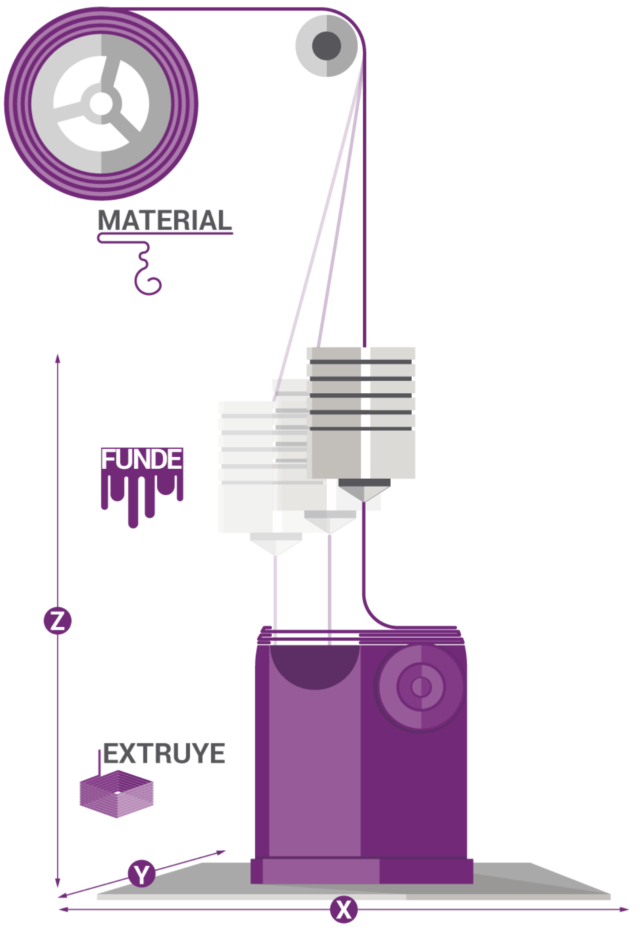
Design optimization for 3D printing

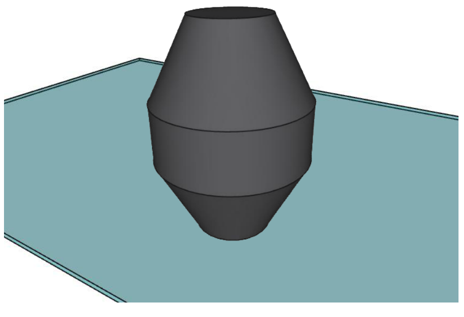
For our 3D part to be printable, it is important to optimize our design. In this post we are going to explain why and what are the previous considerations that we must take into account to optimize our designs before proceeding to their printing with fused deposition modeling (FDM) printers.

Let us remember that the fused deposition printing method is the most widely used additive 3D printing technique, whose operation is based on a coil of plastic material that is melted and expelled through a nozzle into fine threads that are cooled and solidified on a base. flat, extruding the piece layer by layer according to the required geometry.

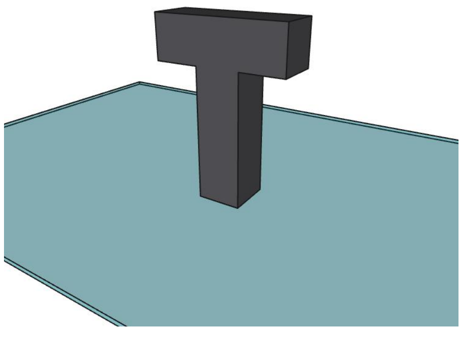


Why is it necessary to optimize the design?

Since the piece is forming layer by layer on a flat base, we can find situations in which our printer cannot find support to place the molten material, for example in the following cases:



In this case, at the ends of the lower part of the piece, the nozzle would not find a support where to deposit the material.



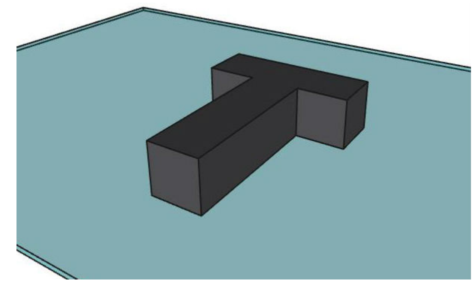
If we go to a more extreme case, for example a letter T, the problem will be even greater when trying to print the horizontal pillar of the letter, since in that part the nozzle will not find support on which to deposit the material, it would be deposited literally at the air.

If we were to print the part, we would find this result:

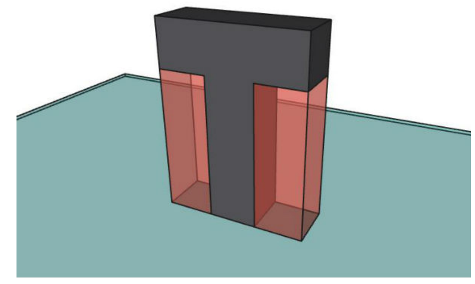
How can “open” printing parts be avoided?

There are three ways to avoid this problem:

- The **first** is to rotate the part in the laminating software until the position is found in which the areas of the part exposed to lack of support are minimized to the maximum. In the example of the T, we solve the problem by resting the part completely on the build platform.



- The **second** is to use support material for printing. This support material is the same as the rest of the piece, it is extruded together with the rest of the layers, but with a lower density so that it can be easily removed. Support is generally generated with laminating software, but we can include it as part of the design.



- The **third** way, if none of the previous ones gives us the expected result, would be to redesign the piece until we achieve an optimal way of printing it.

The combination of these techniques will allow us to improve the final quality of the piece, save material (more supports imply greater cost of it) and reduce printing times (more supports also mean more printing time).

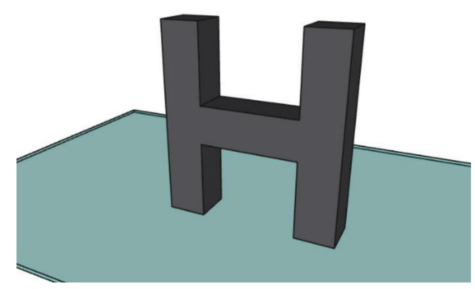
Other considerations when printing

Other cases that we must take into account when designing parts are explained below:

- Bridges

  Parts in which the extruder expels material in an area where there is no support, such as a letter H. Going from a point where it finds support, to a part without it, and returning to an area with support would give As a result, the print will not be a completely horizontal layer, but a layer in which some strands of material will remain as if we were holding a cable from its ends and it was slightly curved.

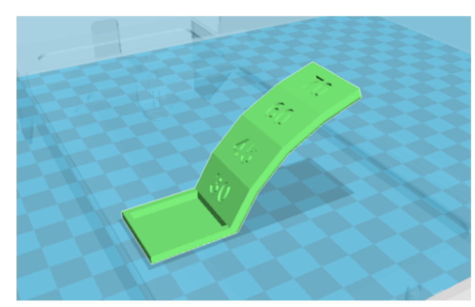
  It can be minimized by increasing the speed of the extruder or increasing the air flow of the fan so that it cools faster and the curve of the hanging part is less.



- Outgoing

  They are cantilevers or inclinations, which print worse the greater the angle with respect to the vertical, especially from 45º.

  As solutions, a lower layer height can be configured in the laminate of the piece, thus obtaining more support between the lower and upper layers.



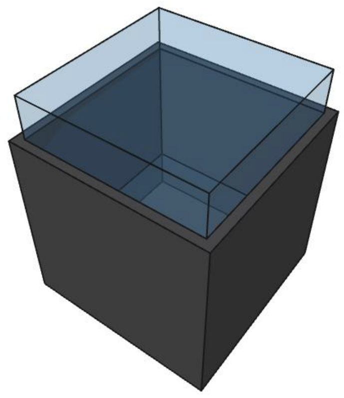
Limitations to consider

Once we are clear about the operation of the FDM printer and we have seen how to optimize our designs so that they are printable, we will have to go one step further and perform a critical analysis of our part to bring it to reality in 3D.

This analysis has to do, more than with the technology used for the FDM, with the precision and state of maintenance of our printer model.

Hole tolerances

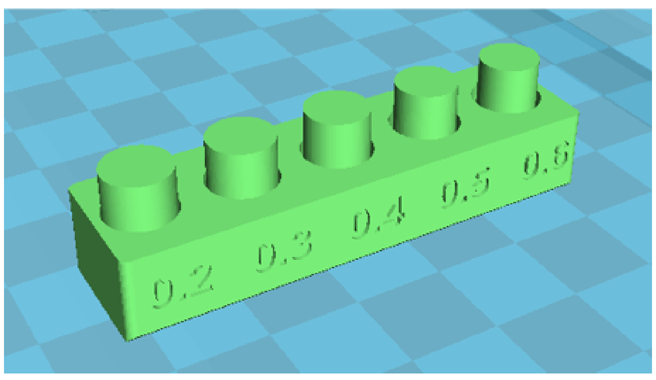
When we design in 3D, we can add shapes that serve to introduce parts of some parts into others.



Let's imagine we have a cubic box and we want to create a cube that fits inside it. If the hole in the box has a value of 5 cm long, 5 cm wide and 5 cm deep, we can create a cube with a 5 cm edge that would theoretically fit perfectly inside our hole, but this does not it is so.

Remember that FDM printing is based on extrusion of molten material, layer by layer. For this reason, the dimensions with which an object is printed are not exact to those of the original design, but it will depend on the diameter of the extrusion nozzle, the distance between layers, the time it takes for the material to cool, etc. .

A prior knowledge of our printer is needed to know what tolerance is adequate, that is, what size difference should the gap and part to be inserted. In the case of the cube, it would be enough to reduce its size by a certain value and check once printed if it fits in our box.



Now let's imagine that we want to design circular elements that can be inserted into another part, for example a cylinder. A good technique to test the correct cylinder radius is to design and print a part like the one in the image.

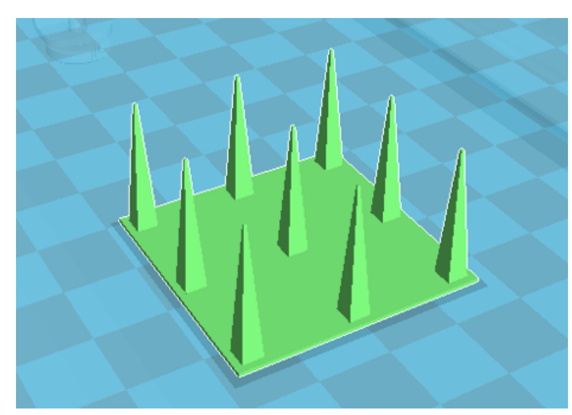
Sometimes we may need a greater friction between the pieces (for example if we want to place holes to pass screws), and other times we will need a greater clearance. The important thing is to make the necessary adjustments until we achieve what we need by modifying our design.

For models like the Witbox 2 or the Hephestos 2, it is not advisable to lower those differences between the pieces that we are going to use to fit one inside the other below 0.25 mm.

Small details

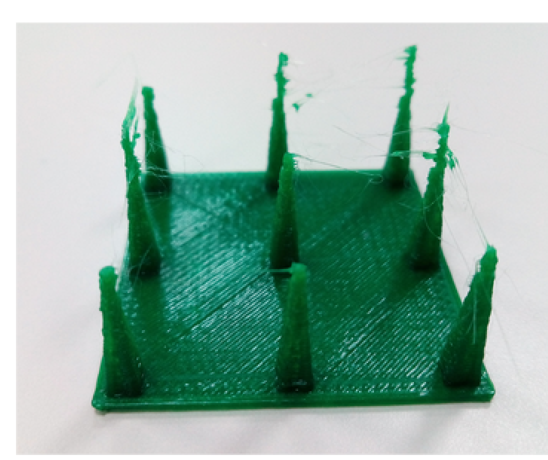
The second point that we have to take into account in our designs when printing is the inclusion of very small details. As with hole tolerances, hole size and inclusion are highly dependent on the accuracy and fit of our 3D printer.

The following design consists of a base of approximately 4 cm on which nine small pyramids with straight faces are supported. During printing, we will encounter different problems, especially in the upper part of the pyramids, due to the fact that the small size of the piece and the rapid passage from one layer to another can cause the extruder to deposit material on a layer that has not yet solidified.



Another problem we can find is due to the material retraction system that the printer is equipped with. If the interval between the point where you eject material and the next is very small, it will leave fine plastic threads between those points.

In the attached image you can see the result when printing the piece.



The solution to this is complicated. First of all, we will have to have our printer well calibrated and adjusted, so good maintenance is of vital importance. Second, we can modify with the laminating software a slower print speed and expand the ventilation flow so that it cools faster. Finally, we advise using high-quality material.

Tips

All 3D design is complex and the combination of all the exposed cases can be given in the same piece. This guide serves as a starting point for designing properly, as well as getting the best possible print, but only experience and testing with your own printer will lead you to the desired result.

Go into some repository of 3D parts and look at some different designs. Perform a visual analysis of the pieces and look for possible problems when printing it (protrusions, small details, use of printing media) ...