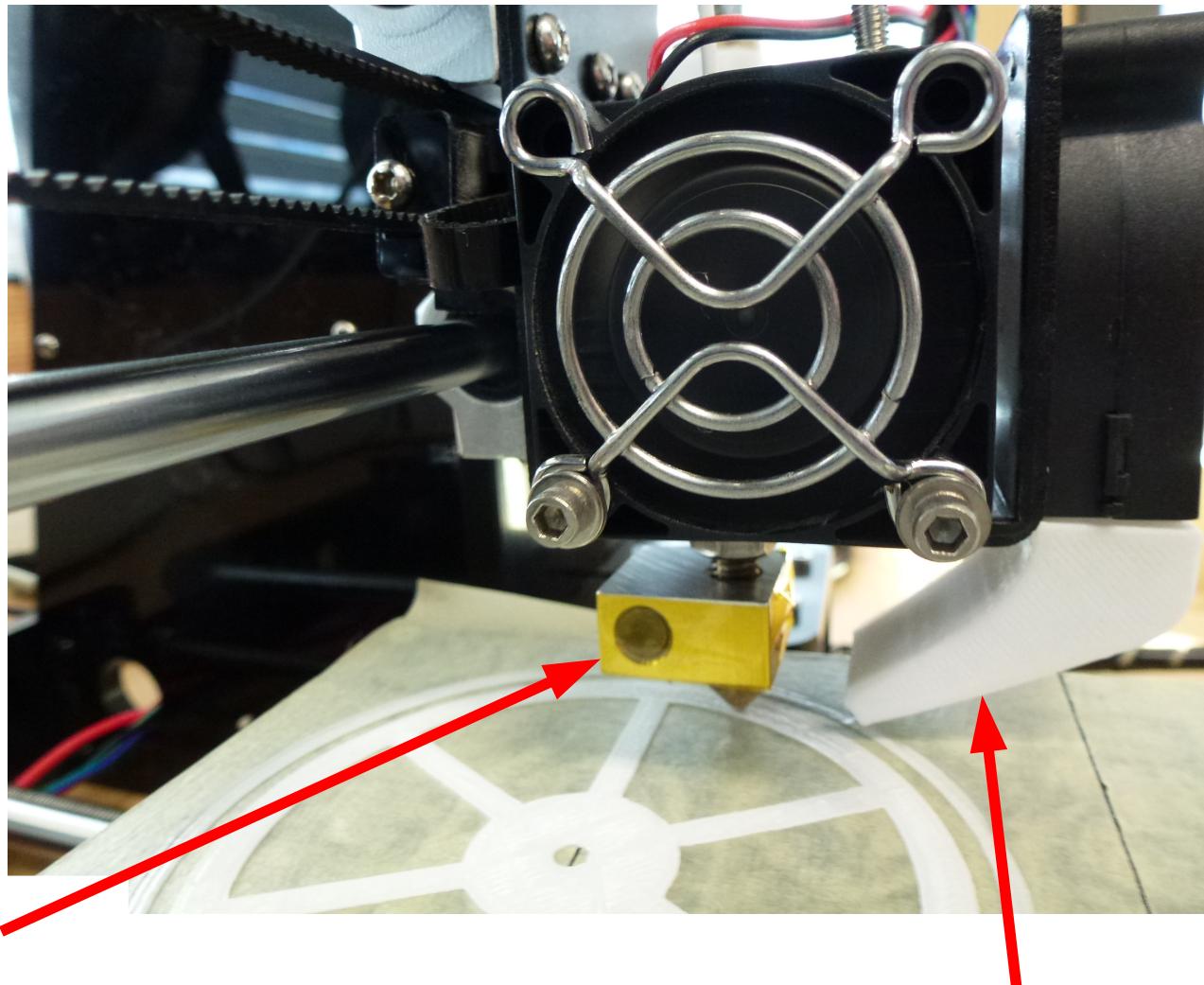


# 3D printers - extruder



Hot end

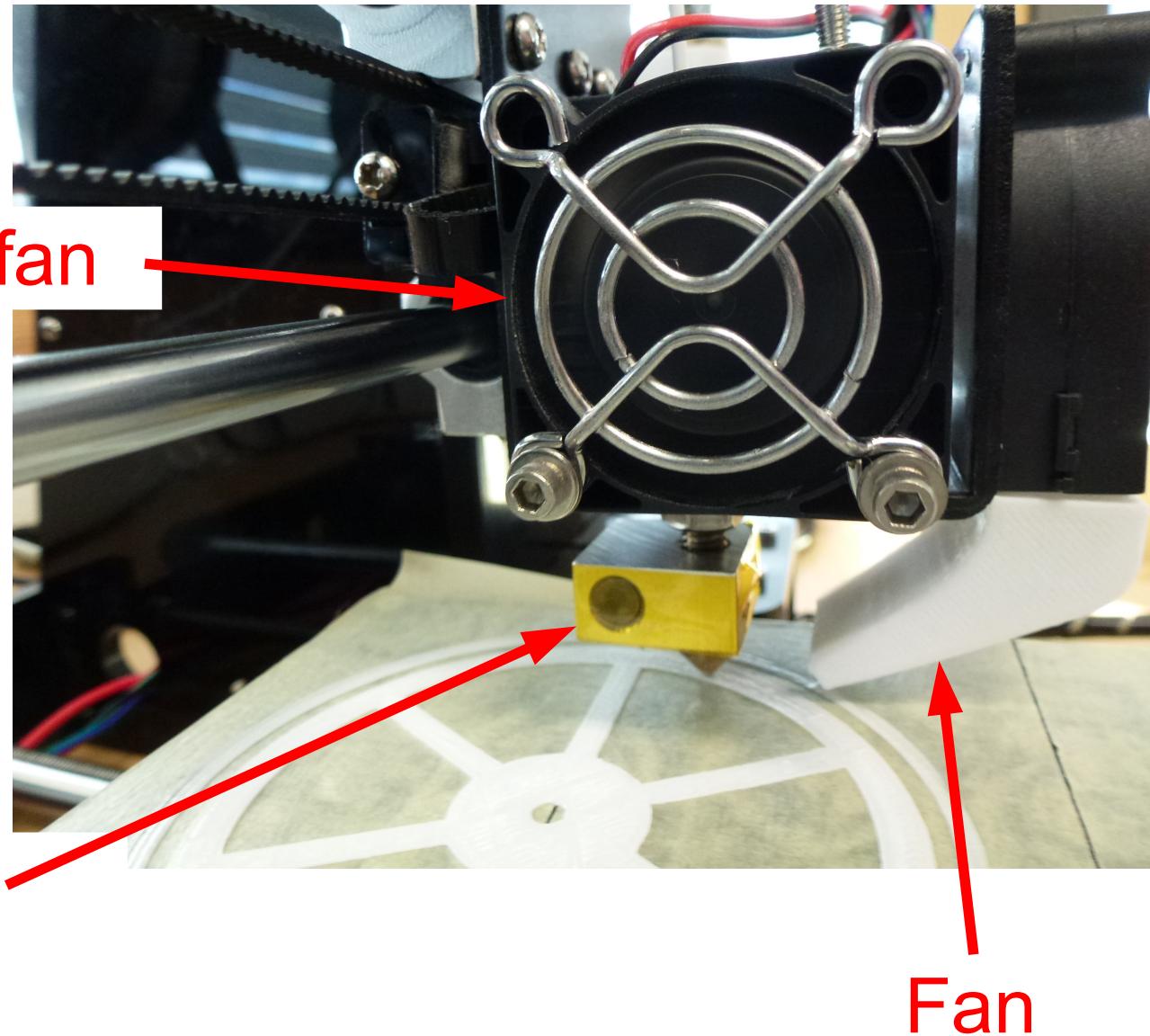
# 3D printers - extruder



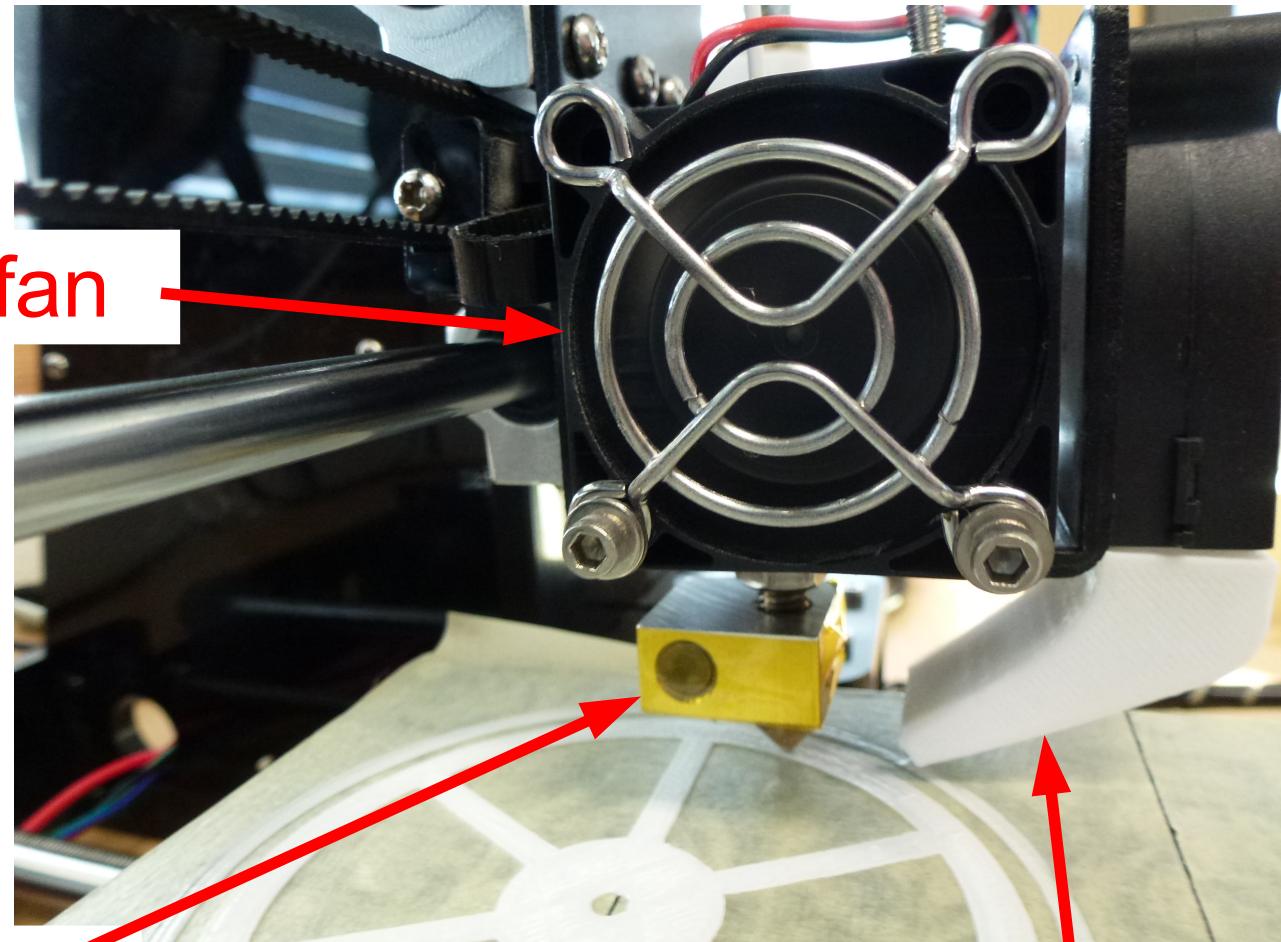
Hot end

Fan

# 3D printers - extruder



# 3D printers - extruder



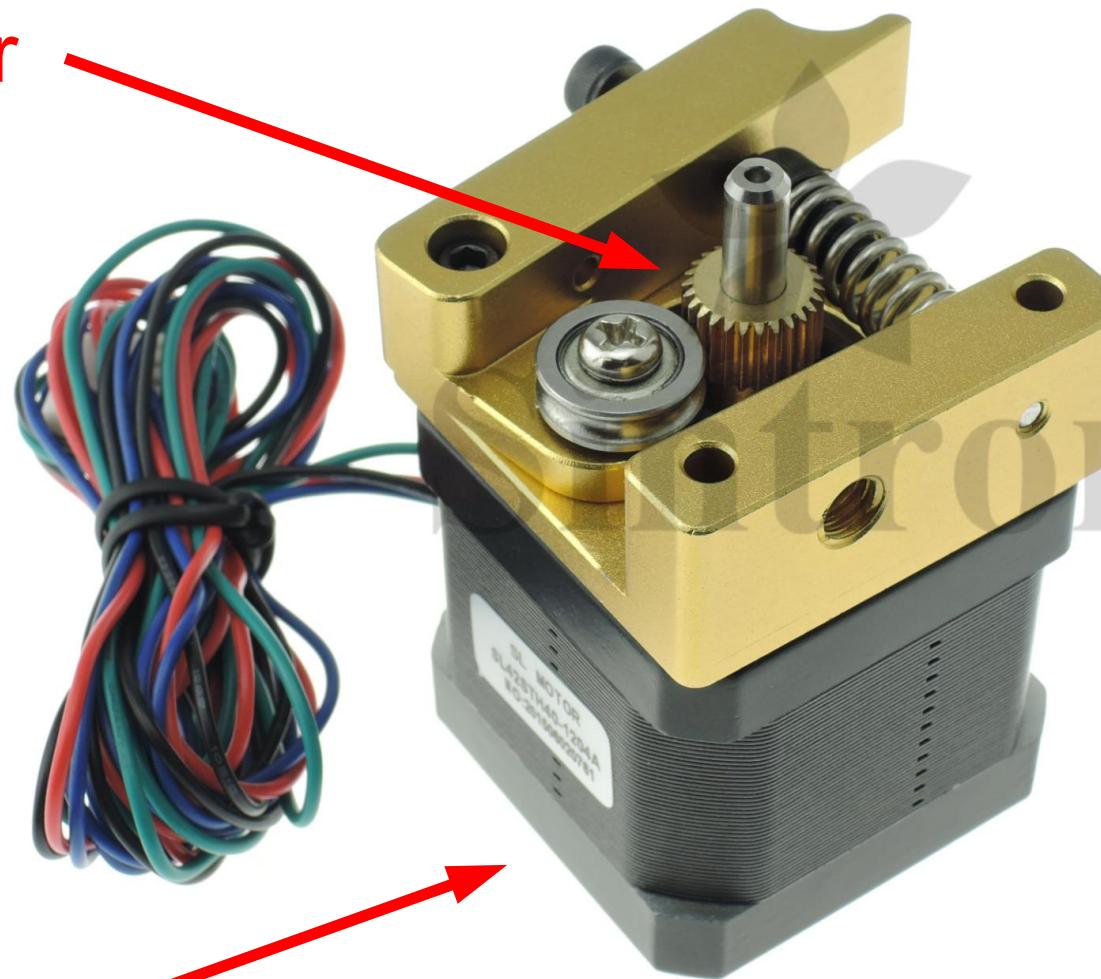
Extruder fan

Hot end

Fan (duct)

# 3D printers - extruder

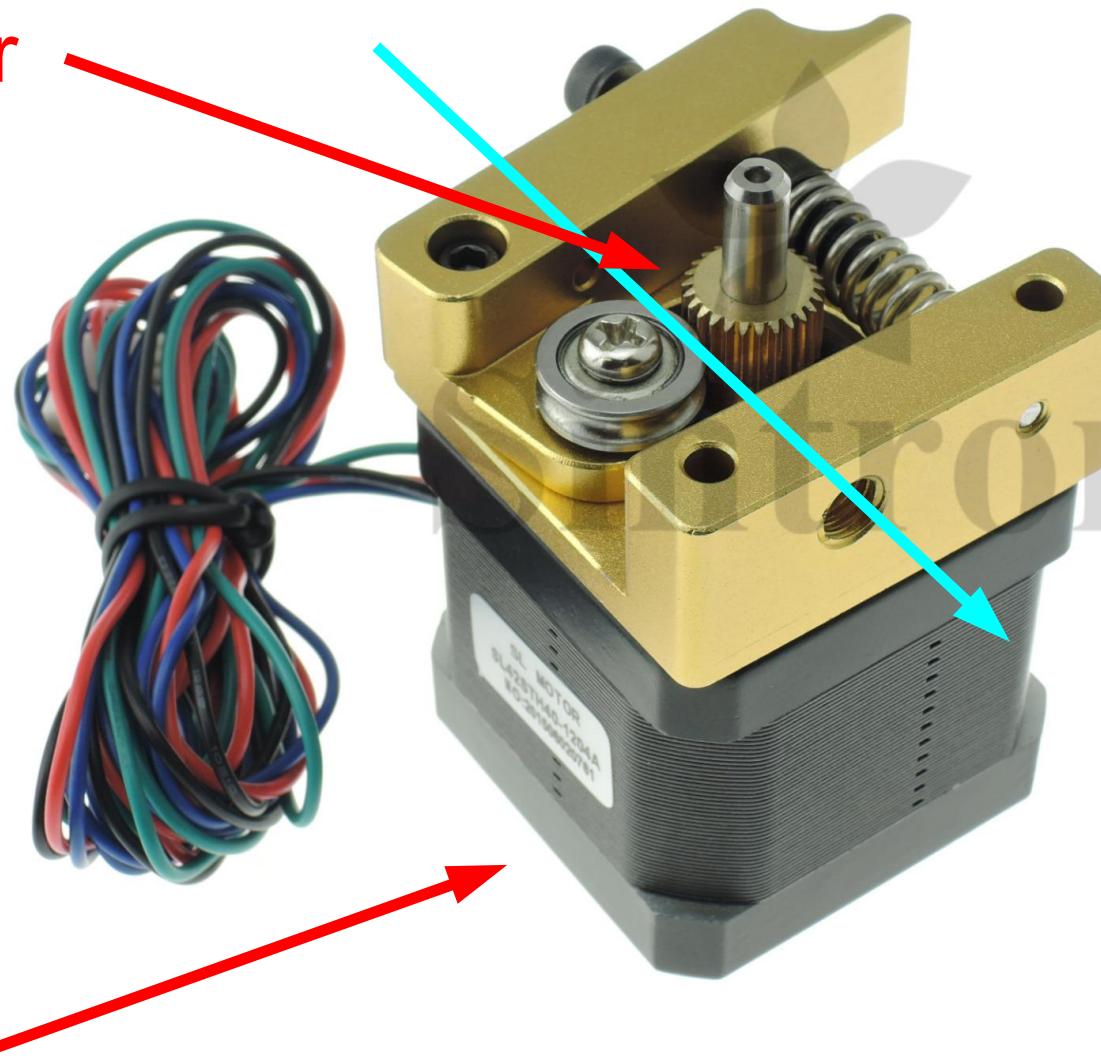
Drive gear



Motor

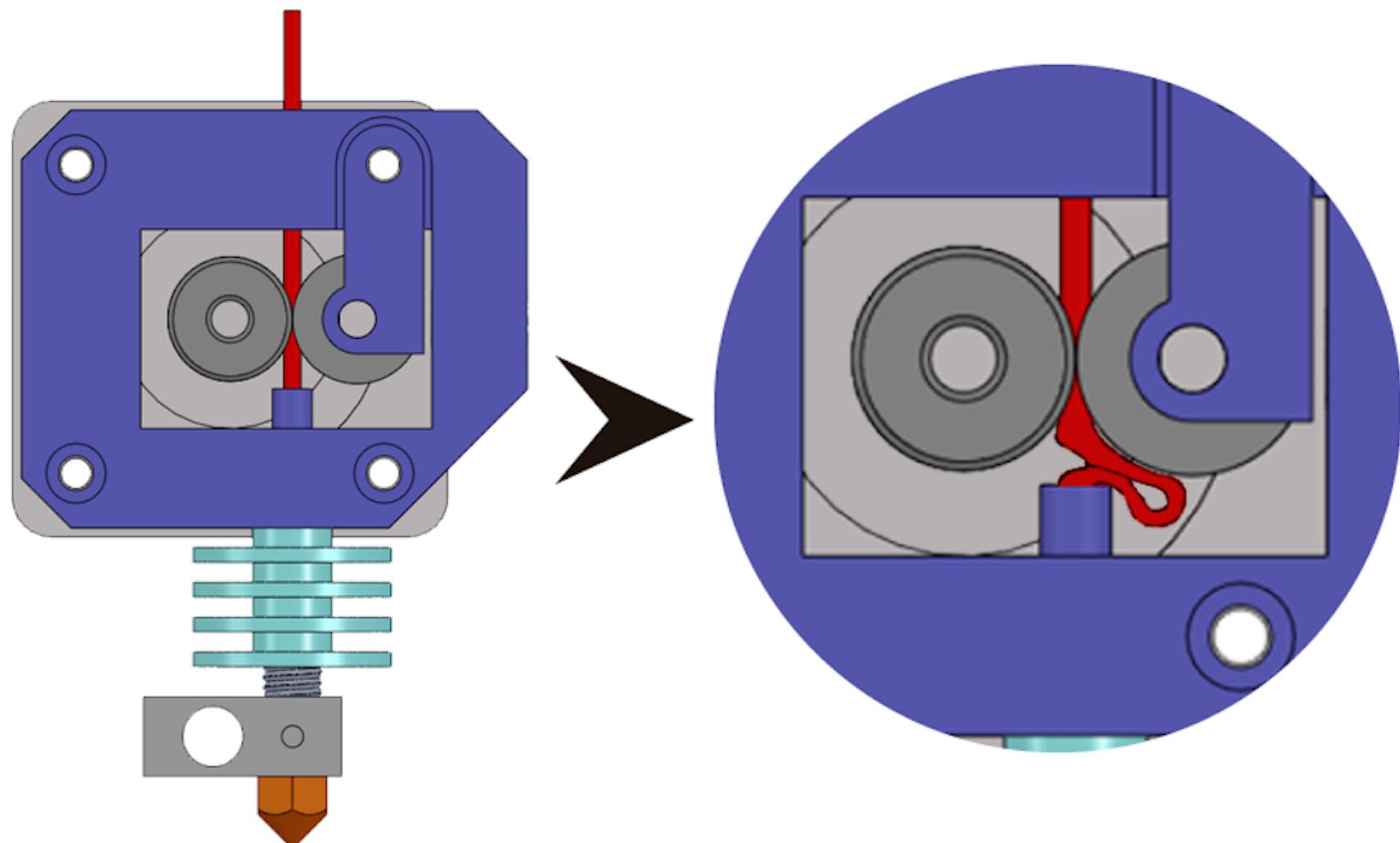
# 3D printers - extruder

Drive gear

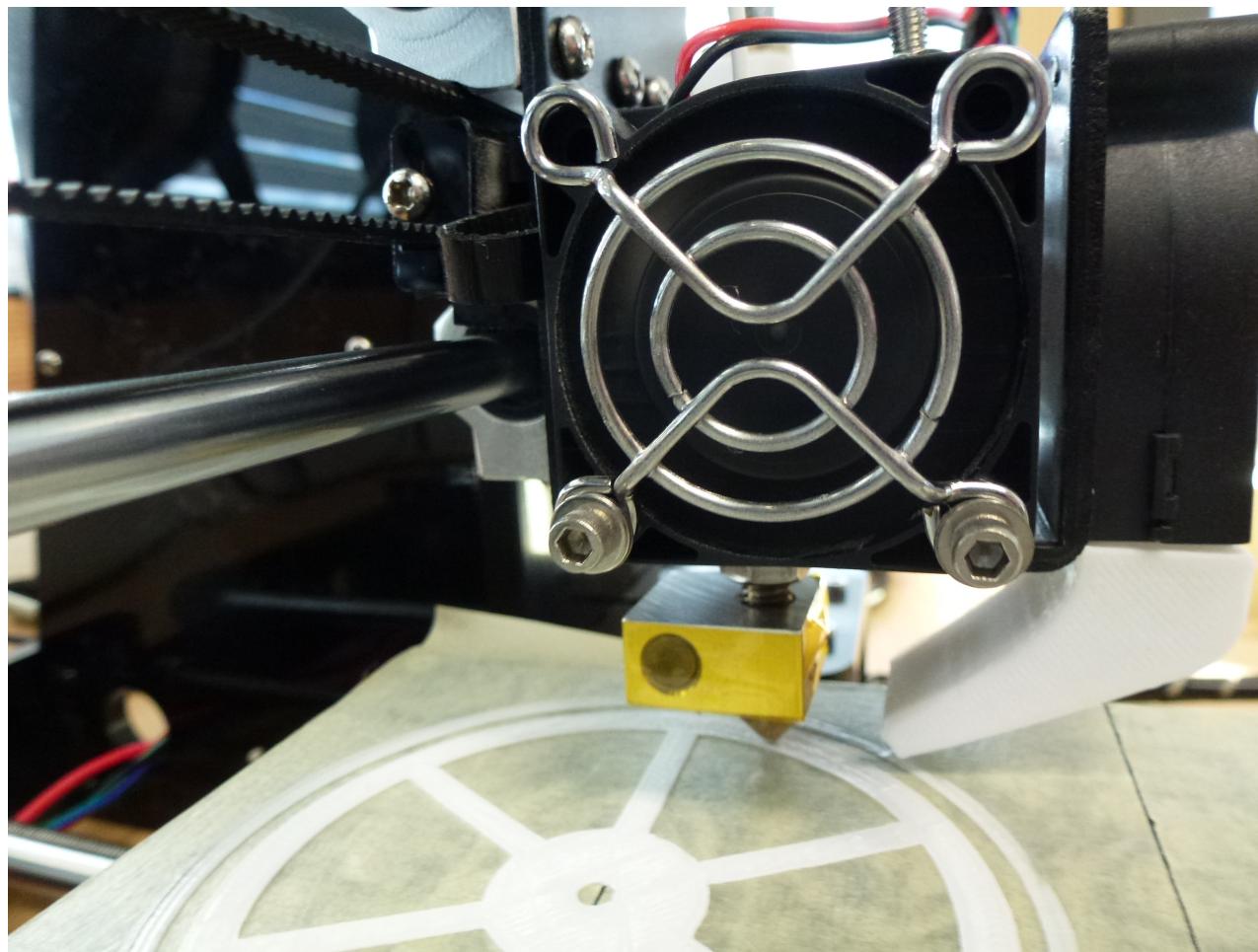


Motor

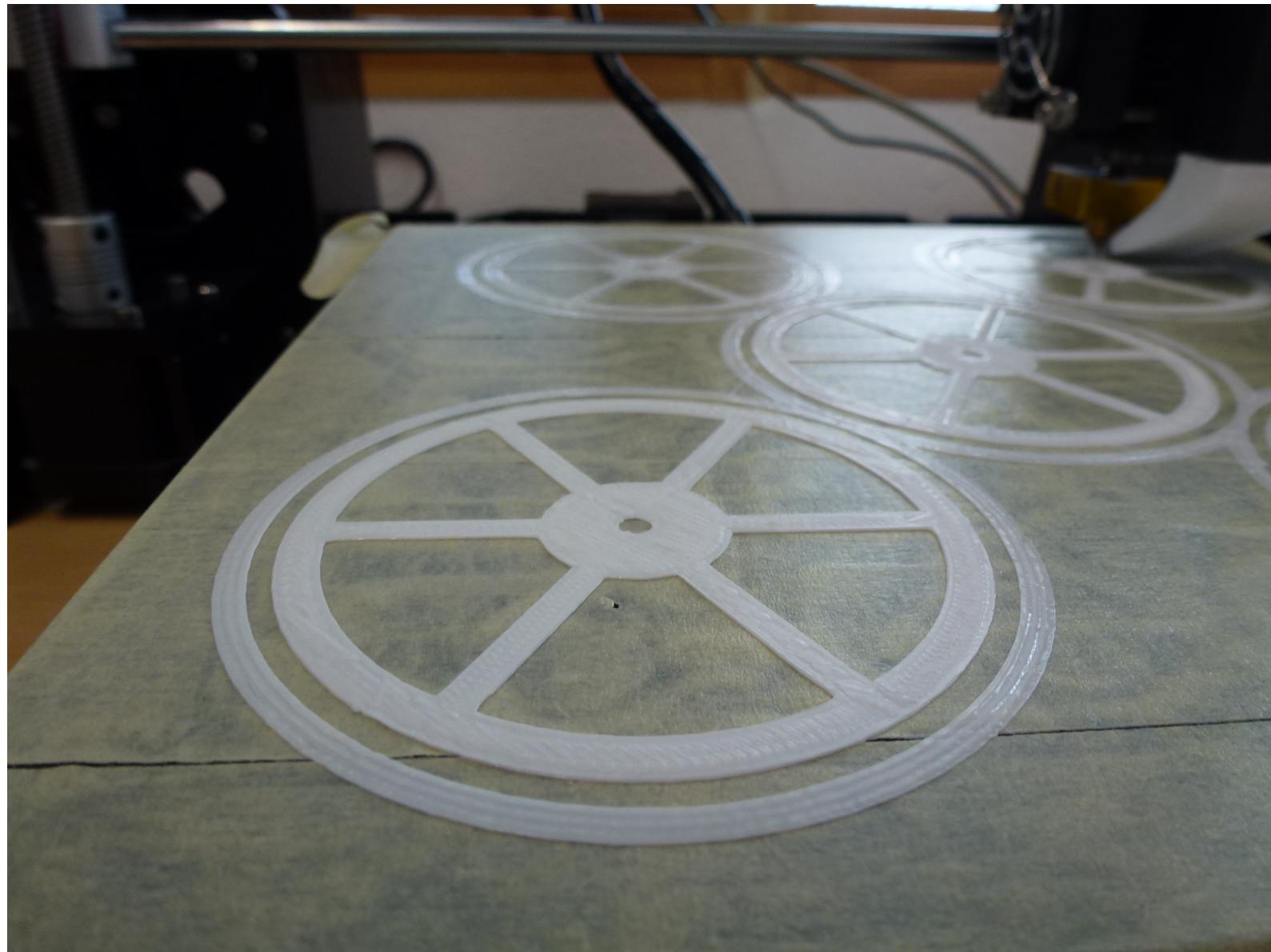
# 3D printers - extruder



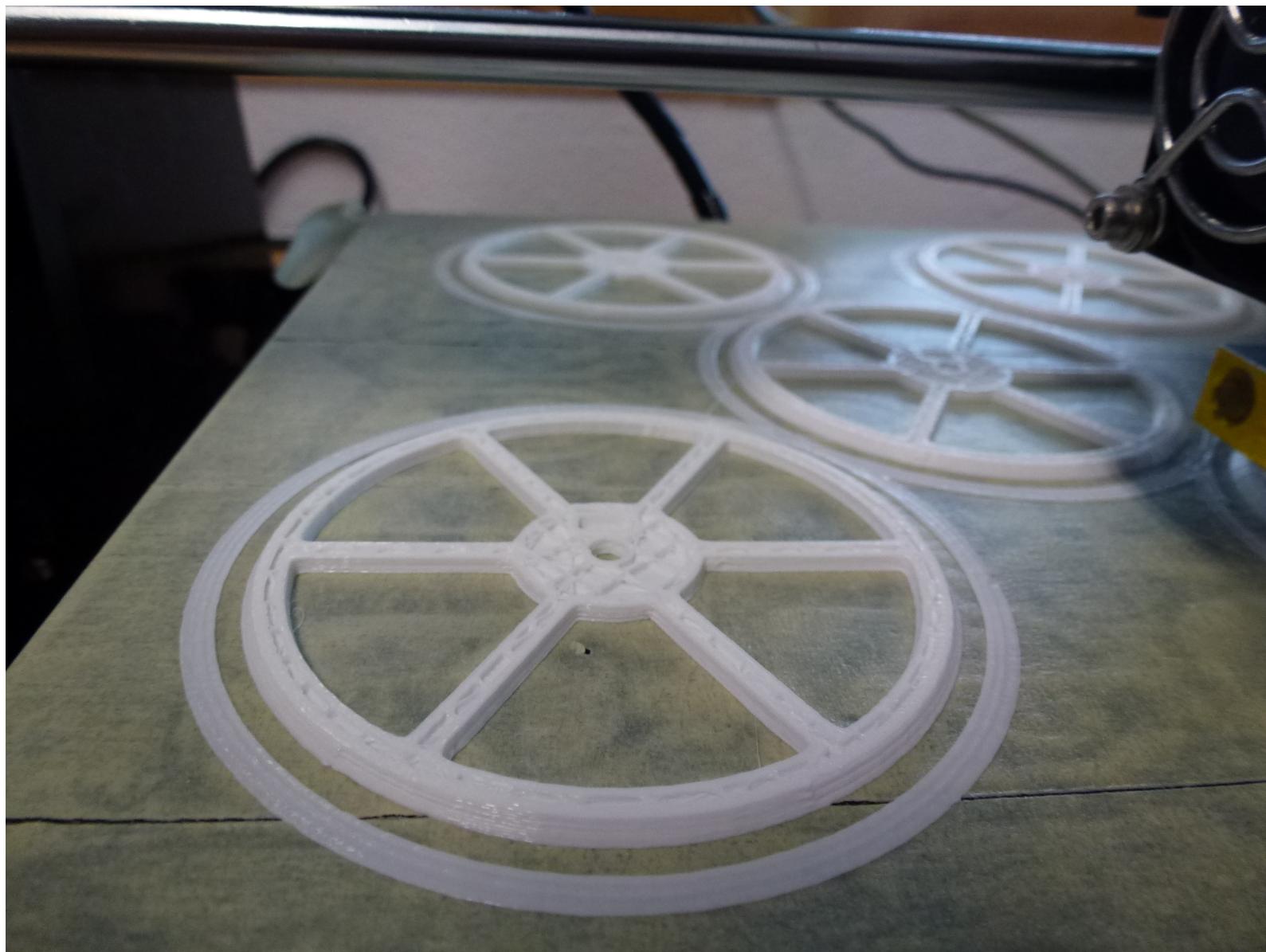
# 3D printers - printing



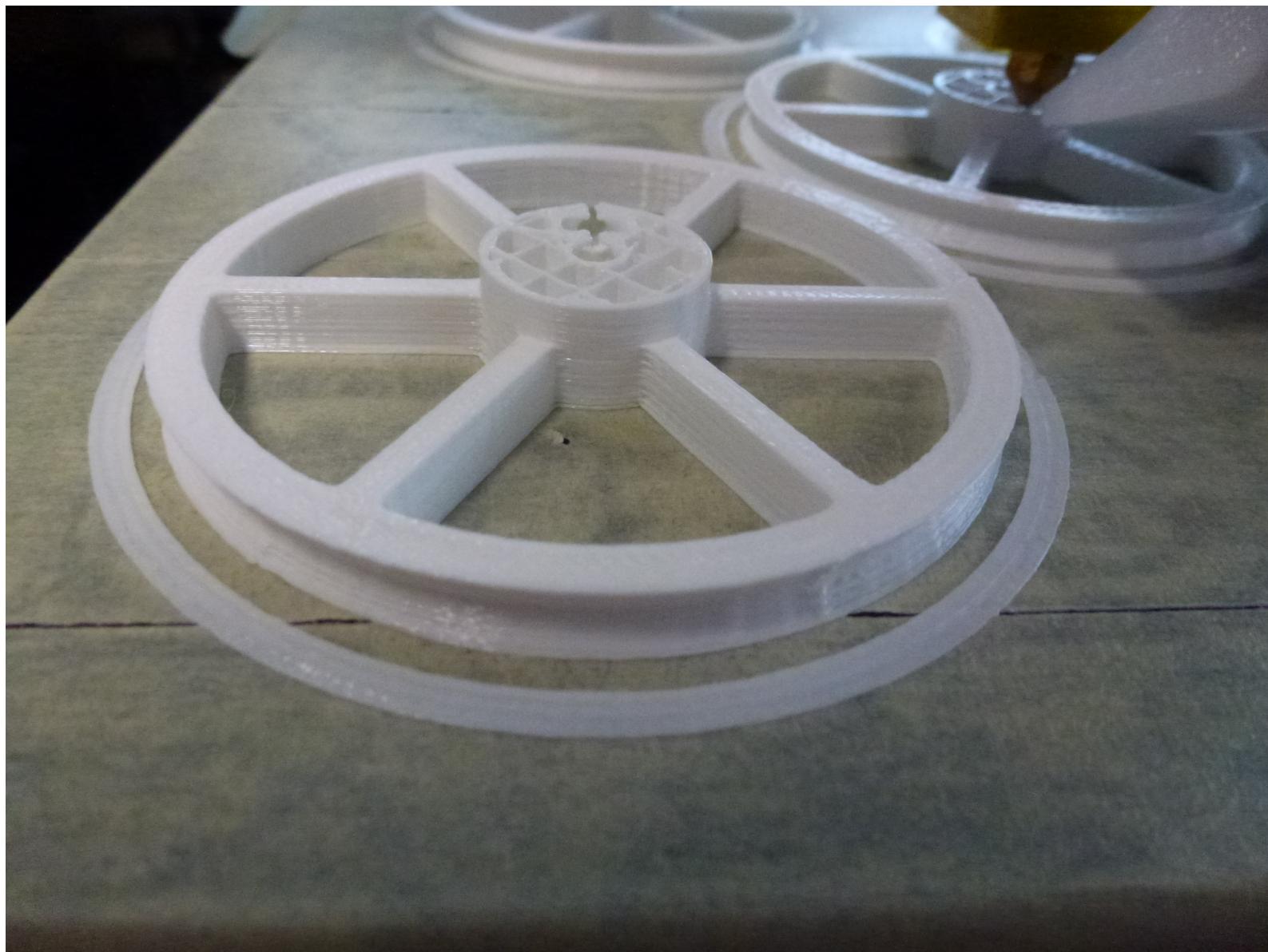
# 3D printers - printing



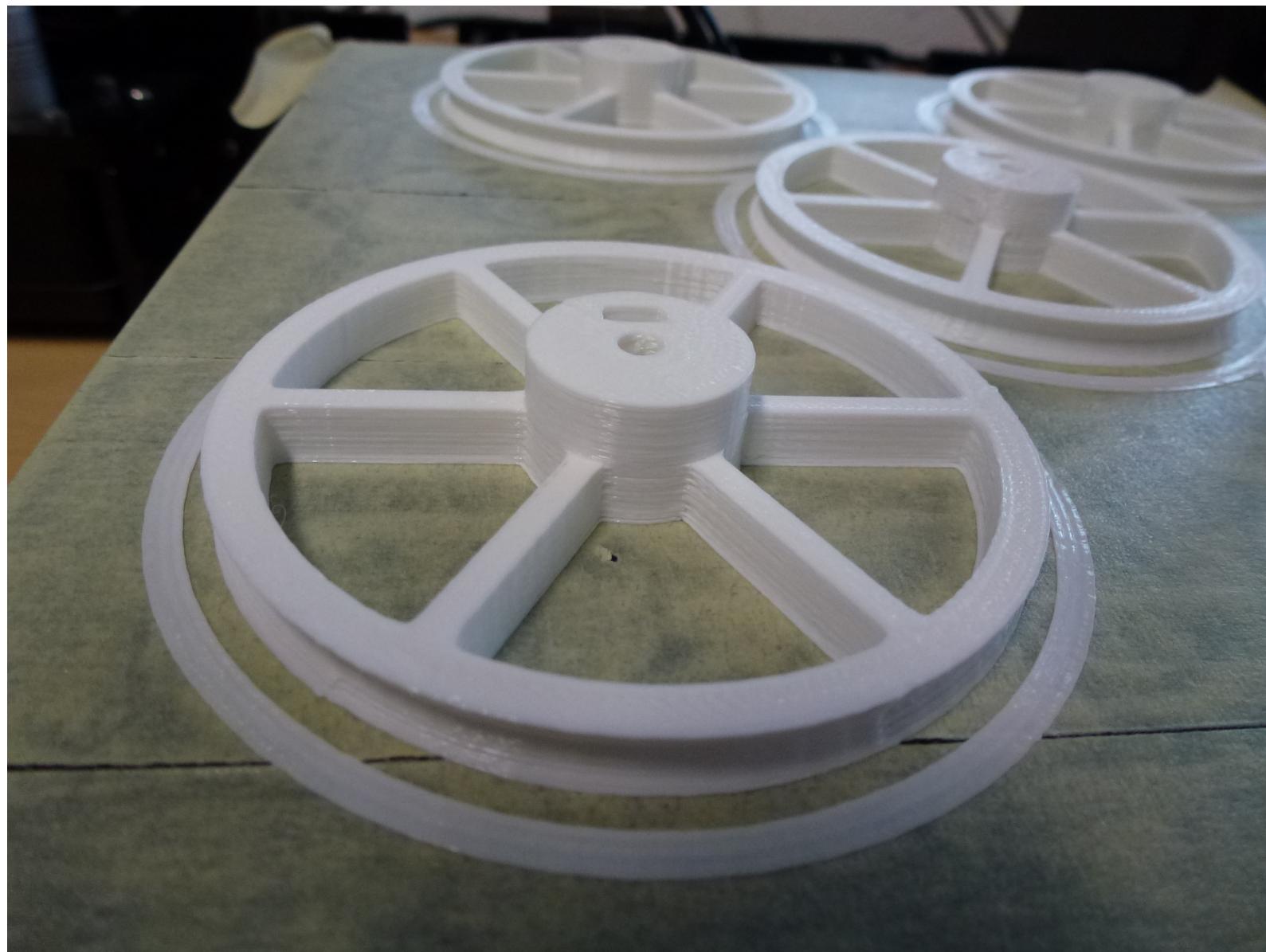
# 3D printers - printing



# 3D printers - printing

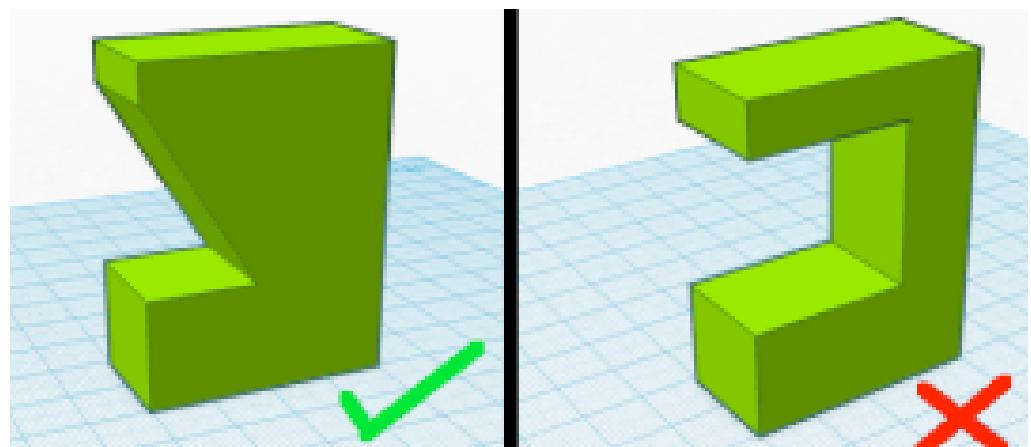
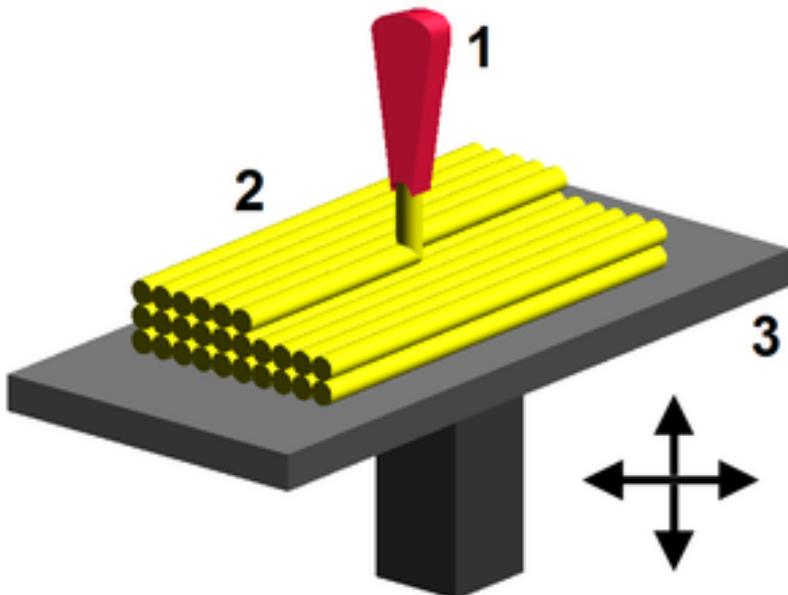


# 3D printers - printing

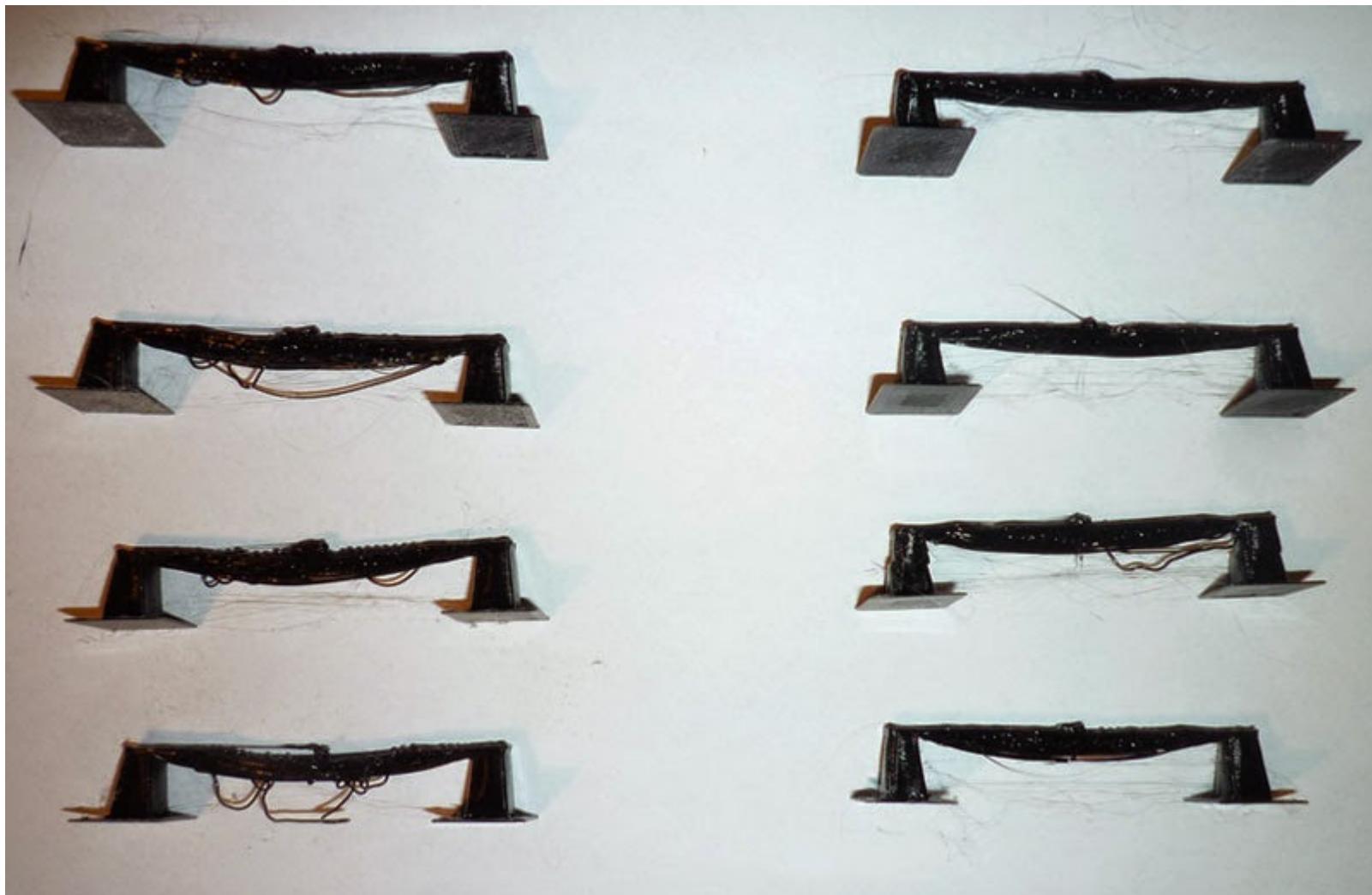


# Limitations

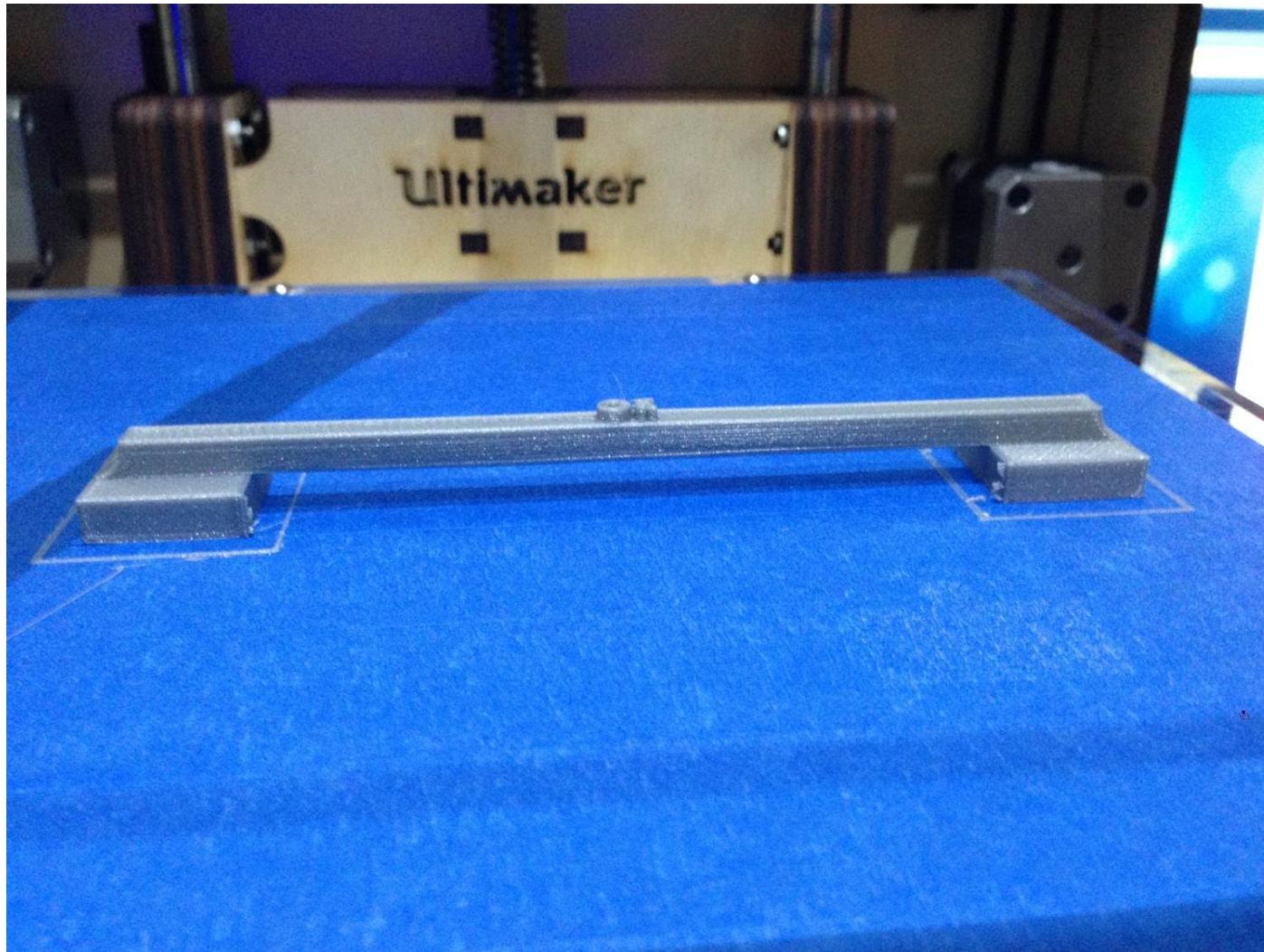
- Structures are printed layer by layer
  - They need a surface
  - Can't print in air (without support on the other side)



# Limitations



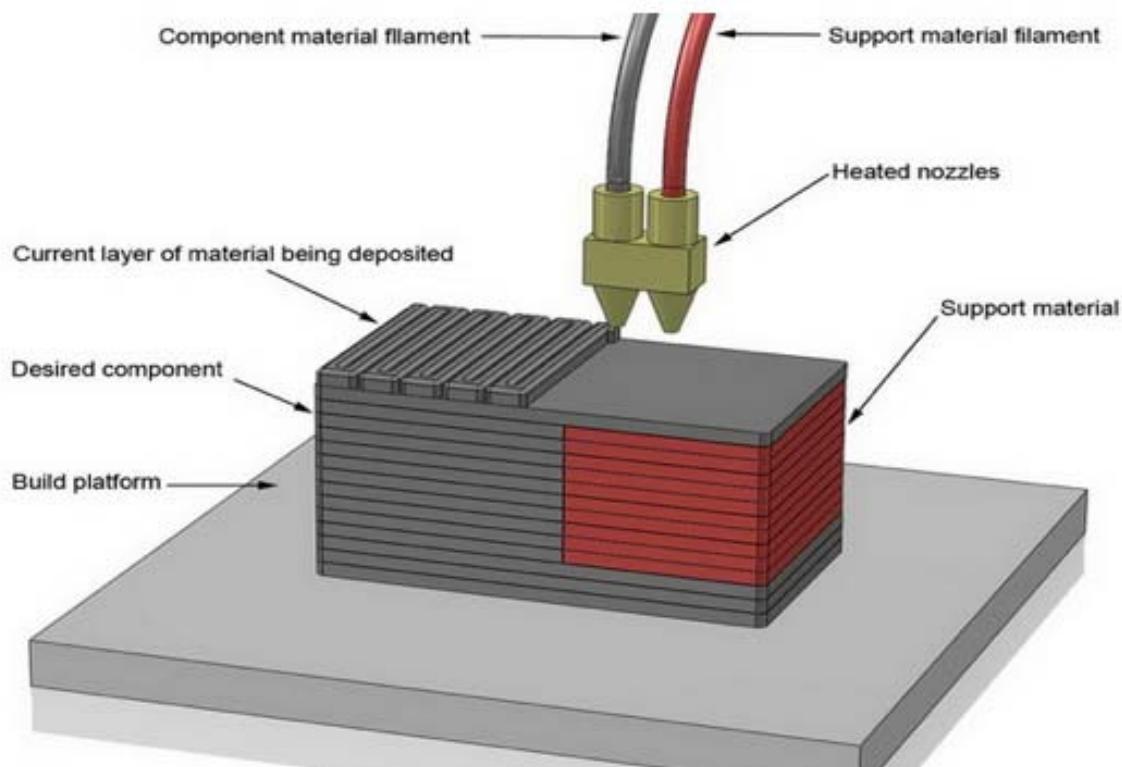
# Limitations



# Support material



# Support material



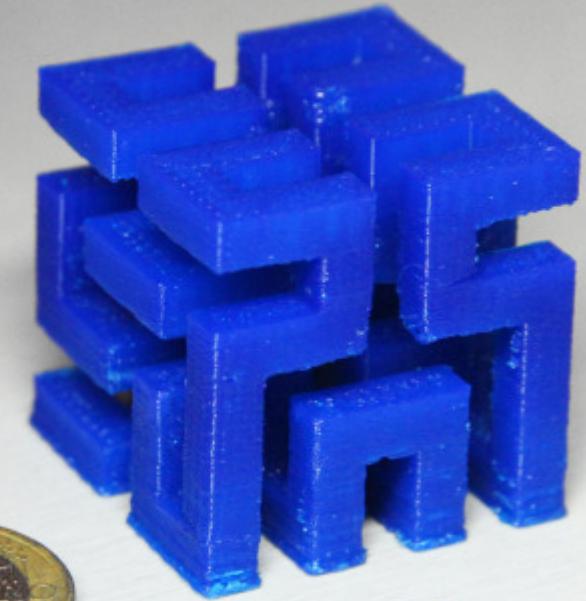
# Support material



1.



2.



3.

# Designing your components



# Designing your components

- Graphical CAD tools:
  - Unigraphics (\$8000 - \$15000)
  - Solidworks (\$3995)
  - FreeCAD (Free)
- Scripted CAD
  - OpenSCAD (Free)

# Designing your components

- Graphical CAD tools:
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  - Solidworks (\$3995)
  - FreeCAD (Free)
- Scripted CAD
  - OpenSCAD (Free)

# OpenSCAD

- Get OpenSCAD
  - <http://www.openscad.org/downloads.html>
    - MAC
    - Windows
    - Linux
- Very useful:
  - <http://www.openscad.org/cheatsheet/>

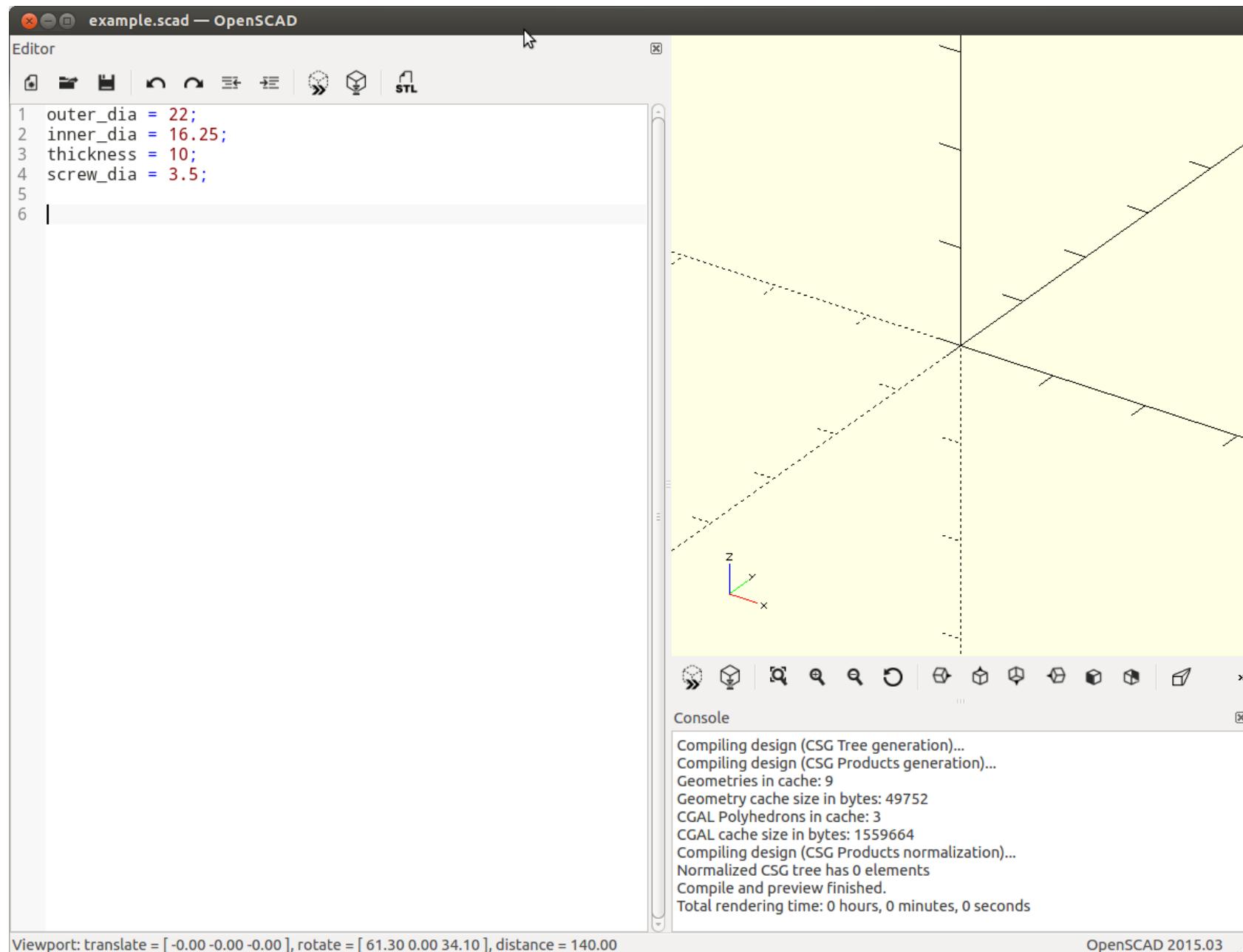
# OpenSCAD

- OpenSCAD works by combining and subtracting primitive objects.
  - constructive solid geometry
- Supports variables and math operations so your designs can be completely parameterized.

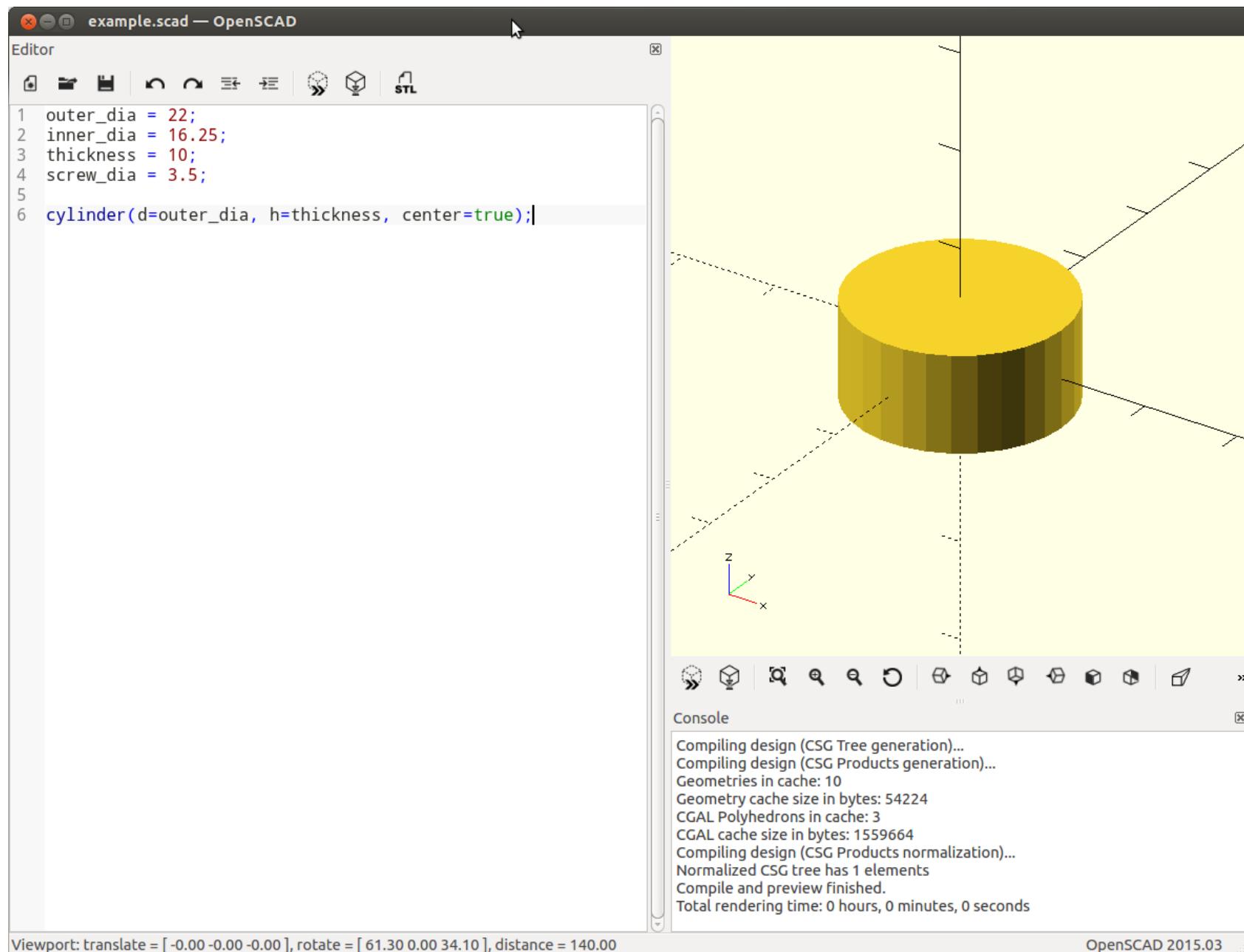
# Example: MPPT clamp



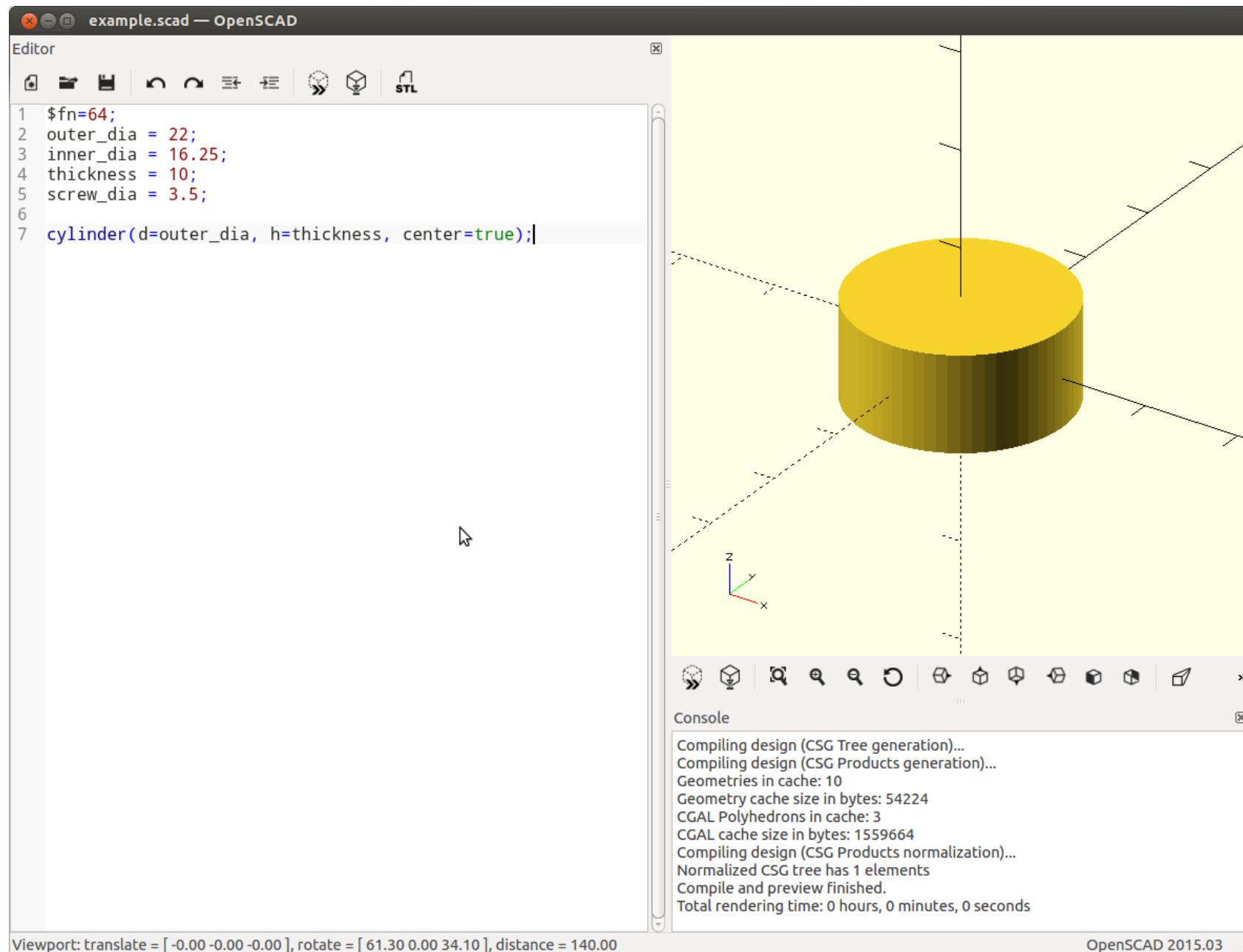
# Example: MPPT clamp



# Example: MPPT clamp



# Example: MPPT clamp



# Example: MPPT clamp

example.scad — OpenSCAD

Editor

```
1 $fn=64;
2 outer_dia = 22;
3 inner_dia = 16.25;
4 thickness = 10;
5 screw_dia = 3.5;
6
7 union()
8 {
9     cylinder(d=outer_dia, h=thickness, center=true);
10    translate([15,0,0])
11        cube([10,8,thickness],center=true);
12 }
13
```

Console

```
Compiling design (CSG Tree generation)...
Compiling design (CSG Products generation)...
Geometries in cache: 10
Geometry cache size in bytes: 54224
CGAL Polyhedrons in cache: 3
CGAL cache size in bytes: 1559664
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 = [ 61.30 0.00 34.10 ], distance = 140.00

OpenSCAD 2015.03

The screenshot shows the OpenSCAD application window. On the left, the 'Editor' tab displays the SCAD code for a MPPT clamp. The code defines variables for \$fn, outer\_dia, inner\_dia, thickness, and screw\_dia, and uses a union() command to combine a cylinder and a cube. On the right, the 'Preview' tab shows a 3D rendering of the clamp, which is a cylindrical base with a rectangular slot cut out of it. A coordinate system (x, y, z) is shown at the bottom left of the preview area. At the bottom, the 'Console' tab shows the output of the compilation process, including geometry statistics and compilation logs.

# Example: MPPT clamp

example.scad — OpenSCAD

Editor

```
1 $fn=64;
2 outer_dia = 22;
3 inner_dia = 16.25;
4 thickness = 10;
5 screw_dia = 3.5;
6
7 difference()
8 {
9     union()
10    {
11        cylinder(d=outer_dia, h=thickness, center=true);
12        translate([15,0,0])
13            cube([10,8,thickness],center=true);
14    }
15
16    cylinder(d=inner_dia, h=thickness+0.1, center=true);
17}
```

Console

```
Compiling design (CSG Tree generation)...
Compiling design (CSG Products generation)...
Geometries in cache: 10
Geometry cache size in bytes: 54224
CGAL Polyhedrons in cache: 3
CGAL cache size in bytes: 1559664
Compiling design (CSG Products normalization)...
Normalized CSG tree has 3 elements
Compile and preview finished.
Total rendering time: 0 hours, 0 minutes, 0 seconds
```

Viewport: translate = [-0.00 -0.00 -0.00], rotate = [61.30 0.00 34.10], distance = 140.00

OpenSCAD 2015.03

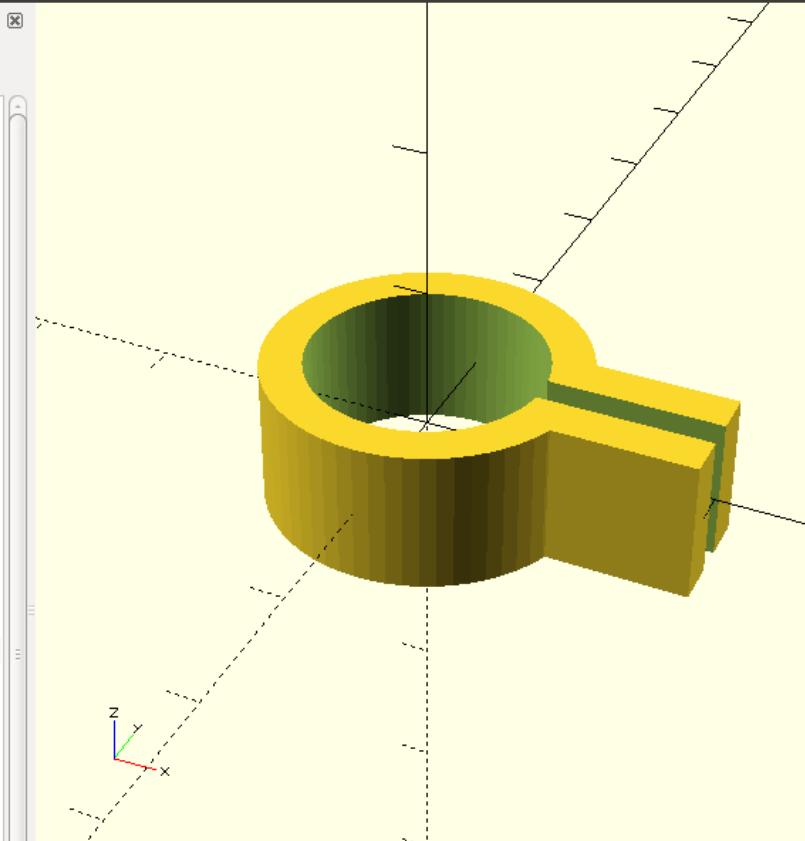
The image shows the OpenSCAD application window. On the left is the 'Editor' pane containing the SCAD source code for a MPPT clamp. The code defines variables for \$fn, outer\_dia, inner\_dia, thickness, and screw\_dia, and uses the difference() and union() operations to create the clamp's shape. In the center is the 'Preview' pane displaying a 3D model of the clamp, which is yellow with a green cylindrical hole. A coordinate system (x, y, z) is shown at the bottom left of the preview area. At the bottom is the 'Console' pane, which outputs the compilation process and performance statistics. The status bar at the bottom shows the current view parameters.

# Example: MPPT clamp

example.scad — OpenSCAD

Editor

```
1 $fn=64;
2 outer_dia = 22;
3 inner_dia = 16.25;
4 thickness = 10;
5 screw_dia = 3.5;
6
7 difference()
8 {
9     union()
10    {
11        cylinder(d=outer_dia, h=thickness, center=true);
12        translate([15,0,0])
13            cube([10,8,thickness],center=true);
14    }
15
16    cylinder(d=inner_dia, h=thickness+0.1, center=true);
17
18    translate([13,0,0])
19        cube([14+0.1,2,thickness+0.1],center=true);
20
21 }
```



Console

```
CGAL cache size in bytes: 2213840
Total rendering time: 0 hours, 0 minutes, 0 seconds
Top level object is a 3D object:
Simple: yes
Vertices: 252
Halfedges: 756
Edges: 378
Halffacets: 256
Facets: 128
Volumes: 2
Rendering finished.
```

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

OpenSCAD 2015.03

# Example: MPPT clamp

example.scad — OpenSCAD

Editor

```
1 $fn=64;
2 outer_dia = 22;
3 inner_dia = 16.25;
4 thickness = 10;
5 screw_dia = 3.5;
6
7 difference()
8 {
9     union()
10    {
11        cylinder(d=outer_dia, h=thickness, center=true);
12        translate([15,0,0])
13            cube([10,8,thickness],center=true);
14    }
15
16    cylinder(d=inner_dia, h=thickness+0.1, center=true);
17
18    translate([13,0,0])
19        cube([14+0.1,2,thickness+0.1],center=true);
20
21    translate([15,0,0])
22        rotate([90,90,0])
23            cylinder(d=screw_dia, h=20, center=true);
24
25 }
```

Console

```
Compiling design (CSG Tree generation)...
Compiling design (CSG Products generation)...
Geometries in cache: 13
Geometry cache size in bytes: 56408
CGAL Polyhedrons in cache: 5
CGAL cache size in bytes: 2213840
Compiling design (CSG Products normalization)...
Normalized CSG tree has 6 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

# Example: MPPT clamp

- Exporting your file for printing:
  - Press F6 for full render
  - Export the STL file.
- Send your STL files to the printer assistants, they will check your design and print it.
- Printer assistants: we have to plan a meeting