DESIGNING FOR ADDITIVE MANUFACTURING



While 3D printing is known for design freedom, this manufacturing method still has its own hurdles when it comes to design. Appropriately named additive manufacturing, each layer builds on the previous one and needs physical substance to build on. Additional design guidelines are result of the printer technology, and the method in which a part is printed.

3D Printing vs. Traditional Manufacturing Methods

3D printing allows for freedom in the design process not always possible with other methods of manufacturing.

Design Benefits

There are less limitations when designing a part for 3D printing. Wall thicknesses do not have to be consistent throughout the part, and they can also be much thinner than that of a part that is injection molded or machined. In addition to this, draft angles do not have to be considered like they do when designing for injection molding.

Features like hollow cavities, pockets, and undercuts are much easier to achieve with 3D printing than they are when injection molding or CNC machining. 3D printing does not require considerations of sprues, like that of injection molding, or the use of multiple tools, like CNC machining. Organic shapes are much more achievable for printed parts than for machined parts as a result of printing technology and the design freedom that comes along with it.



Cost Benefits

An additional benefit of 3D printing in comparison to other methods of manufacturing is the low cost of an individual part. For most traditional methods of manufacturing, it is not cost effective to build parts at low quantities. However, with additive manufacturing, thousands of end-use parts can be printed at a low cost as opposed to the high cost of a mold or machining labor hours.

There will always be a point in which using a different method of manufacturing will be the cheaper method once a large enough quantity is needed. However, design changes are often expensive and have long lead times for parts manufactured with traditional methods. If a high quantity is needed, the design of a part can be tested and changed in a matter of hours, rather than days or even weeks required to implement design changes with injection molding or CNC machining. This allows for the design of a part to be perfected at a low cost before going through with implementing the final design, rather than having expensive design changes.





B9Creations Design Guide

When designing parts for 3D printing, there are a few design elements to keep in mind in order to ensure your print turns out as desired.

The design and supporting of a model varies based on the resin used.

Overhang Angle



Overhang angle is the angle between the build table and the part. The closer overhanging features are to being parallel with the build table, the less selfsupporting they will be.

Minimum Self-Supporting Angle: 45° - 50°

Angles less than 45° require supports

Length of overhanging features is important – longer overhanging surfaces need more supporting

Vertical Wall Thickness



Vertical walls are walls that are perpendicular to the build table. The thinner they are, the less structural integrity they have.

Minimum Thickness: 0.1 - 0.4 mm

Thin walls connected on more than one side are more likely to print successfully

Horizontal Wall Thickness



Horizontal walls are walls that are parallel to the build table. The thinner they are, the less structural integrity they have.

Minimum Thickness: 0.1 - 0.3 mm

Walls connected on more than one side are more likely to print successfully

Horizontal Bridge Distance



A horizontal bridge is a bridge that is parallel to the build table and supported at both ends.

Maximum Distance Without Supports: 1 - 8 mm

Bottom layers of bridge prone to warping first



Horizontal Hole Diameters



Vertical Hole Diameters



Horizontal holes are holes that are cut parallel to the build table and run in either the X or Y direction.

Minimum Diameter: 0.25 - 0.70 mm

Holes smaller than the minimum will not print successfully

The bigger the hole gets, the less self-supporting it becomes

Vertical holes are holes that are upright and run in the Z direction.



Pin Diameter



Minimum Diameter: 0.25 - 0.5 mm

Vertical holes smaller than the minimum will not print successfully and will blend in with surroundings

A pin is a rod extruding from a surface, at an angle between 0° and 90°. The thinner the pin, the less structural integrity it has.

Minimum Diameter: 0.1 - 0.3 mm

Same considerations as that of an overhanging feature if applicable

Same considerations as that of a horizontal bridge if applicable

Hollow Cylinder Wall Thickness



Hollow cylinders are cylinders that are not filled in. They typically print best if they are perpendicular to the build table to avoid having to place supports within the cylinder.

Minimum Thickness: 0.1 - 0.3 mm

If cylinder is too small, the hole on the inside may fail and the feature will print as a solid cylinder

Ratio of Material to Print Size



This ratio describes the amount of material necessary in comparison to the size or build volume of a print.

A print that is large is going to lack structural integrity if it is made up of only very thin walls that do not connect frequently

In order to have a stable print, the part needs to be thick enough with sufficient connected feature points so that it can support itself throughout the print process



Hollow Cavities and Pockets



Embossed Details



Engraved Details



Engraved details are details that are etched or inscribed into a surface. Typical details that are engraved include words, logos, patterns, etc.

Hollow cavities and pockets tend to lead to suction forces which can ultimately

Part orientation is important – orient parts so that open pockets are perpendicular to build table Hollow cavities and pockets whose orientation cannot be fixed require escape holes, or holes

Embossed details are details that are raised up from a surface. Typical details

that are embossed include words, logos, patterns, etc.

Height is dependent on the width - the wider the feature, the taller it can be

Minimum Width: 0.1 - 0.2 mm

lead to failure of the print.

that relieve suction forces

Minimum Width: 0.1 mm

Depth is not dependent on width – as long as the width of the engraved detail is greater than the minimum, the depth can be as deep as needed

Model Orientation

The orientation of a part on the build table is a critical factor to success. The three things to consider when orienting a part are dimensional accuracy, where surface finish is important for the end use, and minimizing the number of supports to be added.

- Dimensional accuracy needs to be considered given that the first layers of the build are often over-cured in order to ensure adhesion to the build table, which can affect the dimensions of those first few layers.
- Surface finish is important because there is slight damage to the finish of the areas in which supports are to be removed, therefore the most important surfaces should be oriented away from the build table to avoid the need for supports.
- Minimizing the number of supports minimizes the areas in which the surface finish will be damaged from support removal.

Features like hollow cavities and pockets allow for suction forces during the print process. Adjusting the orientation of the part so that it is positioned in such a way that the opening to the cavity is perpendicular to the build table fixes this problem, as it does not allow for resin to be trapped between the part and the vat. However, doing so is not always feasible. To combat this, adding escape holes is recommended in order to relieve the suction forces, allowing material to escape and air to be let in.

*All information will slightly vary depending on the different resin and resolution of your print.



