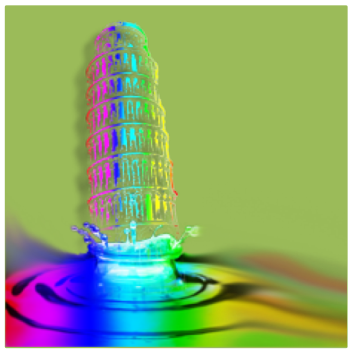
CENTRO E. PIAGGIO
Bioengineering and Robotics Research Center



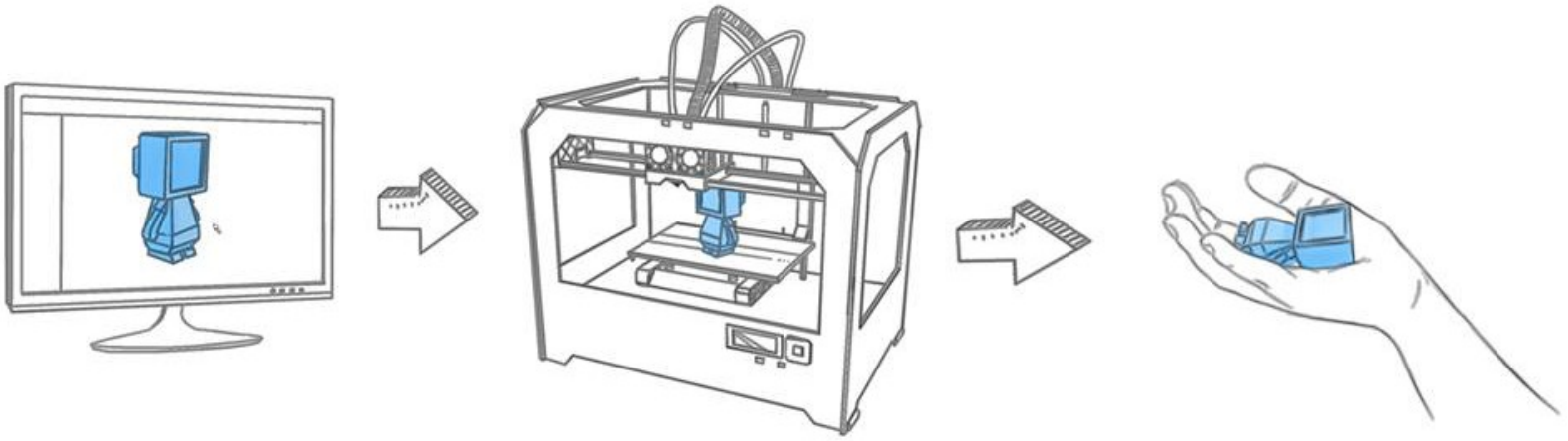
Additive manufacturing

Cura Software

Deacutis.aurora@gmail.com

Additive Manufacturing (AM)

- The process of joining material to make object from a 3D digital model data, usually layer upon layer, as opposed to subtractive manufacturing methodologies.



AM = Rapid Prototyping (RP)

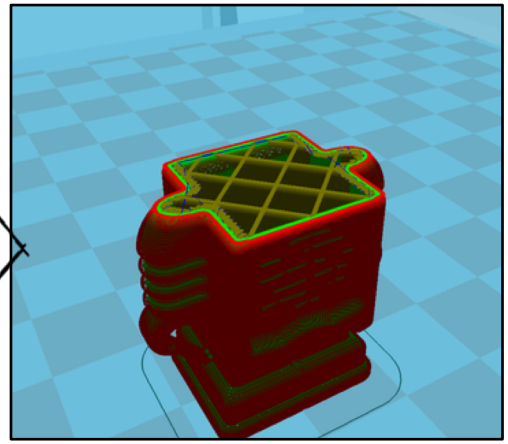
Avaible RP techonolgies in TE

- RP technologies based on the extrusion process;
- Ink-jet printing;
- RP technologies based on the light-curing process;
- Hybrid RP technologies

From design to object (from .STL to GCODE)

.STL model

Slicer software



GCode

Introduction to Ultimaker Cura Software

What does Cura software do?

Cura slices 3D models. It translates the 3D STL file into a format that the printer can understand (Gcode). Cura takes the 3D model and works out how those layers are placed on the print bed and creates a set of instructions for the printer to follow layer on layer.

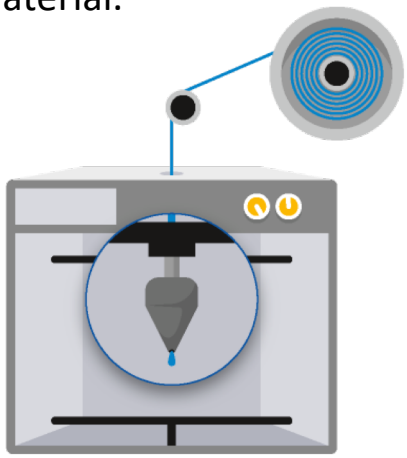
These instructions are the G-Code, a text document that ends with the file extension .gcode. Open the file and you'll actually be able to read through quite a bit of the code and understand what it's telling the printer to do.

```
G0 F7200 X19.698 Y28.262 Z.36
G1 F1500 E0
G1 F1350 X22.467 Y26.175 E0.15654
G1 X23.338 Y25.568 E0.20447
G1 X24.246 Y25.027 E0.25218
```

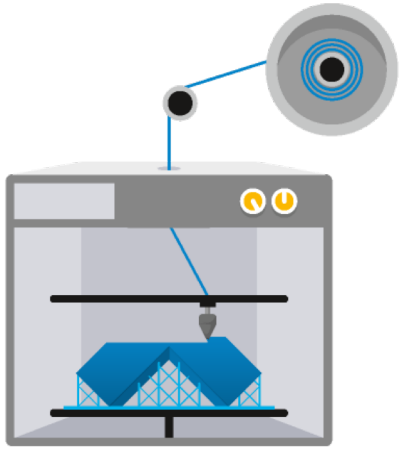
Fused Deposition Modelling (FDM)

FDM is a filament-based technology where a temperature-controlled head extrudes a thermoplastic material layer by layer onto a build platform. A support structure is created where needed and built in a water-soluble material.

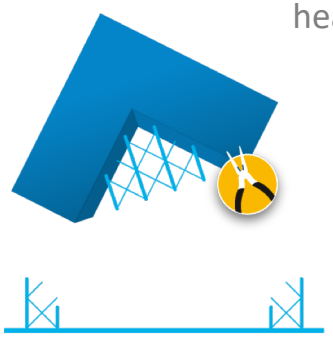
The 3D model



The thermoplastic modeling filament is heated to a semi-liquid state



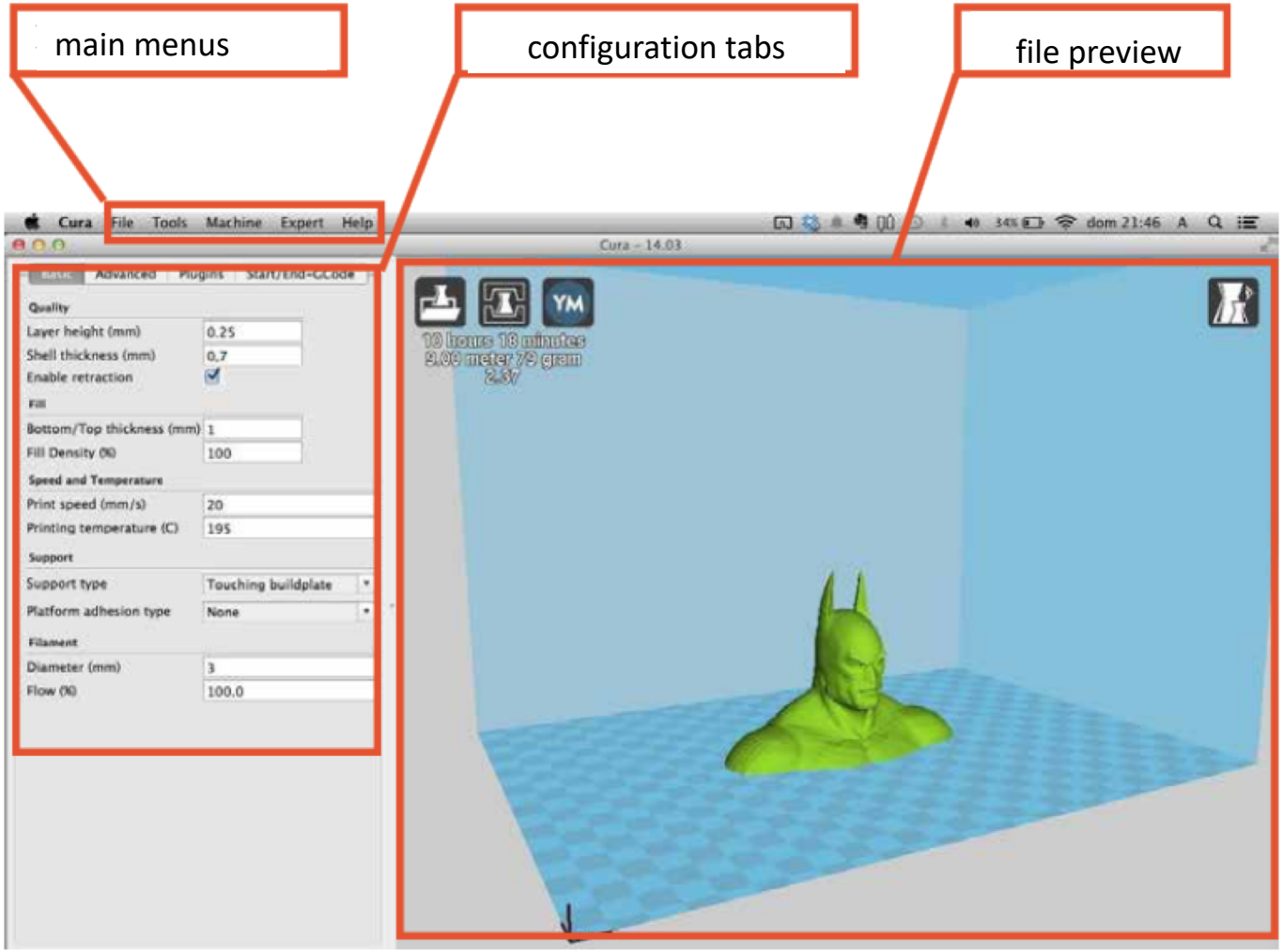
The extrusion head 'draws' the part and its support one layer at a time



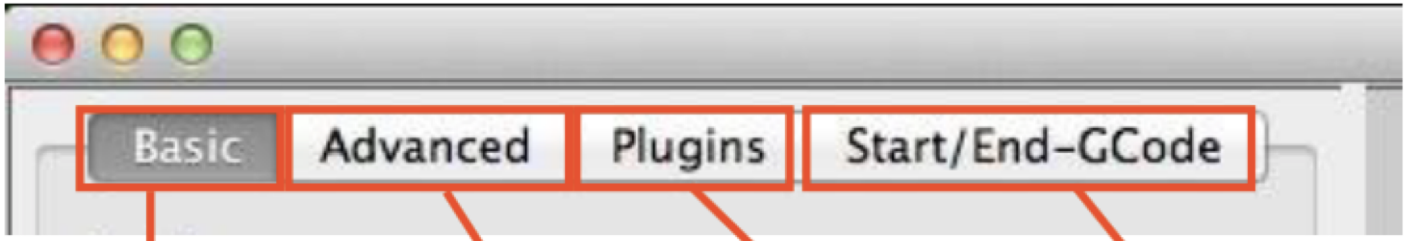
The support is removed

The part is finished

Cura Interface



Configuration tabs



Here are all the basic settings. These parameters vary from piece to piece, and they define the print quality in a sensible way.

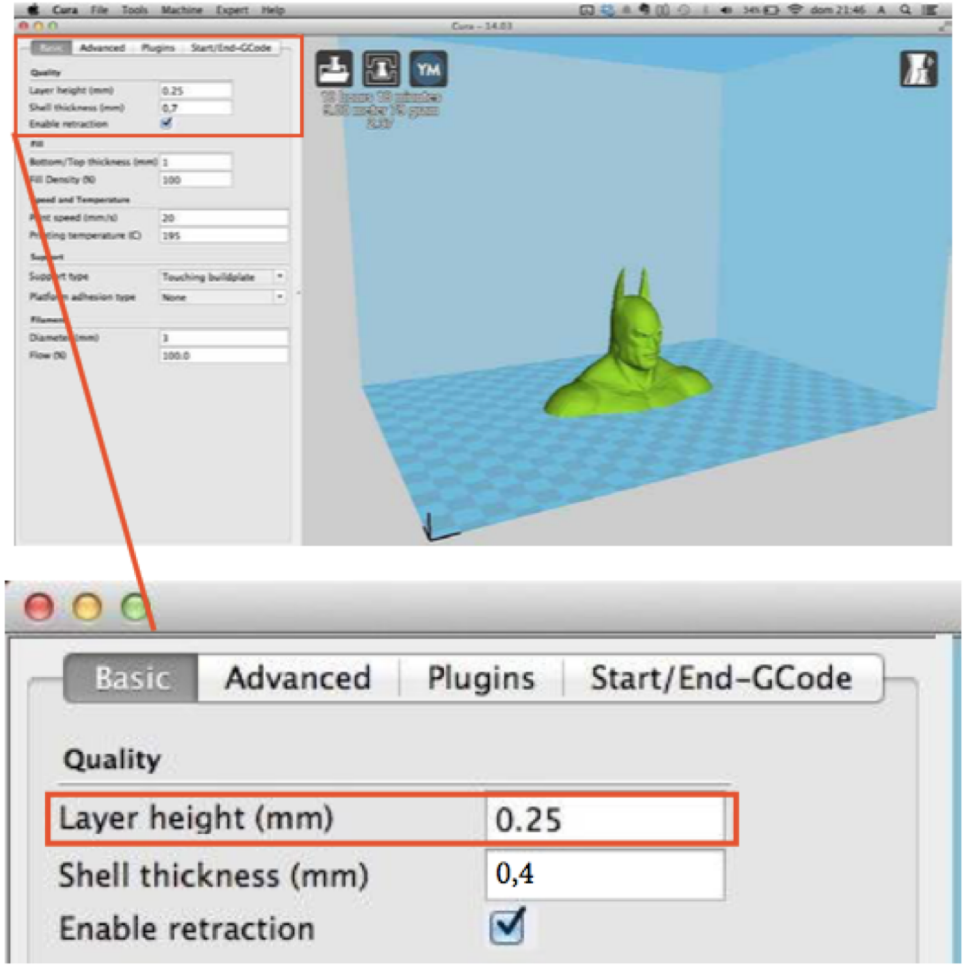
Most advanced but less variable configurations. Once they are set it is less common to change them.

Additional plugins can be installed and configured.
In the latest version are already present: Pause at Height Tweak at Z

Here you can manually enter GCode instructions that the machine will run at the beginning and end of the print.

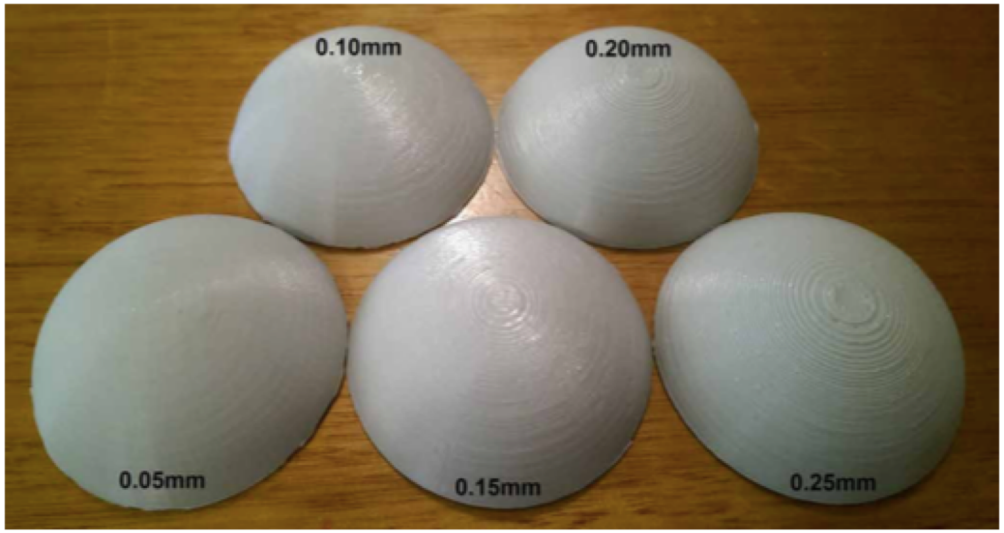
Basic - Quality

LAYER HEIGHT

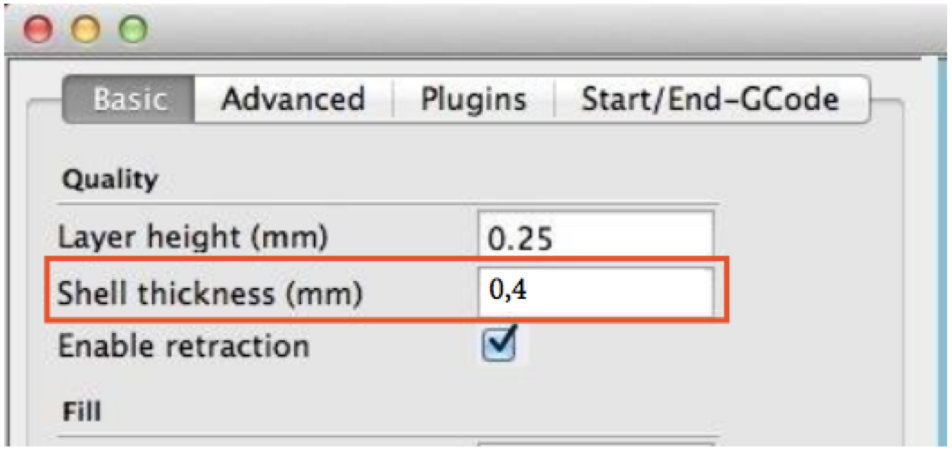


- LAYER HEIGHT defines the surface finish of the printed part and the printing times.
- High values of layer height are possible but with a decrease in the final quality of the workpiece.
- The print time is directly proportional to the height of the layer.
- This setting will vary depending on the printer type

Differences between layer height in time-quality ratio



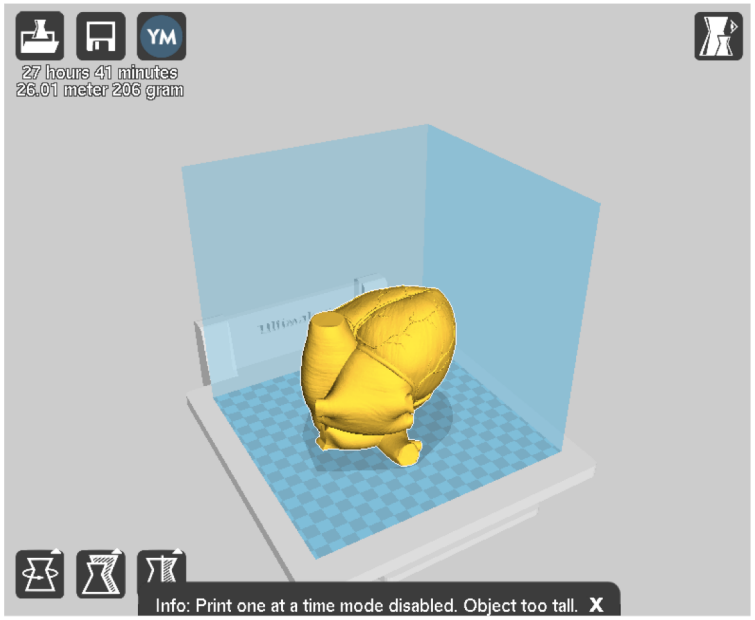
Basic - Quality
SHELL THICKNESS

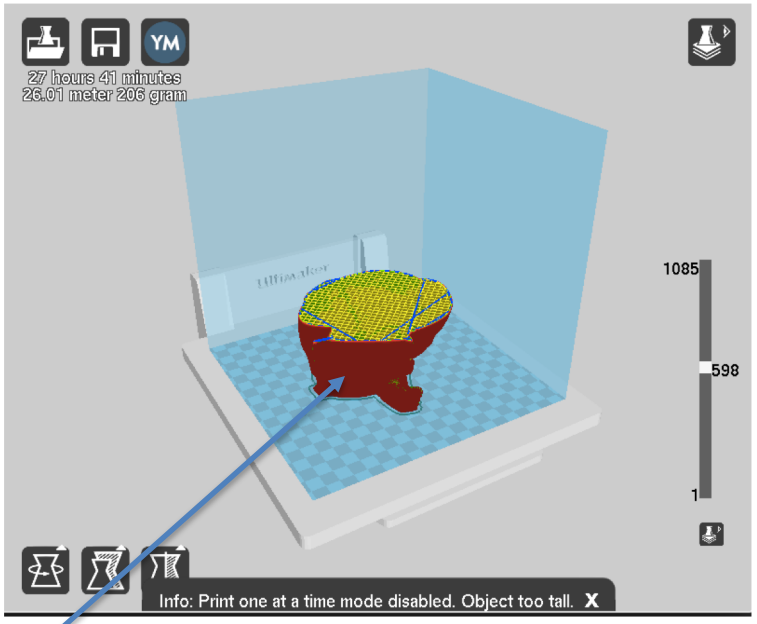


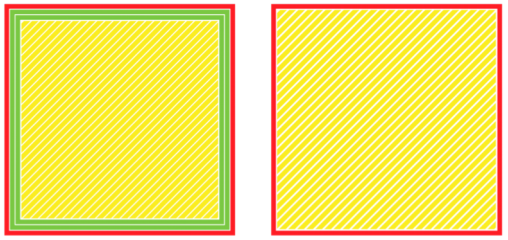
- This refers to the thickness of any walls that your model might have, as well as the thickness of the bottom and top layer. The Shell has to be an integer multiple (1x, 2x, 3x, etc.) of the nozzle diameter.
- Thicker wall will create a stronger print and will generally get a better finish.
- However, increasing the thickness of the "shell" causes a significant increase in the printing time

Basic - Quality

SHELL THICKNESS



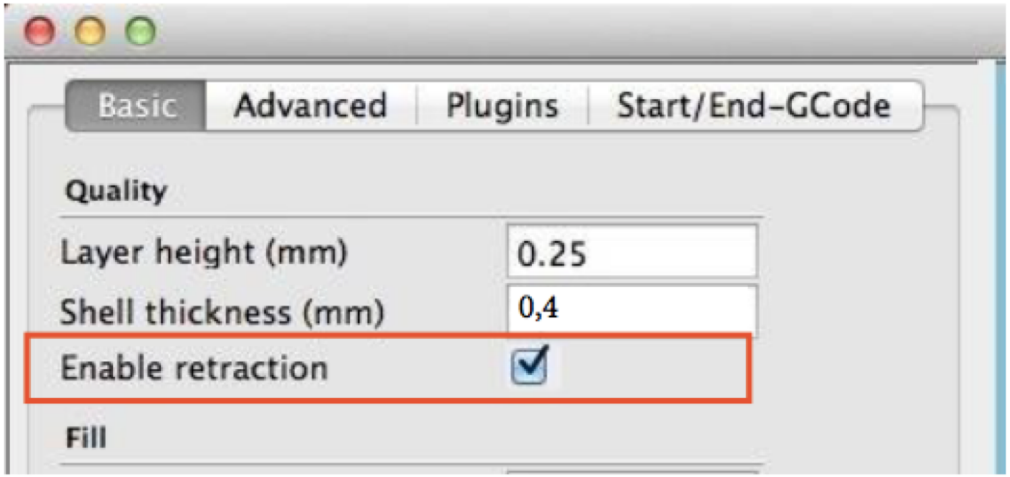




Shell

The model on the left has 3 walls,
the model on the right has a single wall.

Basic - Quality
ENABLE RETRACTION



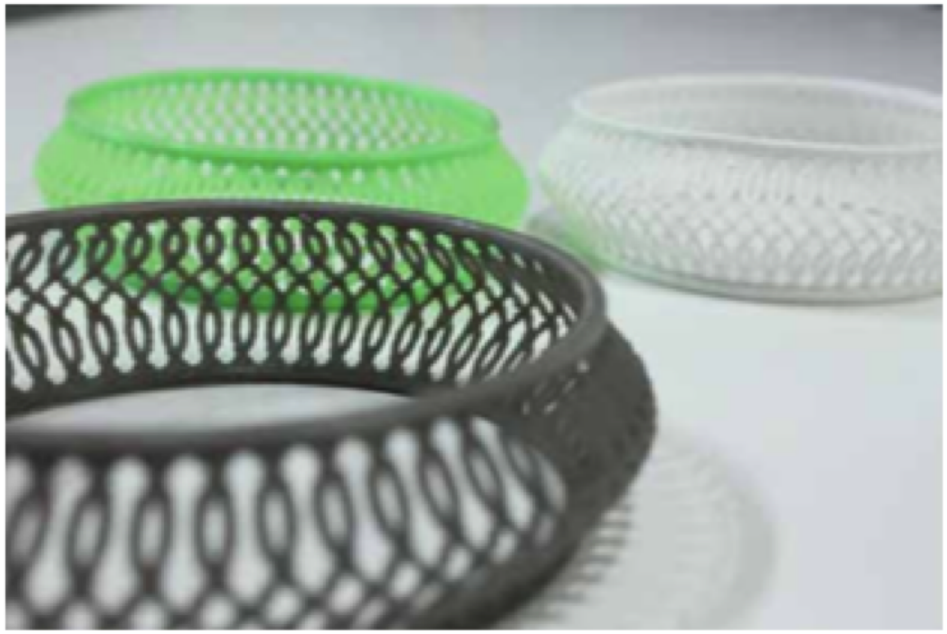
- Make sure that this setting is switched on. Retraction pulls filament back into the nozzle as the print head moves and helps to avoid unsightly stringing.

- It is used to avoid drips and tears of material during the print head movements from one side to the other of the object to be printed.

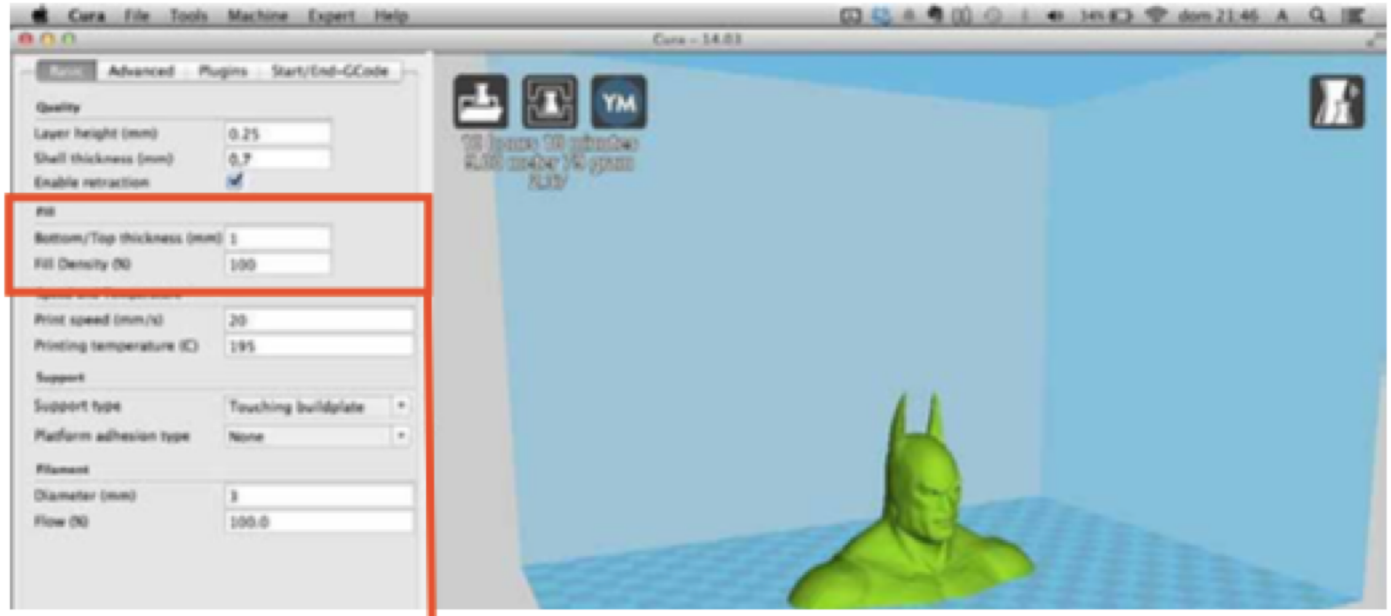
- Always keep this option ticked when our pieces do not have a continuous profile.

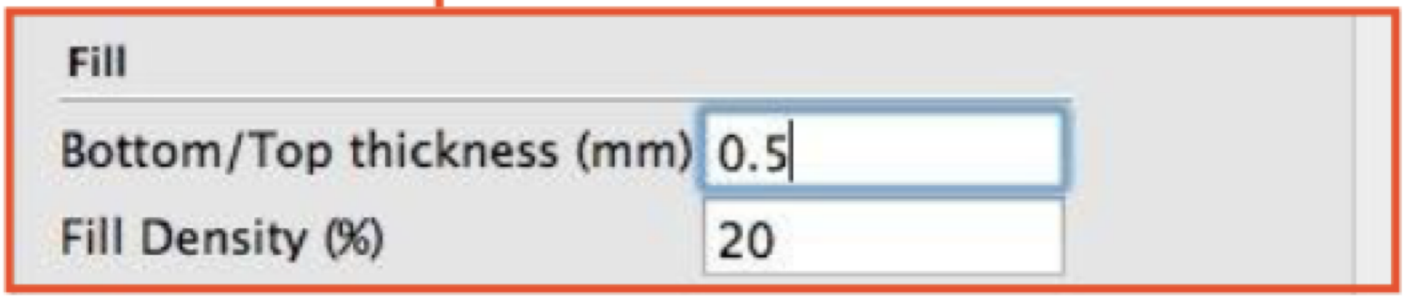
Basic - Quality

ENABLE RETRACTION

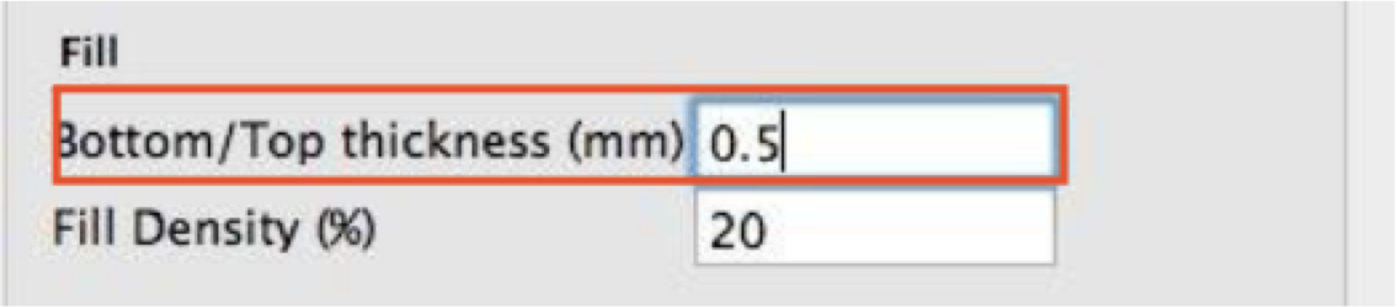


Basic - Fill


Cura File Tools Machine Expert Help
Basic Advanced Plugins Start/End-GCode
Quality
Layer height (mm) 0.25
Shell thickness (mm) 0.7
Enable retraction
Fill
Bottom/Top thickness (mm) 1
Fill Density (%) 100
Print speed (mm/s) 20
Printing temperature (C) 195
Support
Support type Touching buildplate
Platform adhesion type None
Filament
Diameter (mm) 3
Flow (%) 100.0


Fill
Bottom/Top thickness (mm) 0.5
Fill Density (%) 20

Basic - Fill
BOTTOM / TOP THICKNESS

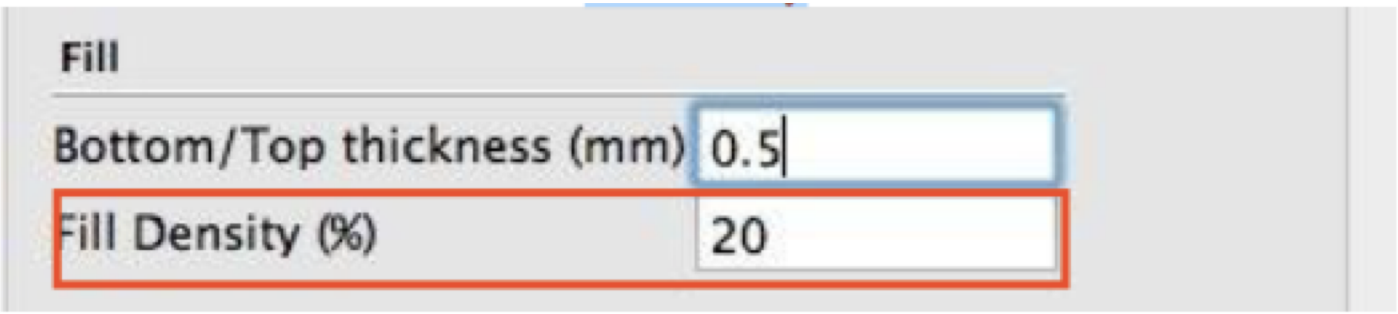


- This value determines the thickness of the initial and final layers of the printed object. Multiple Layer Height values are recommended.

- This parameter is very important as it determines the good adherence of the workpiece to the print plane and the quality of the surface finish of the last part of the workpiece.

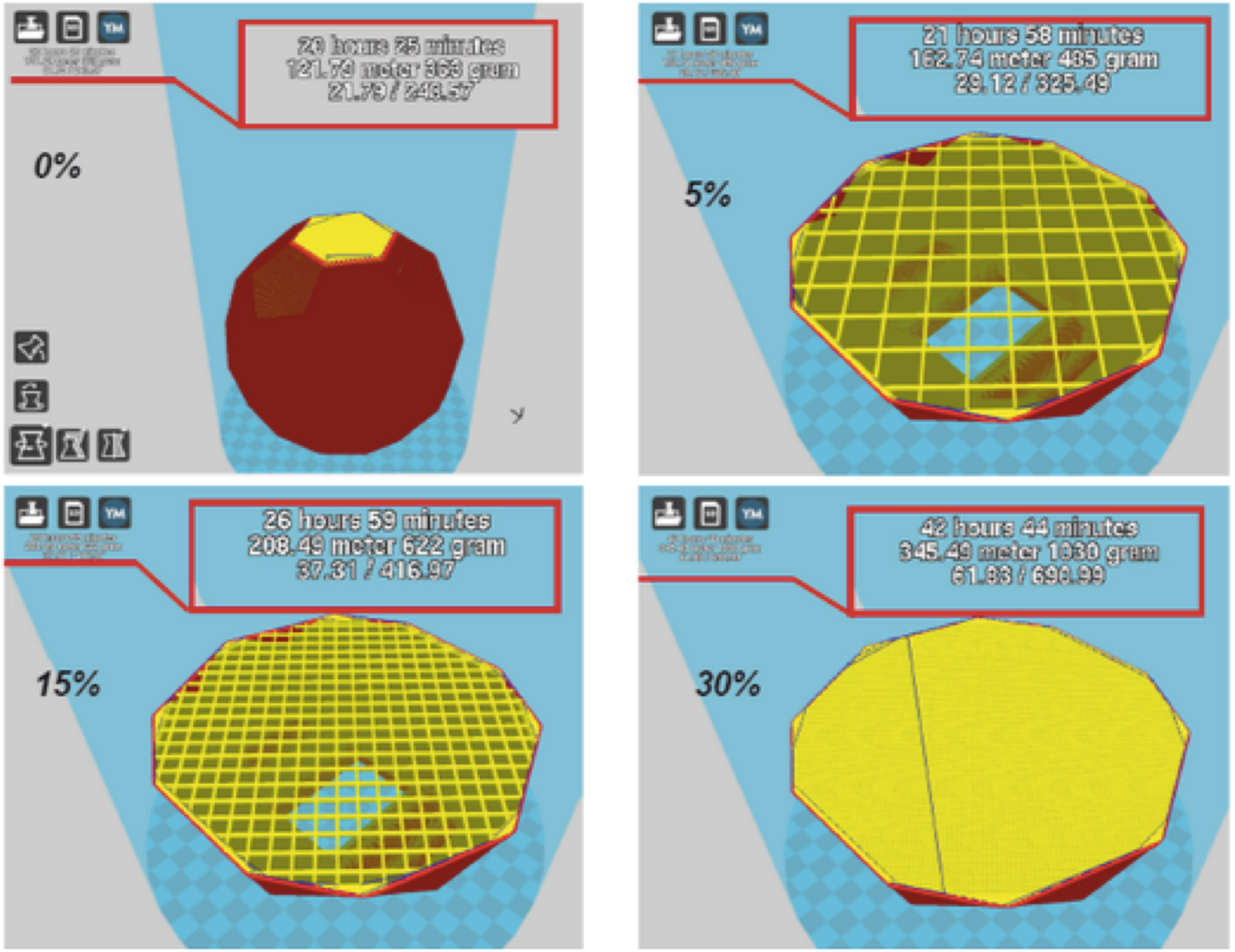
Basic - Fill

FILL DENSITY



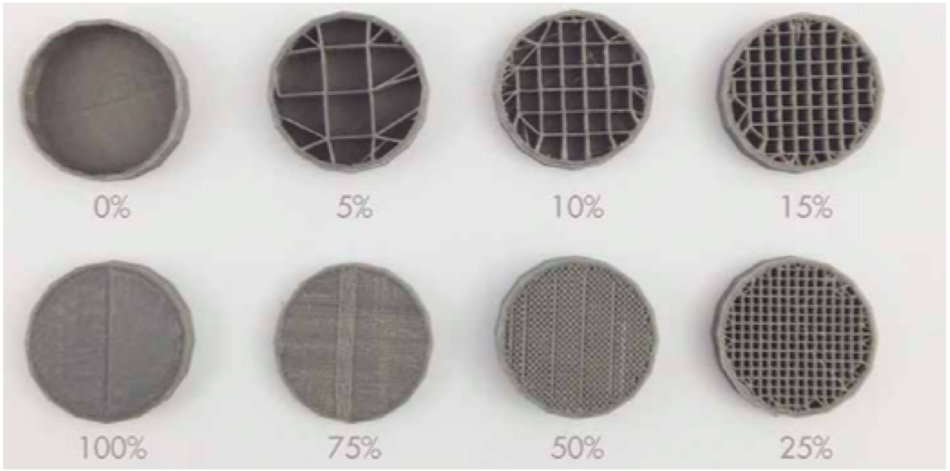
- This is the amount of material that is printed within the model. The higher the density the higher the strength. This is usually set to 20%.
- For example, for mechanical parts we use higher values tha 20%, for purely stylistic pieces can be down to 0%.
- High fill values generate a solid piece but greatly lengthen the printing time and the amount of material used

Basic - Fill


20 hours 25 minutes
121.79 meter 363 gram
21.79 / 248.57
0%
21 hours 58 minutes
162.74 meter 485 gram
29.12 / 325.49
5%
26 hours 59 minutes
208.49 meter 622 gram
37.31 / 416.97
15%
42 hours 44 minutes
345.49 meter 1030 gram
61.83 / 690.99
30%

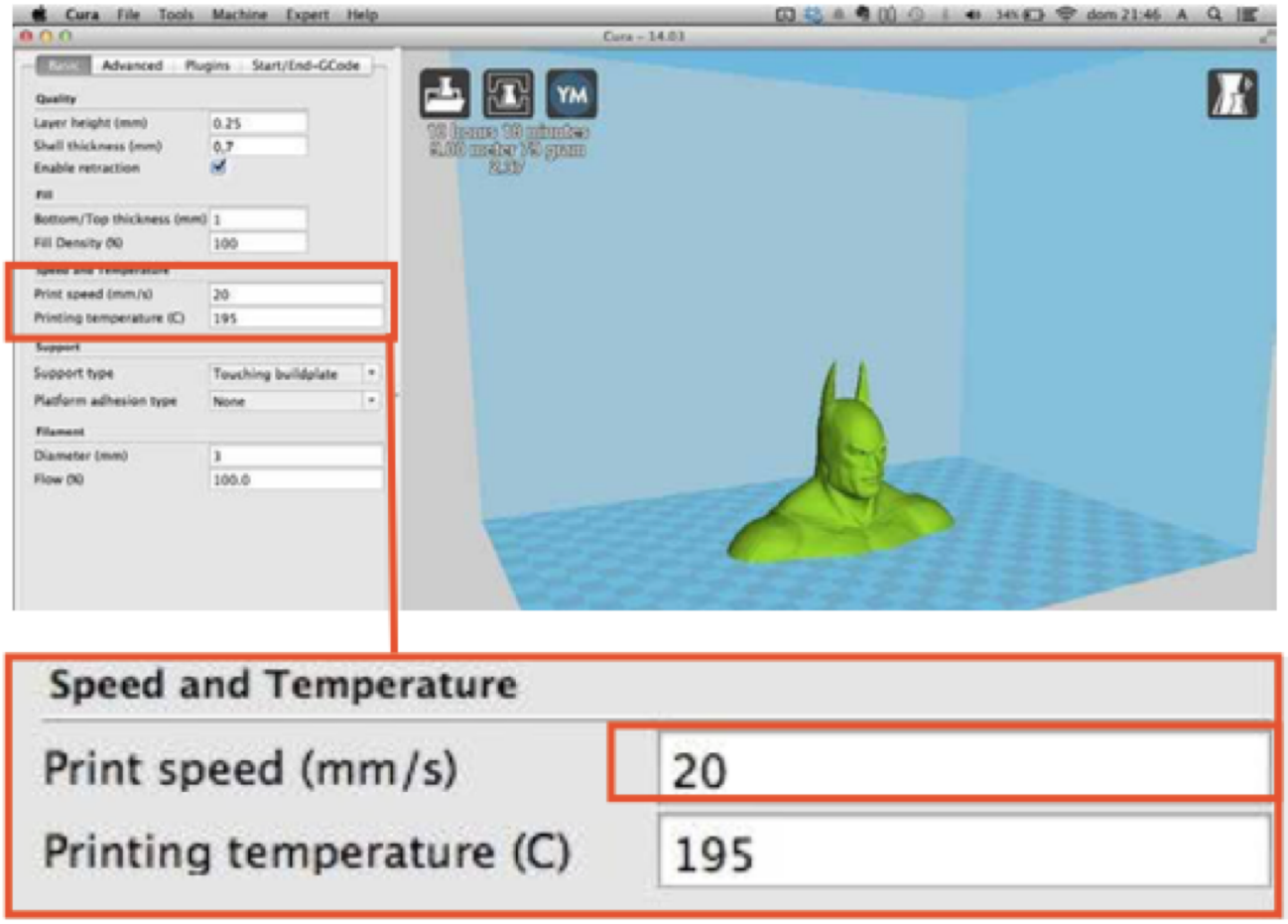
Basic - Fill

FILL DENSITY



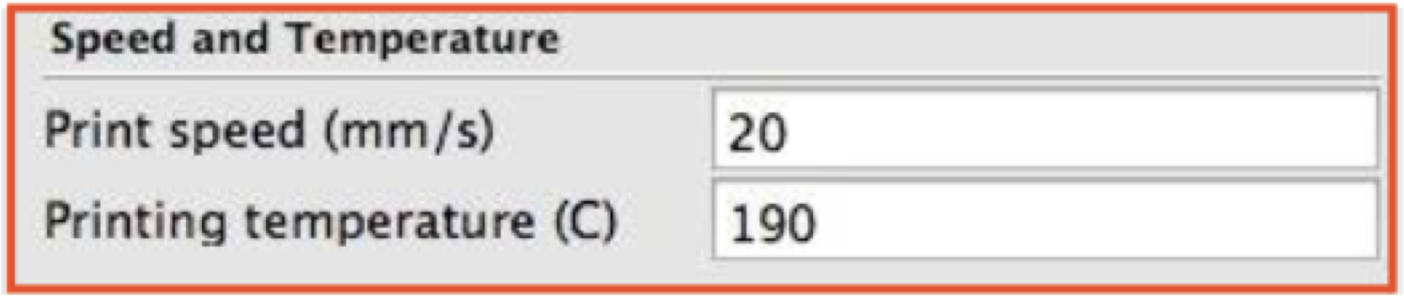
Basic - Speed and Temperature

PRINT SPEED



Speed and Temperature

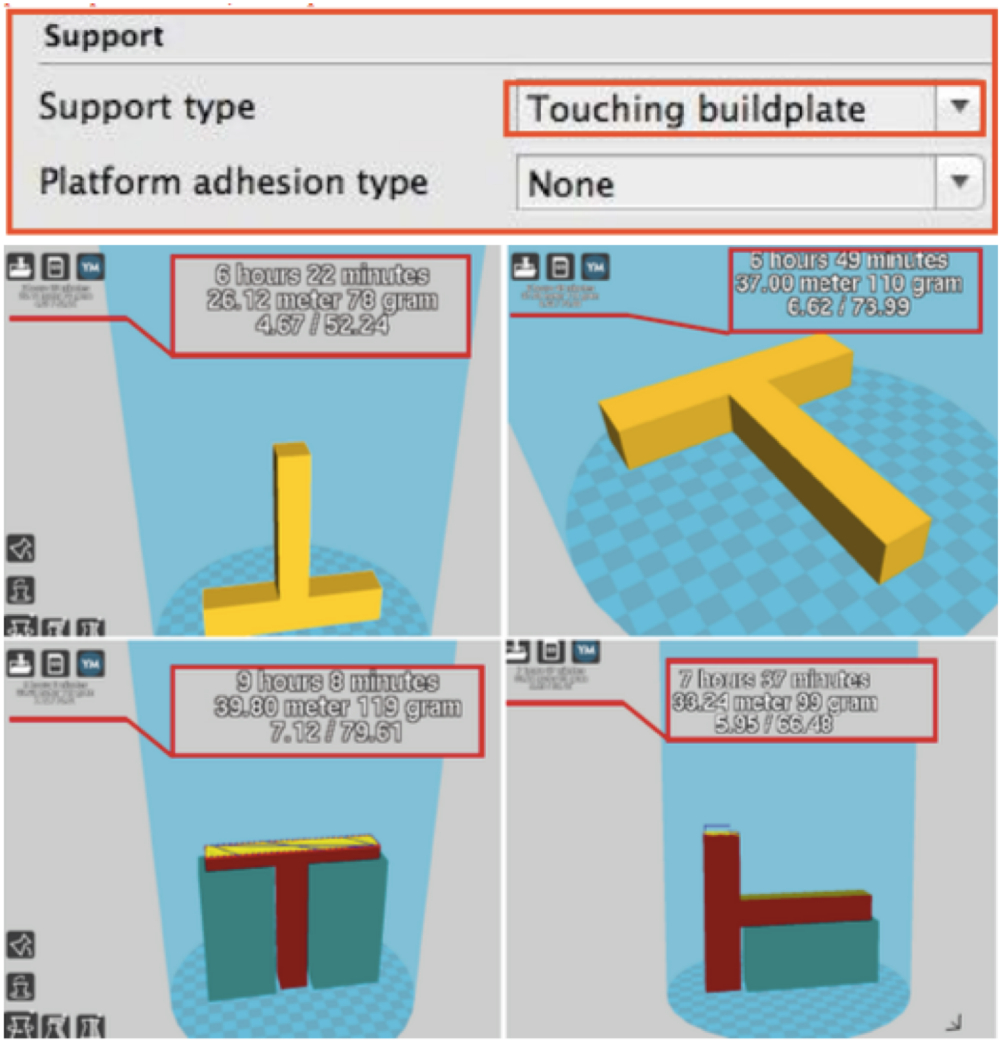
Printing temperature



- The temperature depends mainly on the printing materials.
- The recommended temperatures are: PLA: 190 ° -Nylon: 210 ° these values refer to materials supplied with the printer, materials from other suppliers may require different values that will be supplied with the filament.

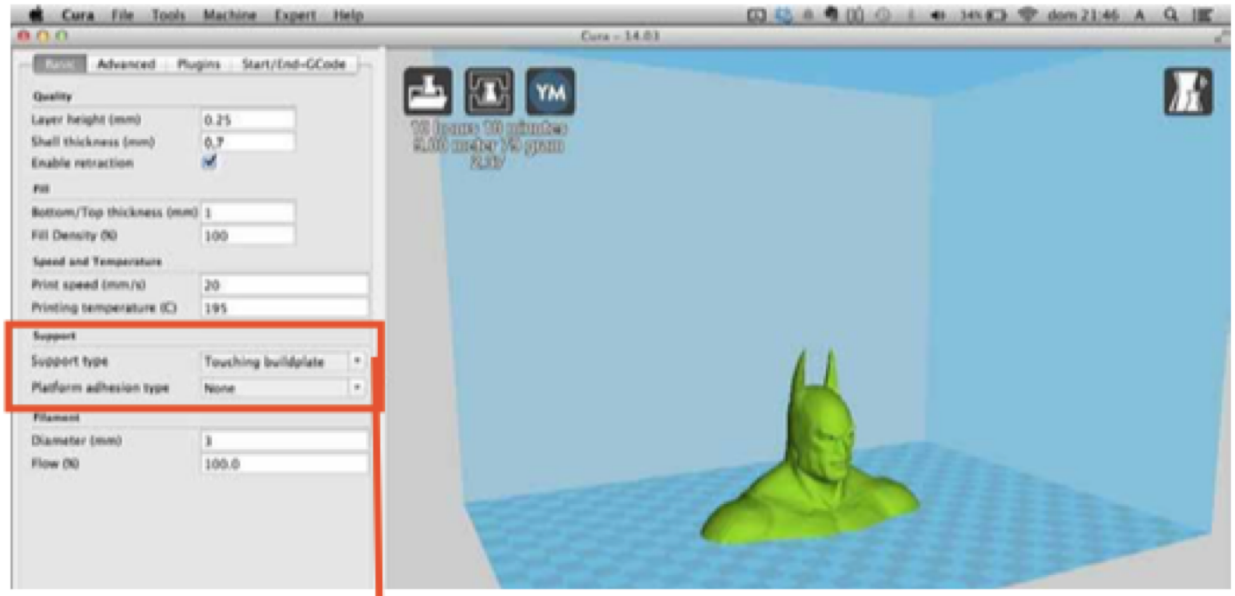
Basic - Support

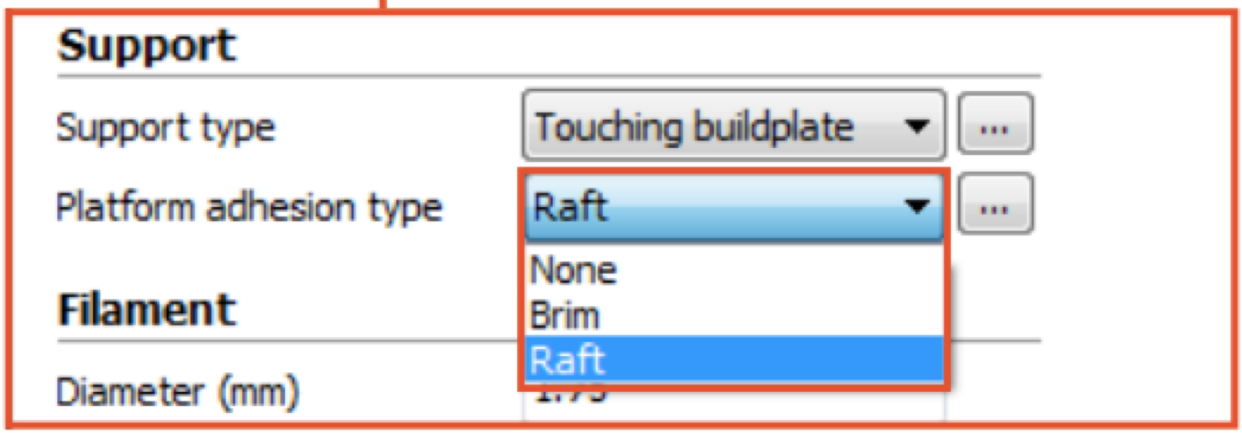
SUPPORT TYPE



Basic - Support

Platform Adhesion Type

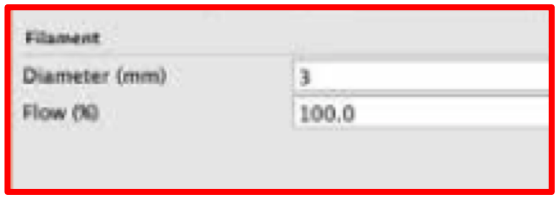




Basic - Support

Platform Adhesion Type

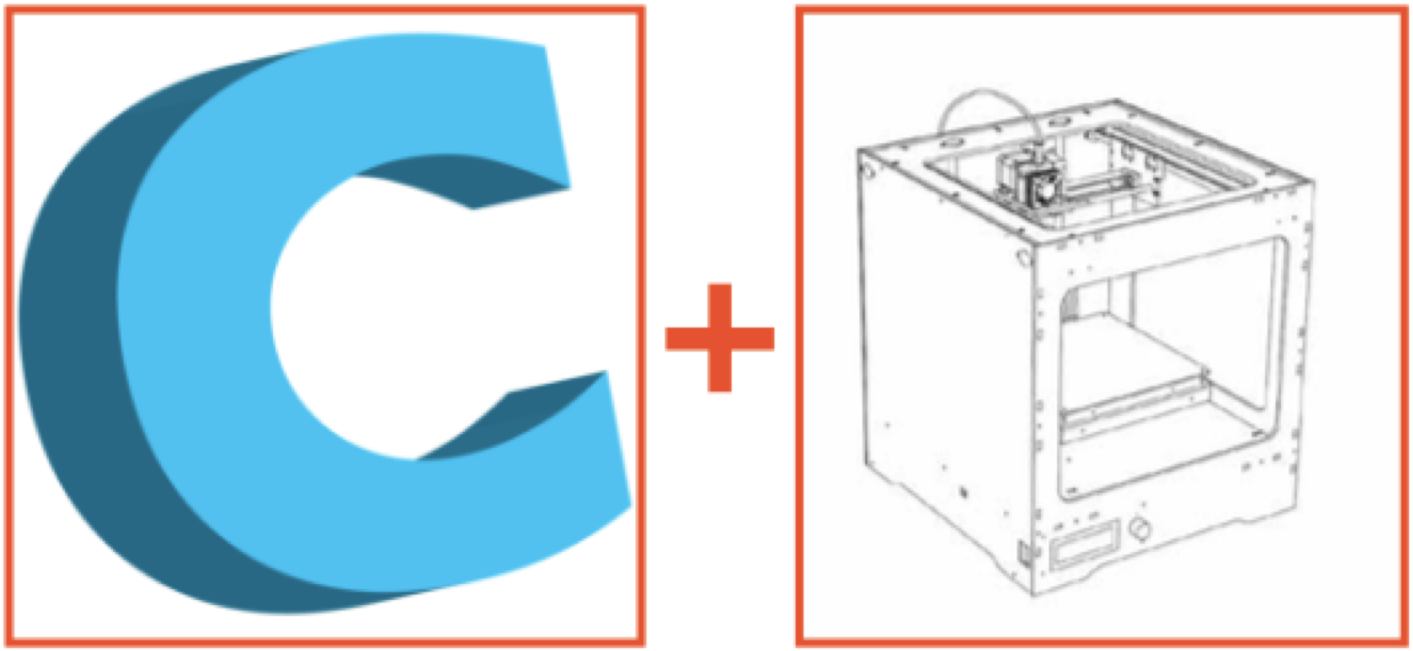
Filament



1,75 mm

3 mm

FR = Nozzle size (mm) x layer height (mm) x print speed (mm/s) = mm^3/s.



To Do…

- Design cubes with a different interconnected porosity
- Get the GCODE to print them with a pyston-aided 3d printer using gel based material