

*.STL files



PART III

Solids

3D printing

Probably the reader is aware of the main features and parameters of 3D printing, but we want to clarify and remember some of them before going into 'drawing 3D printing', chapter IV.

The 3 most important features for basic 3D printing are:

1. Model units: mm
2. Watertight model (closed, solid).
3. Export as *.stl file extension.

In the next chapter ('drawing 3D printing') it will not be necessary for the model to be watertight or export it as *.stl file. But before that, let us remember some important features for 3D printing.

Design for 3D printing requires thoughts on how the machine works and knowledge of the advantages and limitations of the process. The basic behaviour of a 3D printer is to extrude material onto a build plate adding layer upon a layer.

Adding material from zero allows us to create also hollow parts or decide a percentage of internal infill, something that is impossible with other manufacturing methods.

Advantages

Almost all geometries are possible

Create hollow parts

Low cost machine and start up

Wasteless process

Limits

Anisotropic properties

Lower mechanical properties compared with other methods

Design limitations due to force of gravity

Slicer software.

In 3D printing the CAM software used is called 'slicer' because it literally slices the solid and creates a path to move the printhead. The most common slicers are: Slic3r, Cura and Simplify3D. These software are really useful and powerful but sometimes they can limit our imagination.

Each slicer has many different parameters that can be edited, but they all share these basic parameters:

Wall thickness.

Modelling in meshes or in surfaces allows us to make objects with walls without thickness, but in case of 3D printing, every wall must have a thickness to explain to the software how much material is needed.



1 layer 

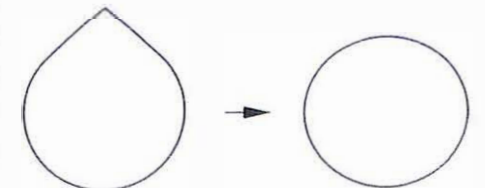
2 layers 

4 layers 

Overhangs.

Overhangs are a big nightmare in 3D printing, especially in FDM or LDM technology. Overhangs are created from printing at an angle - internal or external - more than 45°. Over this range, the material does not have the support of the layer behind and may fall down due to the force of gravity. If the overhang begins and ends in a pillar, the slicing software recognizes the 2 pillars and the software will program a movement called a 'bridge' that consists in a small overflow of material and an increase in printing speed to stretch the material extruded and cover the distance between the pillars. To know the maximum distance achievable by your 3d printer a bridge test could be made.

In order to avoid overhangs, there is an old trick that helps the printer to create almost perfect horizontal circles by designing the hole like a 'tear drop', which prevents the top layer from falling down - take a look on the RepRap logo.



Support structure

In a standard plastic FDM 3D printer, it is possible to add support materials that represents some form of structure that helps the final part to sustain itself during the printing process. They can be easily removed. It is a helpful tool to support parts where the angles of extrusion are more than 45°. This structure is made of columns that support from the build plate to the required part.

Usually the support material leaves some marks on the final object surface depending on the settings and the quality of the printer. Some 3D printers have a double extruder. This way it is possible to print the support materials with a soluble material that can be dissolved in water or other chemical products after printing, leaving the final object perfect. The use of soluble materials is more recommended when the object has internal channels or thin parts that would be broken by removing the supports by hand.

When 3D printing clay, support materials are a possibility, but they leave visible marks on the final part and are not recommended; also the support materials would have to be kept in the kiln to avoid the collapse of the object during firing.

Tolerances.

Depending on the type and model of 3D printer, the tolerance changes according to the nozzle diameter and the material used.

E.g. when printing with a clay 3d printer and using a 2mm nozzle, the minimum distance to print walls that don't touch each other is 1mm, whereas printing in plastic with a 0.4 nozzle the tolerance normally is 0.2mm.

In some slicing software it is possible to define whether the path should follow the internal or external part, but the common method is following the centre line.

Details.

Avoid details that are thinner than the nozzle diameter or geometries where the slicing software cannot generate paths for.

Origin / orientation.

The object should be designed from the position $X=0$ $Y=0$ $Z=0$ because many slicing software identify the same origin position of the CAD Design. Choose the correct orientation of the model by checking the overhangs, choose a plane surface as a base to guarantee a good build plate adhesion and, when printing in clay, check if the bottom part can bear its own weight. Also check out the right layer deposition direction according to the stresses the part will be subjected to.

Also, ensure that there is a proper surface of the model that touches the base to avoid the detachment of the model during the print.

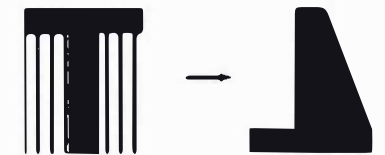
THEY Rule

The 'THEY' rule is a set of basic rules that explains how to orient the parts to print according to their geometry.



The rule says we can only rotate the pieces in the XZ plane.

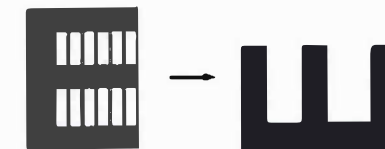
T needs supports. The problem is solved if we rotate it 180°.



H needs supports. It has a bridge. Some slicers have a 'bridge' option that increases the printing speed in that area avoiding the supports.



E needs intermediate supports. The problem is solved if we rotate it 90°.



Y is the maximum angled shape to 3D print without supports. It could be 3D printed as it is, or rotated 180°, improving the contact with the build plate.



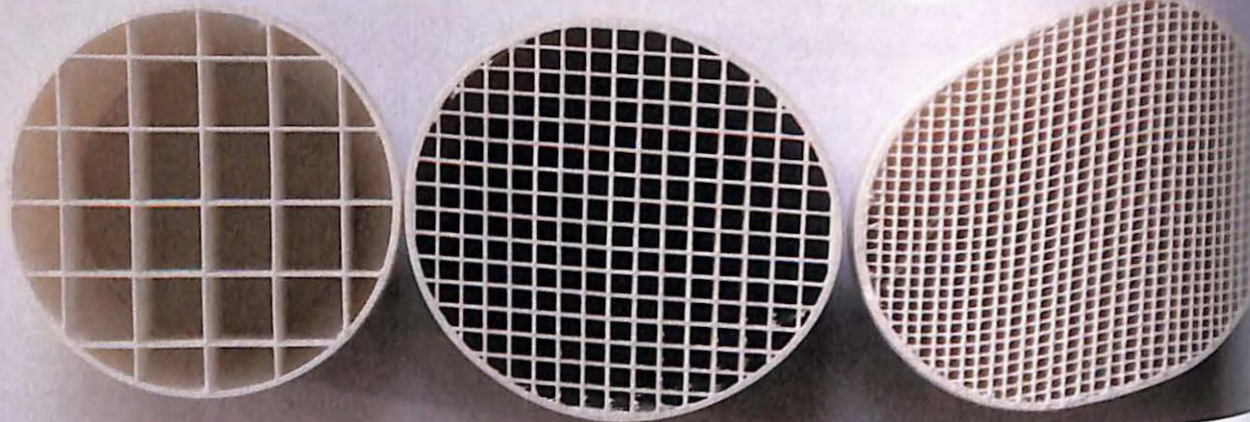
Infill

The infill is one of the big advantages using additive manufacturing, as we can choose the percentage of material that we want to fill an object with. The basic infill pattern is a square grid where we can set the density from 0% to 100%.

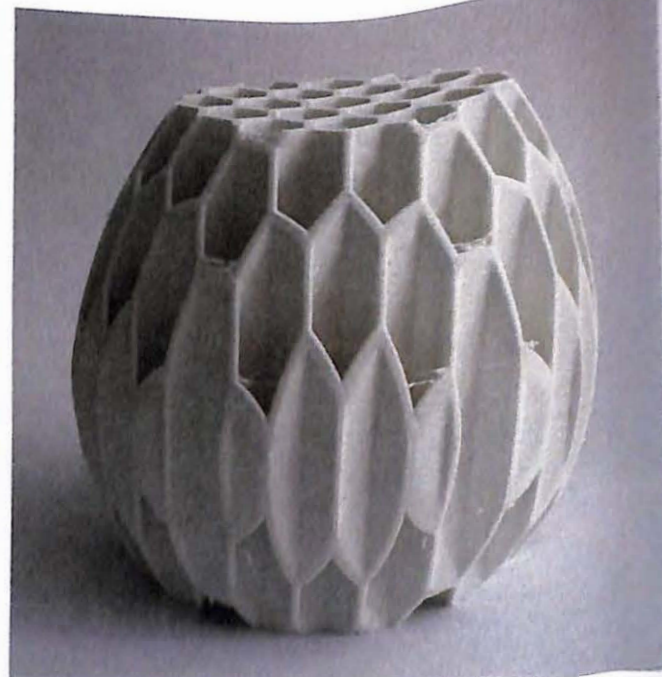
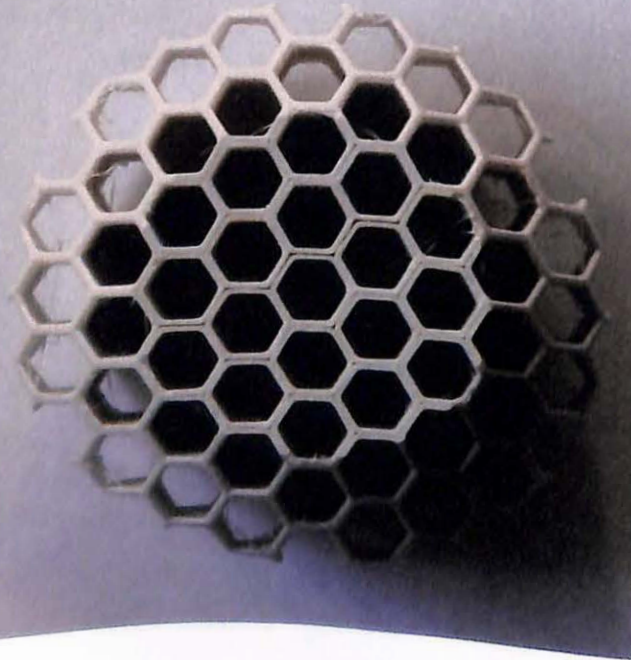
10%

30%

50%



With some 'slicer software' we can choose other infill patterns, such as honeycomb, triangular, wobble, gyroids and more. Infill can be also used as an aesthetic feature.



Layer height.

With the layer height option we can change the quality of the print, as a thinner layer means greater quality and finer details but this also means a longer printing time.

Depending on the material that we are using, we need to consider the deformation of the material, a thin layer thickness means that the material may become deformed. However, there are some plastic materials such as PLA or PETG that do not suffer as much because the extruder's fan can quickly cool the material as it sets. In case of clay, choosing a layer height that is too thin results in imperfections on the external finish (as in the picture).



Speed

The slicing software can determine 2 kinds of speed, the printing speed and the travel speed:

Printing speed is the speed during the extrusion process, so when setting the value, you have to consider the extruder limits. It is standard that travel speed is higher than printing speed.

Travel speed is the speed during the travel movements without extrusion, such as moving from point a to point b.

Speed is expressed in mm per second or mm per minute. Depending on the 3d printer and the material it is possible to set different speeds. For a "safe print" with most 3d printers, the common range speeds are:

PLA -> 1800-4200 mm/m

CLAY -> 1200-3000 mm/m

Temperature

In the common slicing setting, there are 2 temperature parameters when we work with plastic:

The extruder temperature: it is the temperature required to melt the filament, depending on the material used. For example, for PLA the temperature range is from 190° to 220°.

And the build plate temperature with a range from 50° to 80°. It helps to the adhesion of the part to the build plate and avoids extra warping in big pieces. Adhesive material can be used to stick the model down to the plate as hair spray, blue tape, etc.

It is not necessary to heat any component when 3D printing clay.

Watertight

The model should be 'watertight' or 'solid' avoiding double surfaces and collisions. It has to be clear what is the interior and what is the exterior.

In Rhinoceros® there is an interesting tool to check out: _ShowEdges.



This tool can distinguish from 'All', 'Named' and 'Non-manifold' edges.

The 'naked edges' are the open ones. Where the 3D object is not watertight.

The 'non-manifold edges' are edges that belong to three or more faces (Breps or meshes). Those are usually a big issue for 3D printing as they can create other 'interiors' and several problems. By default Rhinoceros® does not allow to create non-manifold edges. They mainly happen when importing files from other software.

SAVE the file

In Rhinoceros®, export to *.stl and save as Binary as the file describes the mesh in such an efficient way as ASCII format and it is a smaller file.

If the model was done in Grasshopper®, then 'Bake' it to Rhinoceros and export as *.stl file extension.