

Design and printing guidelines



Introduction

A print job can be created in either of the following ways:

- By using the HP SmartStream 3D Build Manager
- By using plugins added to Netfabb and Materialise Magics

NOTE: HP SmartStream 3D Build Manager supports STL and 3MF files.

The software creates slices that the printer can convert to layers in order to prepare the job. This preparation process may take from 15 minutes to 2 hours depending on the complexity of the job. It can be done while printing other jobs. Once ready, you can select the job to be printed from the job queue on the printer's front panel. If an error occurs while processing, a message appears on the front panel, and the printer cancels the job. To retry, you have to resubmit the job.

The rest of this chapter may help you to avoid such errors.

File preparation

Tessellation

Before sending a job to a 3D printer, the model to be printed needs to be tessellated. That means that its geometry needs to be converted into triangles, which are used by the printer to create layers. It is very important to pay attention to this step: if not done correctly, it can cause problems such as inaccuracy or slow processing.

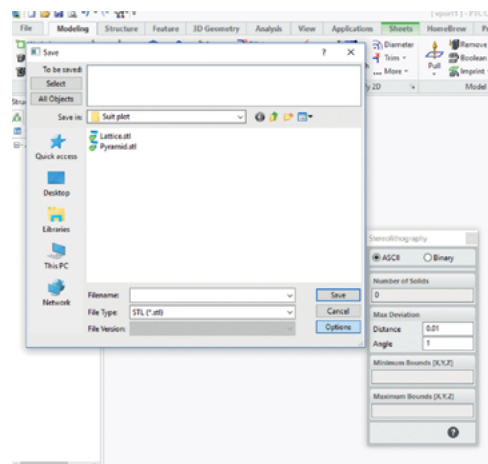
Standard formats in the additive manufacturing industry include 3MF (with more information about the model) and STL.

A normal file size for a model is about 1–30 MB, but the size depends on the type of software that created it, the number of triangles, the number and level of details, and so on.

When exporting to STL in a CAD package, you are often required to introduce some parameters such as angle tolerance and deviation chord height. These parameters define the resolution and file size of the part. The following tips may help you to export with the best surface to file size ratio.

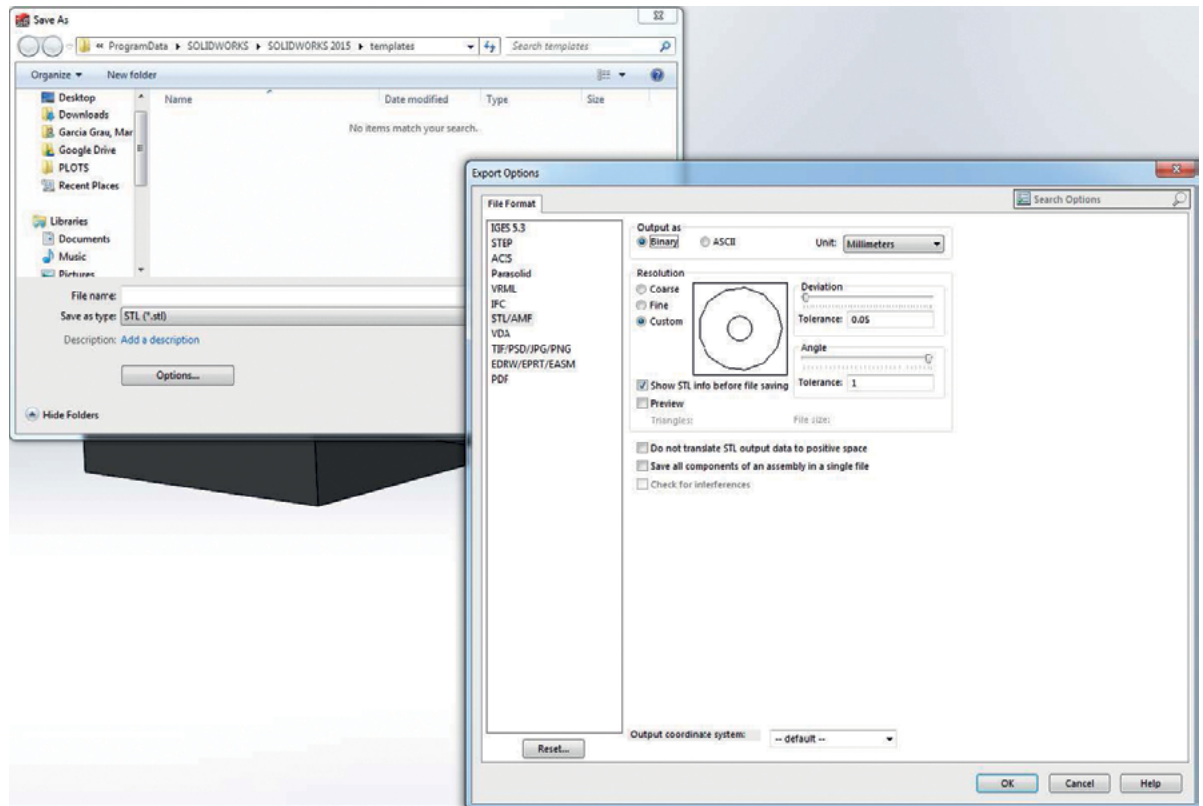
Solid Designer settings

Save as STL, click **Options**, and check the ASCII option. The recommended values for the Distance and Angle box are 0.05 and 1.



SolidWorks settings

1. Click **File > Save As**.
2. Select **STL (*.stl)** as the file type.
3. Click **Options**.
4. Choose the **Binary** option. Binary files are smaller than ASCII files for the same tessellation.
5. Choose **Custom** resolution.
6. Set the deviation tolerance to 0.05 mm.
7. Set the angle tolerance to 1°.



Rhinoceros settings

1. Click **File > Save As**.
2. Select **STL (*.stl)** as the file type.
3. Click **File Name > Save**.
4. Select **Binary**.
5. Select **Detail Controls** from the Mesh Options.
6. Set the maximum angle to 1, the maximum aspect ratio to 6, and the minimum edge length to 0.05.

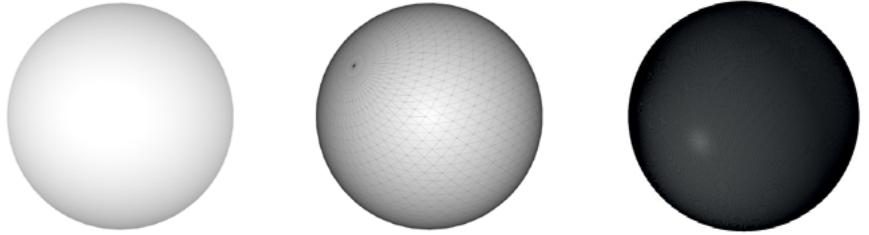
STL problems

- Too many or too few triangles

Too many triangles are difficult to process and, when a certain size is reached, the extra triangles do not provide any further accuracy. For this reason, an excess of triangles could increase processing time for no benefit.

Triangulation of a surface causes faceting of the 3D model. The parameters used to output an STL model affect how much faceting occurs.

Example of tessellation



- Holes in triangles

STL models commonly suffer from surfaces that are not joined to their neighbors, and missing surfaces.

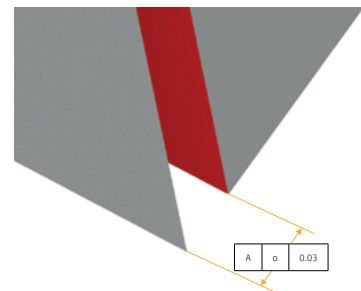
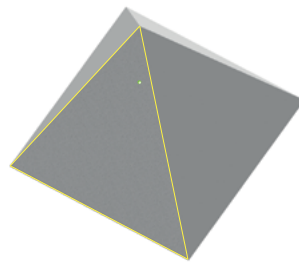
Repairing STL files

STL-repairing software

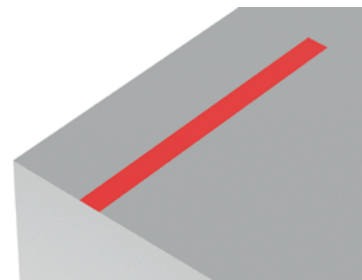
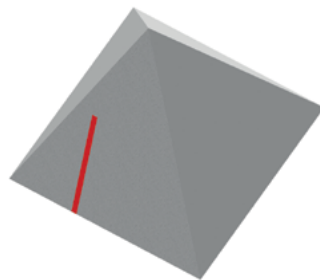
- Magics – Materialise
- Netfabb
- HP SmartStream 3D Build Manager

Common errors in STL files

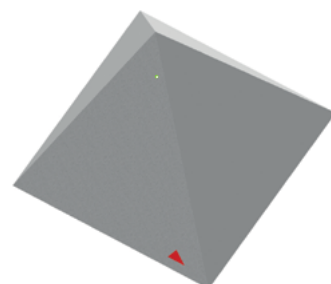
- Triangles not joined



- Overlapping triangles



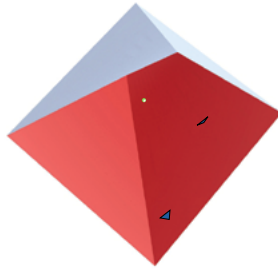
- Holes in parts



- Flipped-direction triangles



- Tiny shells

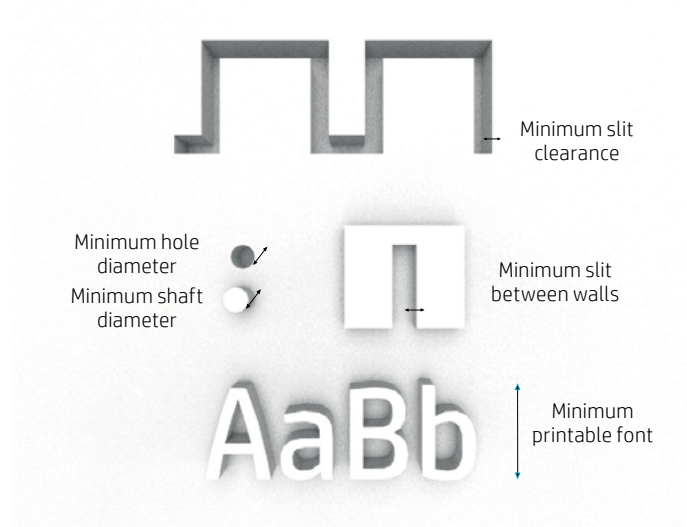


Minimum specification for parts

There are some specifications to bear in mind in order to avoid issues in parts, and to achieve the best quality.

The minimum printable features in planes X, Y, and Z are as follows:

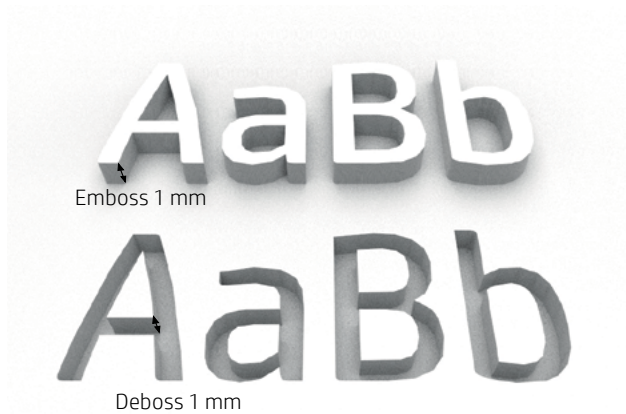
Minimum hole diameter at 1 mm thickness	0.5 mm
Minimum shaft diameter at 10 mm height	0.5 mm
Minimum printable font size for embossed or debossed letters or numbers	6 pt
Minimum clearance at 1 mm thickness	0.5 mm
Minimum slit between walls	0.5 mm



- **Engraving.** Multi Jet Fusion technology allows you to print letters and drawings with a very high resolution and definition.

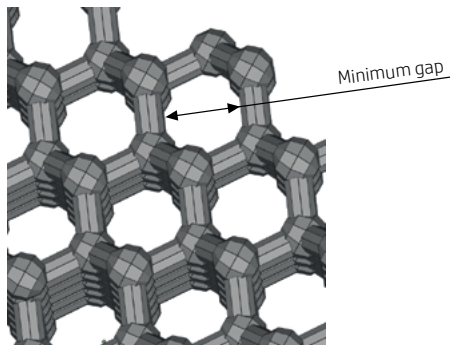
For the best possible output, any text, number, or drawing included in a part is recommended to have at least 1 mm of depth or height.

The best orientation for embossing letters is to place them upside down in the build chamber, while for debossing letters it is better to place them face up to achieve the best resolution.

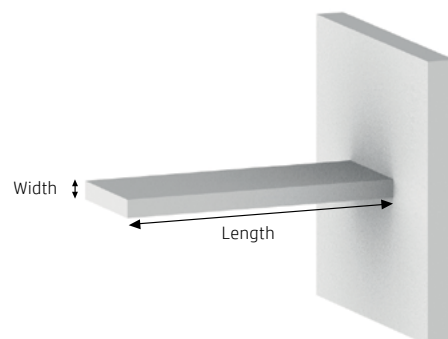


- The **dimensional accuracy** that can be achieved by HP Jet Fusion 3D is ± 0.2 mm up to 100 mm and 0.2% above that value, measured after sand blasting.
- **Solid part or structural fill.** Multi Jet Fusion allows you to print topology-optimized, generative designs or even small lattice structures. This kind of design helps to reduce the weight of the part and the quantity of material used, which not only reduces the cost of the part but also helps to reduce the operating cost in applications that are very weight-sensitive.

The minimum gap recommended in a lattice structure to ensure that all the material inside the part can be removed is 1 mm.



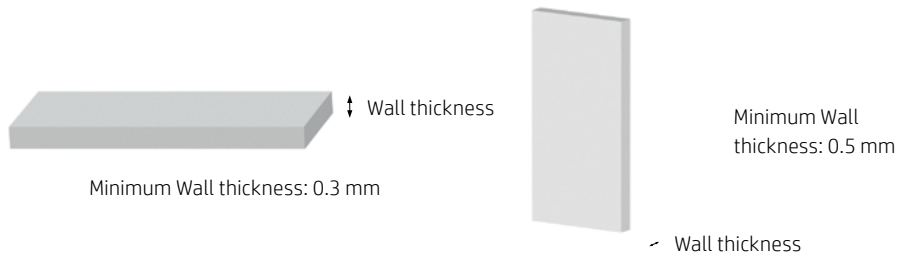
- When printing a **cantilever**, the minimum wall thickness depends on the aspect ratio, which is the length divided by the width.



$$\text{Aspect ratio} = \text{Length}/\text{Width}$$

For a cantilever with a width less than 1 mm, the aspect ratio should be less than 1. There are no specific recommendations for widths greater than 1 mm.

- In general, the minimum recommended **wall thickness** is 0.3 mm for short walls oriented in the XY plane, and 0.4 mm for short walls oriented in the Z plane.

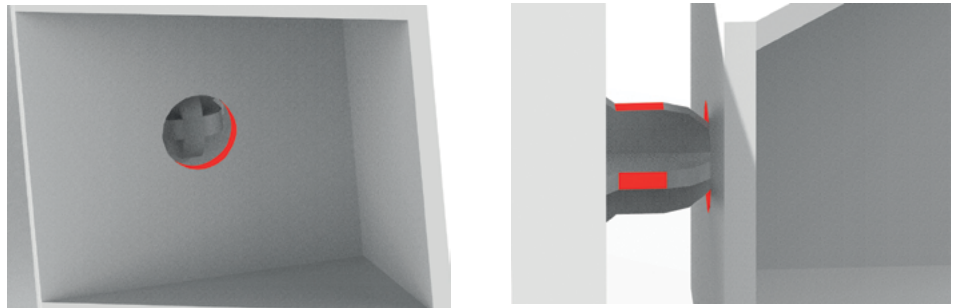


For parts with a high aspect ratio, you are recommended to increase the wall thickness, or add ribs or fillets to reinforce the part.

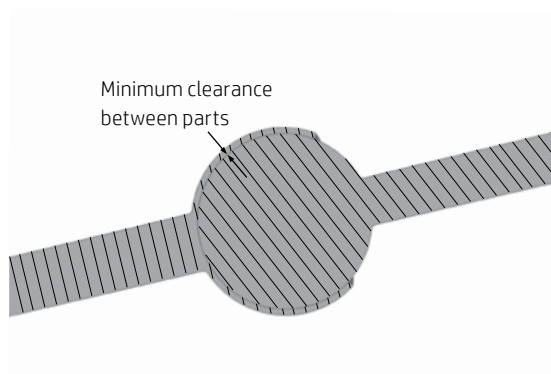
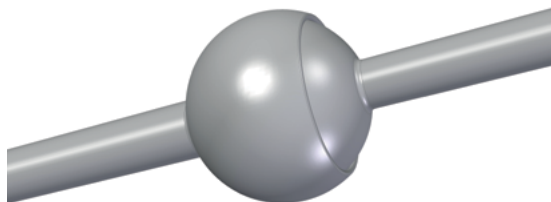
Very thick walls can accumulate heat and cause spot shrinkage in dense areas with an accumulation of material.

TIP: Hollow out the part as much as possible, saving agent and powder, and reducing sink marks.

- Minimum gap** between parts to be assembled after printing. Sometimes a pair of printed parts need to fit together for the final application. In these cases, you are recommended to have gaps of at least 0.4 mm (± 0.2 mm of tolerance of each part) between the interface areas that should fit together, in order to ensure correct assembly.



- Minimum spacing and clearance** between parts printed as assemblies. Assembly parts that are printed together should have a minimum clearance of 0.7 mm. Parts with very thick



walls above 50 mm should have a higher gap in order to ensure proper performance.

- **Hollow closed geometries.** Multi Jet Fusion is a process in which the parts are built by selectively fusing the desired areas of a material-based layer. If the parts printed are hollow, drain holes need to be added to the design to remove the material. The minimum recommended diameter of the holes is 2 mm. You are recommended to include at least two holes.
- **Glue lines.** Parts larger than the maximum build size can be printed with Multi Jet Fusion by splitting them into different parts. They can then be joined together by glueing, welding, or by pin inserts.

If you plan to glue parts together, you are recommended to include interlocking features such as those shown in the pictures below: as a guide to position the parts, to help them to bond together, and to facilitate the glueing process. Remember to leave an additional space of 0.1–0.2 mm between parts for the glue, in addition to the minimum spacing between parts printed as assemblies (see above).



- **Ducts.** To remove material from narrow ducts, consider designing and printing a strip or a chain through the duct. When the parts have been printed, you can pull out the chain to dislodge most of the material. Any remaining material can be removed by the normal cleaning process.



Other considerations

Contraction coefficient

The technology of the process produces a small contraction of the fused area in each layer. For the best possible accuracy, the parts should compensate for this contraction to be as close as possible to the correct size, and this is already done by the HP SmartStream 3D Build Manager, and by plug-ins for Magics and Netfabb.

Further adjustments of the material contraction coefficient can be done through the HP SmartStream 3D Build Manager or any other similar commercial software.

Orientation

There are several considerations to be taken into account when the orientation of the part is defined:

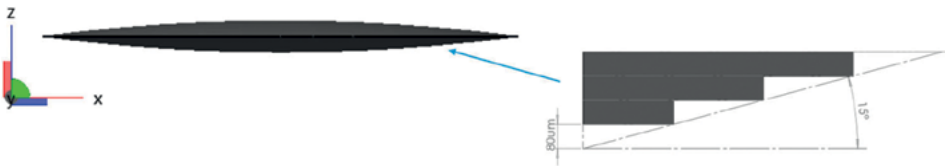
- Aesthetics
- Part curl
- Part strength
- Surfaces that need a higher level of detail
- Accuracy of the part

Aesthetics

Stepping is a phenomenon that appears when printing in Multi Jet Fusion. Multi Jet Fusion is a layer-by-layer process and, in some cases, the layers that form the part can be detected in the finished part.

In order to avoid stepping with curved or sloped surfaces, you are recommended to avoid whenever possible positioning those surfaces with angles less than certain values with the horizontal (XY plane), depending on the thickness of the layer. The minimum and maximum thickness values recommended for each layer are shown below:

Visible stair stepping		
Layer thickness (microns)	Minimum angle	Maximum angle
80	0	20



Accuracy of the part

The accuracy of the parts is better in the XY plane than in the Z direction. Features that require the highest resolution should be placed in the XY plane.

Cylindrical bodies should be oriented vertically as far as possible.

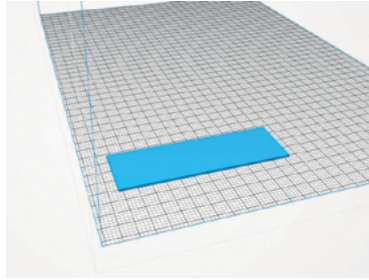


Engraving

The best orientation for embossing letters is to place them upside down in the build chamber, while for debossing letters it is better to place them face up to achieve the best resolution.

Warp minimization

For best print quality and to minimize warping, long flat surfaces should be positioned flat in the horizontal plane. This is especially recommended if the parts are long and thin. Place the parts as low as possible in the build chamber.



For critical parts, you are recommended to place them in the center of the bed to get the most uniform cooling. Minimization of the height in the Z plane will also help to avoid warping.

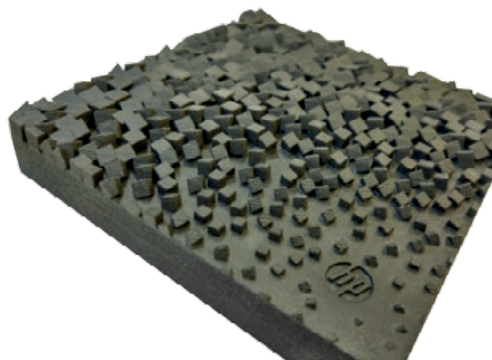
There are other design considerations that can be used to reduce warping, such as increasing the thickness of the part, or adding ribs in the areas that may be affected.

Curved surfaces

Smoothness is best achieved with surfaces that are upside down in the build chamber. If a part needs a particularly smooth surface, it should be positioned upside down.

Sharp surfaces

Sharp surfaces should be printed facing up.



Mechanical properties vary with orientation

The best mechanical properties of a part are achieved in the XY plane (horizontal). Any feature such as a pin or clip that needs to hold a load should be positioned horizontally whenever feasible.

TIP: Hollow out the part as much as possible, saving agent and material.

Positioning of parts in the build chamber

If the part needs to be as accurate as possible, avoid positioning it near the sides of the chamber.

Solid and massive parts are best placed close to the center of the print bed.

Repeatability

For maximum repeatability, try to keep the same printing conditions.

Distance between parts in the build chamber

The distance between parts should be at least 2 mm for the best printer stability.

Printing a part for final production

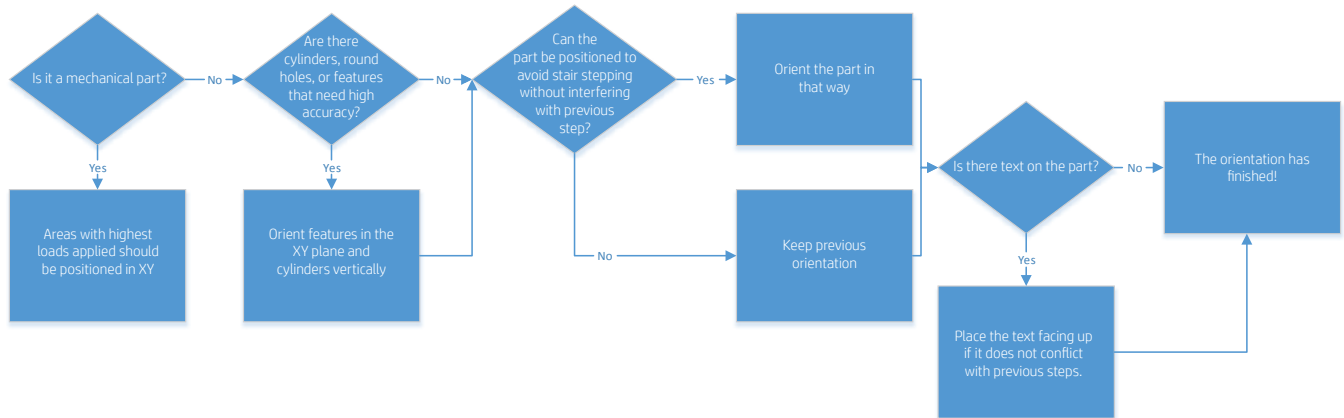
If a part is being made for final production, first try a small job with the part printed in different orientations, so that you can select the best one(s) to use for volume production.

Orientation flowcharts

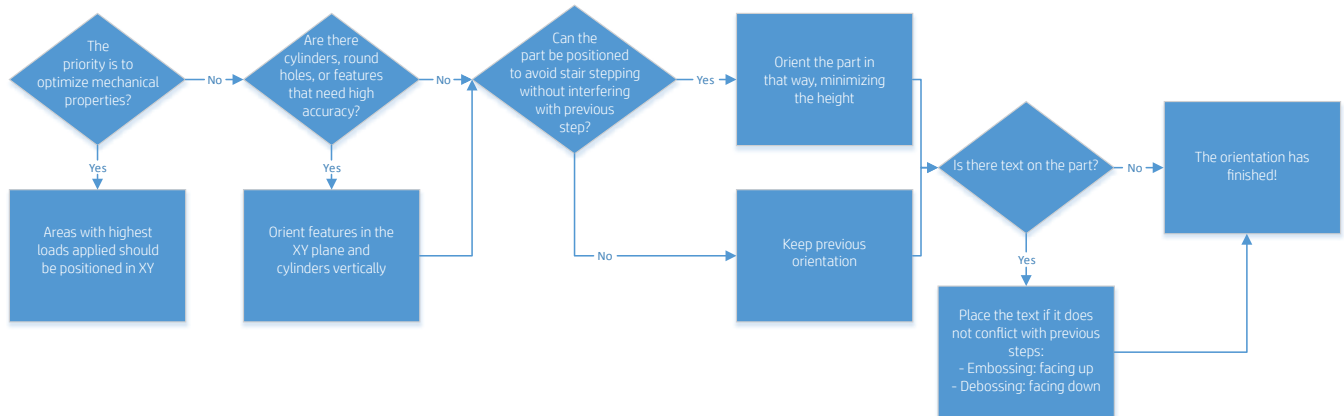
The best orientation of a part in the build chamber depends to a great extent on the final use of the part and the properties that you want to optimize. The main attributes that can be optimized are mechanical properties of certain areas, accuracy of specific features, and the look and feel of the part.

Depending on your objectives, the following workflows are recommended:

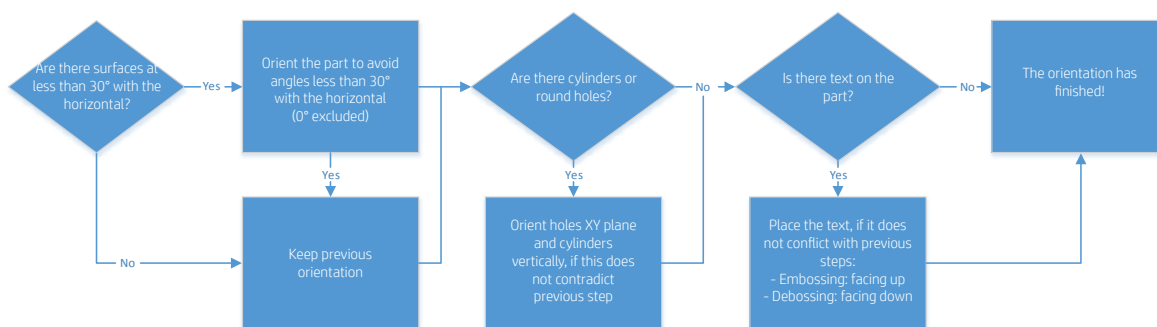
Optimize mechanical properties



Optimize accuracy



Optimize look and feel



Generating a sinter box

Small or delicate parts can be protected by generating a sinter box. Some software will automatically generate one. It also helps in tracking, and names or QR codes can be added with all the information about the parts.

