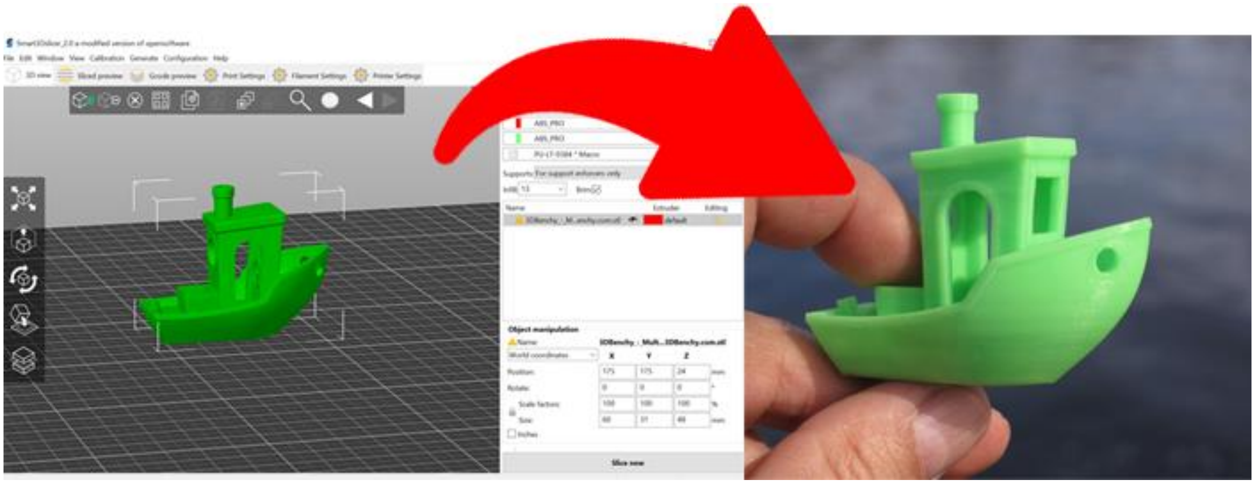
What is 3D printing?

3D printing, or additive manufacturing, is a process that has advanced rapidly during the last decade and turns, using different technologies, 3D models designed on a computer into physical objects.



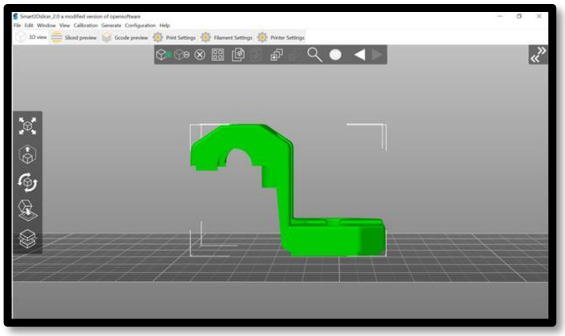
Fused Deposition Modeling (FDM)

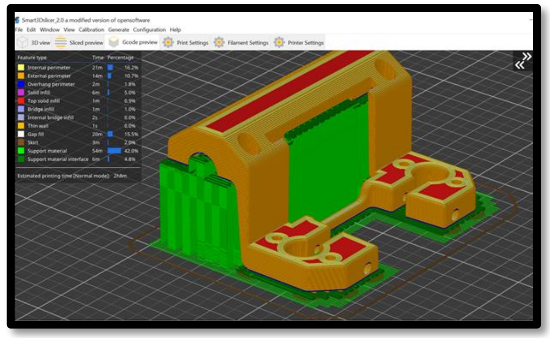
FDM, also known as Fused Filament Fabrication (FFF), is a technology where a 3D printer melts thermoplastic material (or composites) to build a model, by depositing thin lines of material layer after layer.

The first step of this process requires to create an object model in a **CAD software**. The object is then processed through what is called a **Slicer.** A Slicer is a second software that processes the geometries and generates very thin "slices" of the model in the Z-axis. Then each “slice” is processed to generate all lines of material the 3D printer will need to deposit to build that layer.

This set of information is sent to the 3D printer, which is building by deposition each slice, one on top of the other, until the entire object is built.

It is important to understand that a 3D printer requires a print surface to deposit filament (either directly on the build plate or on another layer of filament or support material), as material cannot be deposited mid-air.





3D printing technology can be successfully utilized for producing both prototypes and final production parts, thanks to its versatility and strategical applications when combined with the mastering of CAD design tools.

Advantage

3D printing accelerates validation and prototyping processes. Only a few years ago these would required labourious manual layout work. Today this new technology can reduce weeks of work to few hours. In addition, 3D printing allows the generation of parts with geometries hard or impossible to achieve with other standard manufacturing processes (such as injection molding, machining, forming, and joining).

Scope of this manual

This manual will cover:

- the basics of computer-aided design (CAD),
- design for 3D printing optimiztion, and
- general advice and recommendations.

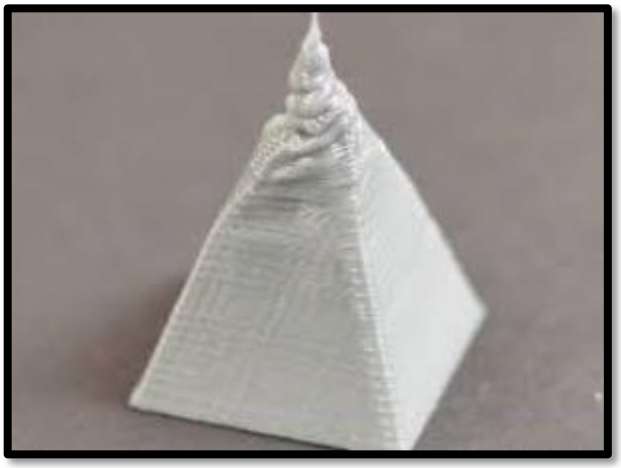
The basics

To achieve best results, it is important to apply specific 3D modeling techniques to reduce certain troublesome occurencies during 3D printing, such as:

▪ Warping

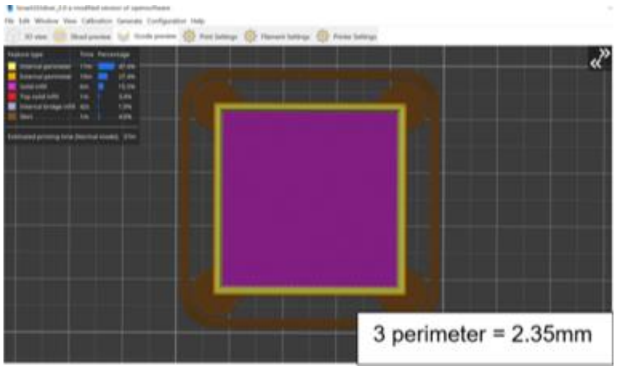
Thermoplastic filaments will expand slightly as it's being melted by a 3D printer (further expansion may be caused by chemical reactions from ambient humidity/water absorption). In addition, depending on the material and the temperature to which it will be subjected to, crystalline chemical structures will form and decompose, thus increasing/decreasing the internal tensions within the material, and consequently in the final piece. Some polymers contract more than others. All these factors will determine the tendency of this material to "twist" while it is being printed and cools down.

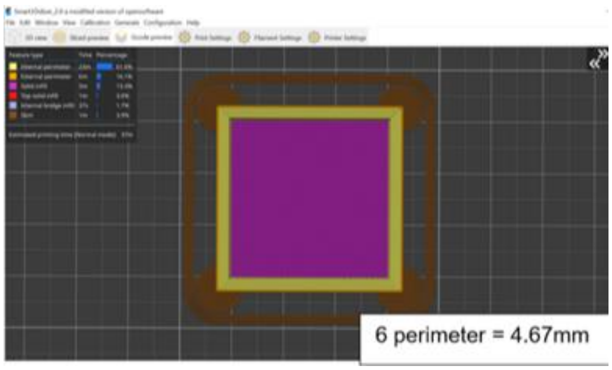
This "twisting" is caused by several factors:



- **How quickly the material cools.**
  If the lower layers of the object are cooled slower than the upper layers, the contraction (and the stress along the lateral surfaces) in the upper layers will be more severe in the same time frame. The lower layers (which are still soft) will yield and twist by the force from the contraction of the upper layers, producing a "crooked" piece. This is why it is of utmost importance to **allow the object to cool evenly and slowly** as much as possible. Smart3D printers have a controlled environment to complete this process and reduce the possibility of twisting the pieces. Materials that are more prone to this phenomenon include ABS, HIPS, and Nylons. It is also vital to have a uniformly heated chamber to help generate and keep the object at a precise temperature throughout the cooling process.

- **Number of perimeters.**
  Perimeters are the exterior lines of the piece (see image below: perimeters are highlighed in yellow). Normally, at least two (2) are being generated: One (1) external perimeter (the outer surface of the piece) and one (1) internal perimeter. But there may be more. As a general rule, the greater the number of perimeters, the more difficult it will be to contain these contraction forces, and the warping tendency will increase.

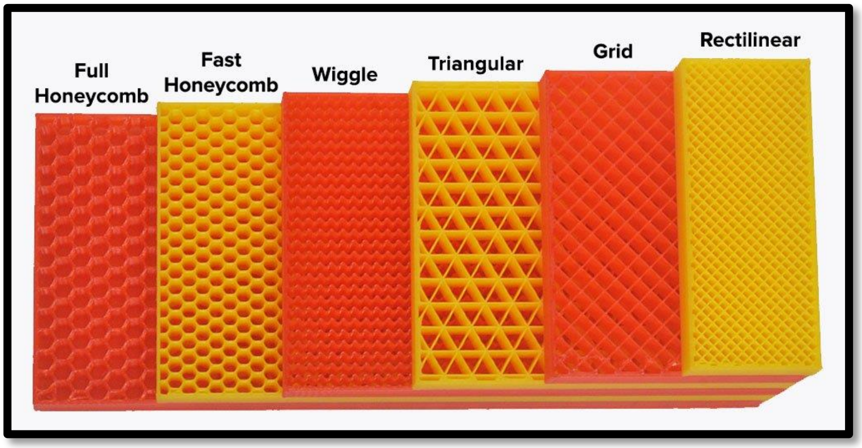




- **Amount and type of filling.**
  The filling of an object (measured in %) defines how much of the internal space will be reinforced with material. Filling can be designed with several printing patterns (see examples in the image below. There are up to two dozens different patterns), depending on the slicing software used. Each

plastic line deposited will contract when cooled. Therefore, the more filling lines deposited, the more the internal contraction forces. Since contraction forces act in the direction in which lines have been deposited, the pattern chosen plays an important role. Some patterns, such as "sparse infill" or "grid infill", have become the state of the art and gained certain consensus that they cause little warping as long as filling is 20% or less. Other very peculiar designs, like the "rounded infill" (property of KISSlicer), were specifically designed to reduce warping.

If a print work suffers from warping, one of the first action we must take is to **reduce infill and perimeters** to the minimum necessary. This includes the use of more advanced techniques such



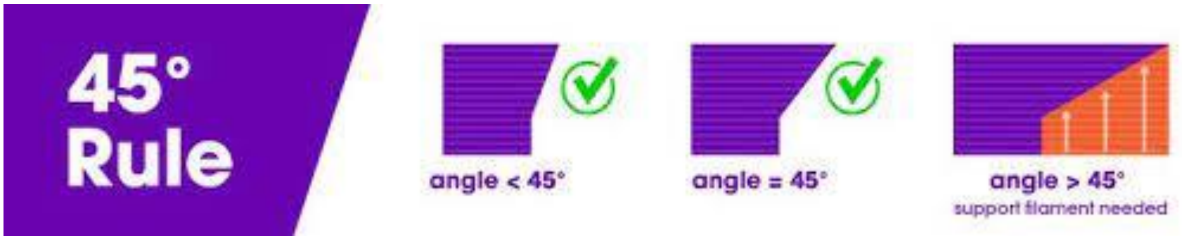
- **Heated bed:**
  Contrary to popular belief, setting the heated bed to very high temperatures will not reduce the possibility of warping. The main objective of a heated bed is to improve bed adhesion and slow down the cooling of the entire object as much as possible. We want to set the bed to a **very high temperature during the first layer** (to improve bed adhesion) **and then reduce it** gradually in subsequent layers, to allow the first layers to cool down and solidify. A common mistake by beginners is to maintain a very high bed temperature throughout the print, which will keep the first layers in a soft state and increase the possibility of warping, especially with poor bed adhesion.

  e.g., If you keep the bed temperature a 95°C while printing ABS you will run into warping or "elephant foot" issues as ABS will remain in a soft state at temperatures above 80°C.

- **Air currents:**
  The heat generated by the heated bed will raise the chamber temperature. Likewise, the heat used to melt the material in the printhead persists untill gradually lost. If layer cooling is sudden/rapid, beacuse of gust of fresh air, contraction will be uneven and severe warping will occur. An e**venly heated chamber** prevents abrupt changes in ambient temperature and uncontrolled air currents.
- **Geometry:**
  The geometry of the object can also affect the warping tendency of the object. **Corners** with sharp edges are a risk, and if possible, should be designed rounded to mitigate possible warping.

Supports

As mentioned, material must be layed on top of another layer to create a tridimensional structure. When part of an object needs to be printed with no layer underneath, supports need to be implemented. Said structures are added to provide alayer of support to parts of the object that would otherwise be built in the air. As a general rule, any printer that uses FDM/FFF technology is capable of 3D printing geometries that have a **maximum inclination of 45 degrees** (from the 90 degrees vertical angle of reference).



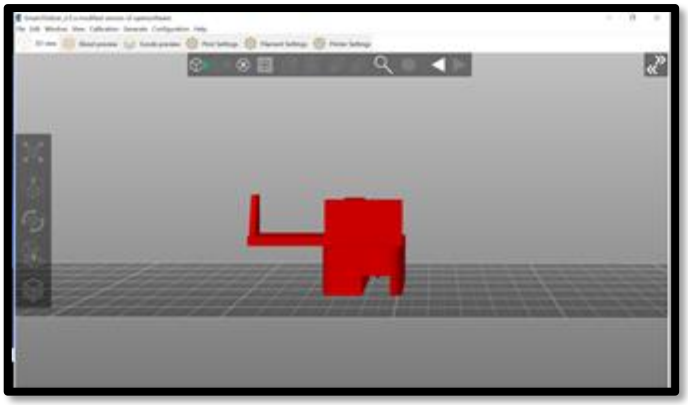
More precisely, the limit of inclination is imposed by the % of the width of the outer perimeter to be printed without support below. If this % is greater than 50% (this is the accepted standard value), that perimeter will be likely printed "in the air" and will certainly affect the printing quality.

For example, if I have a 10cm piece, the percentage width of the outer perimeter should not be greater than 5cm (50%), that way the piece will not need support.

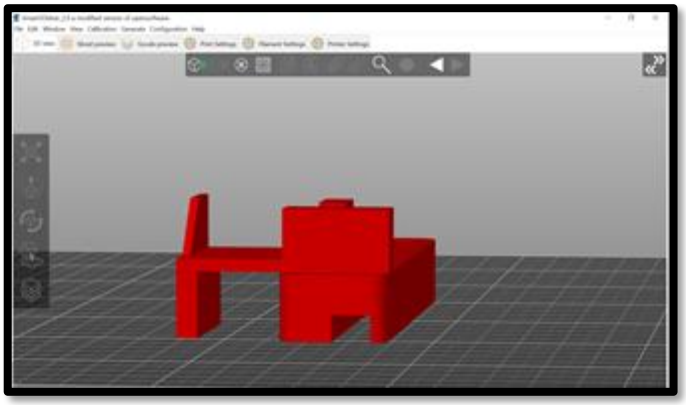
Slicing softwares can detect these angles (also known as overhangs) and generate in these areas supports. The disadvantage of supports is that they may be difficult to remove, they may leave marks on the surface of the piece, and in some cases, their removal is impossible because they are located in areas where the user does not have access. The other disadvantage is time: the generation of these structures usually adds a good portion to the total printing time. Therefore, it is advisable to **avoid them and/or minimize support** as necessary.

Techniques and general rules can be applied when creating objects in our design software:

a. **Manual supports:** The 3D printer can build straight (not curved) "bridges" connecting points A and B rather neatly without any material underneath it and without requiring the generation of support. This allows us to perform a little trick. As can be seen in image A, we see an object with an area that requires supports and we see the same generated by our slicer along with the total time of printing.



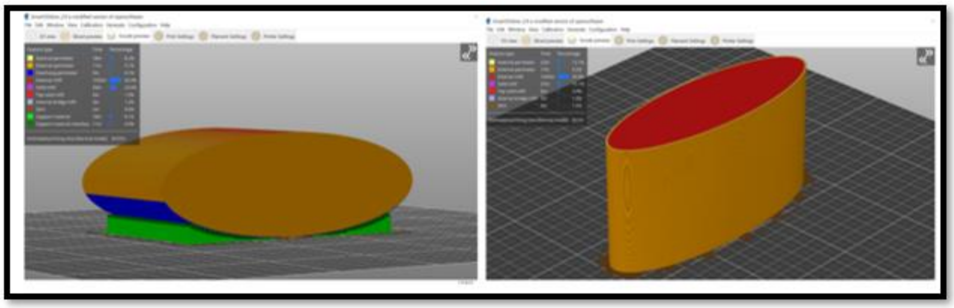
In image B, the same piece, but with an artificial addition that was manually placed in the design This consists of a small construction separated by an articulation in the Z to 0.1mm (so that it is easily removable).



By manually inserting this construction, we force the 3D printer to build "bridges", which reduces material consumption caused by automatic support generation and print time. Generating "bridges" require specific settings to be successful. This is one of the advantages of having printing profiles that Smart3D Printers include, all pre-configured for this purpose.

b. **Chamfers:** These rounded areas, created with tools such as "fillets" in the design software, are undoubtedly visually attractive. However, if we zoom in and perform a small "mathematical" analysis, we will conclude that these zones have; small transition zones with an inclination angle far less than 45-degrees, an incorrect finish, and the slicer will suggest generating support in these areas. We recommend avoiding this tool and using a 45-degree "chamfer" instead for better results and no support needs.

c. **Orientation:** Many times, changing the orientation of the piece in our printing area can have a great impact on the total printing time and the final quality, given that an area of a piece that remains in the air with a certain orientation, can be perfectly printed if we rotate the piece.



d. **Modular pieces:** It is not often possible to avoid generating support. But it is always a good option to analyze the possibility of "splitting" the piece into two (2) or more parts. Design it with post-assembly in mind, and in this way, it will be easier to print since we will have more and better positioning options that will safely prevent us from using media.

Overhangs

An overhang is defined as a part of a model with an inclination less than 45-degree, taking the horizontal line as zero reference grade. At inclination levels below 45 degrees, a portion of each line of the outer perimeter will no longer be supported by a layer below and will be printed mid-air. The plastic begins to behave differently depending on whether the section is a convex curve, concave curve, or straight. This distinction of geometries affects because it influences where the plastic will be contracted that is without sustenance underneath. Curling problems will manifest in these areas and may lead to collisions with the nozzle, and in the best case, irregularities in the surface finish.

When designing the model, take this into account. And, if possible, avoid modeling inclinations lower than 45 degrees. Threads and inverted fillets usually present these areas of overhangs, so they require special care to obtain the best results when they are printed.

Printing of small models / delicate details

It is possible to print incredibly small pieces or very delicate details with FDM/FFF technology by taking precautions when designing and printing. As already stated, the deposition of material is done by melting plastic. This heat must be allowed to dissipate, to solidify the material before new layers can be deposited. If this is not the case, more heat and weight will be added with each new layer, aggravating the problem further. Materials that are prone to retain heat, such as PLA, can cause more severe problems. There are several solutions and strategies to address this:

a. **Reduce the speed:** Reducing the print speed will allow more time for each layer to cool down.
b. **Layer ventilation:** The layer fan has to work in its just a measure. If we have a material cooling problem, increasing the airflow is an option to cool down the deposited material and dissipate the

surrounding heat. This is not applicable to Macro printers as high temperature materials are used, not PLA.

c. **Print several objects at the same time:** "Slow down" the print to give each new layer more time to cool down by multiplying or adding, objects to the printing session. Other times we are faced with situations where the model has delicate details. When designing the object, it is necessary to keep in mind certain minimum values that must be respected. The standard nozzle is 0.4mm which means that the minimum width of the line that the printer can make is 0.4mm. Taking this into account, it is where we should not design sections, walls of the object that are thinner than a multiple of this value. Ideally, always respect (for the case of this nozzle measurement) a minimum value of 0.8 mm (2 perimeters) and it would be advisable that this value is not less than 1.2mm (3 perimeters of 0.4mm), since this value is the one that ensures an acceptable minimum strength for the walls of any object. These values can change according to the measure of the nozzle, the layer height, and of course the type of material used. This rule is the first one that breaks when we build manual supports, where we typically build a structure of only 1 perimeter wide, so as to make it delicately weak and therefore easily removable.

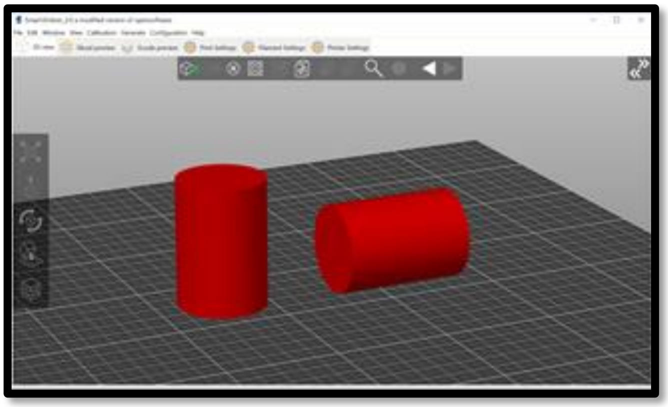
Multipart / large pieces

Not necessarily is the size of an object limited by the printing size of our machine. We can build much larger objects if we split models into multiple pieces to be assembled after being printed. With this in mind, we can use snap-on techniques to affix assemblies.

Orientation of the piece

Each material has its characteristics which will reflect on the printed object. The weakest point of any print by FDM will always be the adhesion between the layers along the Z-axis. It is worth taking strength requirements into account when designing, not only printing, so we are not restricted in its orientation to achieve maximum strength. The orientation of the model will also affect, in some cases, the surface finish quality. In the example, printing a cylinder will not give the same results if we choose one or the other orientation.

Printing the cylinder vertically will give us the best surface finish, but will make it weaker along its length (tending to break at the junction of some layer if subjected to force). On the other hand, if printed "laying down" the cylinder will be much stronger along with its extension at the cost of quality surface finish.



Holes and cylinders

While printing a circle, the melted plastic is deposited as the nozzle moves to form a circle. Due to the nature of the deposition technique, in reality, more plastic will be deposited on the interior side than on the exterior side. The result is that the inner circumference is always less than it should be and must be taken into account when designing the object.

There are two (2) ways to deal with this problem:

a. Run a test print and compare the actual level of contraction of the holes with the theoretical measurements and based on this, calculate a compensation factor. As a general rule, a value of 2% is used as a compensation number for up to 10mm diameter guiding. That is: in our computer model, the holes should be modeled 2% larger.
b. The other method of approach is to use a wick. We know that our holes will definitely not have the exact size, but if we use hole measurements that correspond to measurements of standard wicks, then we can then simply review these holes with the corresponding wick and thus achieve an exact diameter to the desired one.

Important points to keep in mind:
- Holes/cylinders, once printed, can only be enlarged
- Holes/cylinders printed horizontally (perpendicular to the Z-axis) will have a chamfered shape in the upper part, not being entirely circular. The slicer will always detect support needs in these areas, but we can avoid them as they are not necessary.
- The holes/cylinders, printed vertically (parallel to the Z-axis) will always be more precise and neater than those printed horizontally
- Adding more perimeters will reduce the precision of the actual diameter of the printed hole. To improve, change the order of printing of the perimeters, beginning with the outer perimeter. Even so, it remains very difficult to achieve an exact diameter with holes 2mm or smaller. It is advisable to improve your diameter by using a wick after printing.

Threads

Modeling and printing threading using FDM/FFF is relatively easy if certain precautions are taken. The first of the tips is that when you model a thread, you must take into account the type of thread, to respect some of the rules mentioned above (overhangs, rule of 45 degrees). In this aspect, the best results come from ACME-type threading. ACME threading generates trapezoidal contours that perfectly respect the minimum inclination of planes of 45 degrees, which will not require the use of supports or bridges. The other advice: perimeters. To obtain the maximum resistance, we must use the maximum possible perimeter to generate a solid thread.

Design to strengthen the piece

Some final details. 3D printing can be used for prototyping or real-life parts. Whatever the case, it's generally desirable that the part has an endurance that allows its use for its intended purpose and/or allows tests to be carried out that validate that it's not a prototype. For this purpose, it is good to mention some details that affect the strength of a printed object, which can be taken into account when designing on the computer.

a. If the piece will have inserts for screws, the cylinders that house them will be weak by nature and will tend to break when they are fitted with a screw. Then, when designing these cylinders, it is good to place a lateral reinforcement, a fillet, or changer in its base, or if the cylinders are next to a wall, join it to it by one or several ribs that stabilize the cylinder.

b. Strengthen any wall of the object by adding an internal fillet or chamfer in the union to its base.
c. 90-degree corners will benefit in their termination quality if they are designed with a reconditioned termination instead of 90 degrees. In this way, the path followed by the nozzle is continuous avoiding blobs. This will also help a little to avoid warping in the corners.