

# 3D PRINTING BEST PRACTICES

A Guide to Getting  
Started with FFF  
3D Printing





## INTRODUCTION

**Additive Manufacturing (AM)**, popularly known as 3D printing, refers to a family of processes used to fabricate three-dimensional objects through the addition of material. AM contrasts with many traditional manufacturing methods that rely on subtractive processes, such as chiseling a statue out of marble or turning down an acetal rod.

**Fused Filament Fabrication (FFF)** is an AM process that creates parts by printing layers of molten plastic on top of one another until a desired geometry is formed. FFF is one of the most common forms of 3D printing due to the superior integrity of parts created and its low entry cost.

Common applications of FFF include rapid prototyping, concept modeling, small run production, fabrication of objects such as fixtures, tooling, custom components, replacement parts, and much more. Parts can be made in a range of thermoplastic materials including, but not limited to, ABS, PLA Acetal, rubber-like plastics, PET, high temperature plastics, and composite materials.

This guide is written to familiarize those new to 3D printing with fundamental design considerations to begin producing useful parts from the start.

## Table of Contents

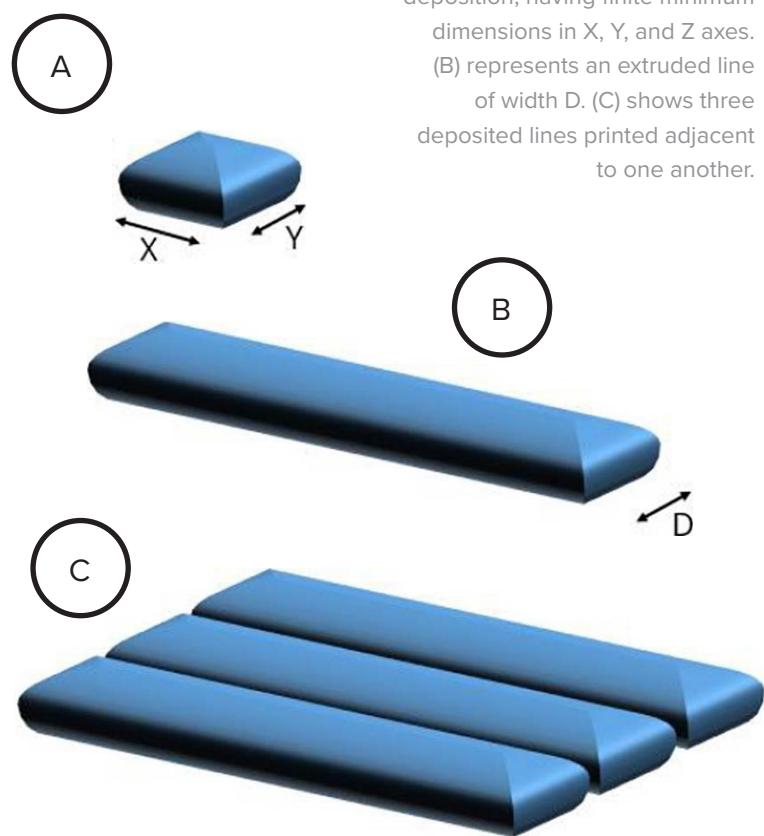
2	Introduction
3	Process Fundamentals
4	Layer Height
5	Overhangs and Bridges
6	Part Strength
8	Tolerances
9	Thermal Distortion
10	Holes
10	Wall Thickness
11	Post Processing

## PROCESS FUNDAMENTALS

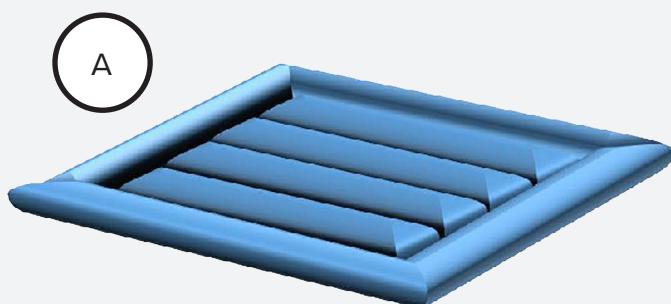
The minimum feature size for a 3D printer is equal to the size of a deposition. A deposition is smallest satisfactory deposit of material that a 3D printer can produce. Depositions are often approximated as **voxels**, the three-dimensional equivalent of a pixel.

In the FFF process, the deposition size is constrained by the diameter of the extruder nozzle and the resolution of the Z stage. The diameter of the nozzle defines the lower limit of the deposition in the X and Y axes, and the Z resolution defines the lower height limit of the deposition.

A 3D printed part may be thought of as the sum of a set of voxels. However, by the nature of the machine controls, FFF is a vector-based process. Accordingly, each layer of a 3D printed part made with FFF may be better thought of as the sum of a set of vectors representing the plastic strands of a printed part, as shown in Figure 1(B) and 1(C). An entire part made with FFF may be thought of as the sum of a series of such layers stacked one on top of another, as shown in Figure 2.



**Figure 2.** (A) represents a single printed layer. Note that each layer is made from at least one external perimeter and a pattern filling in the bulk of the area. (B) represents a basic 3D printed part, three stacked layers.





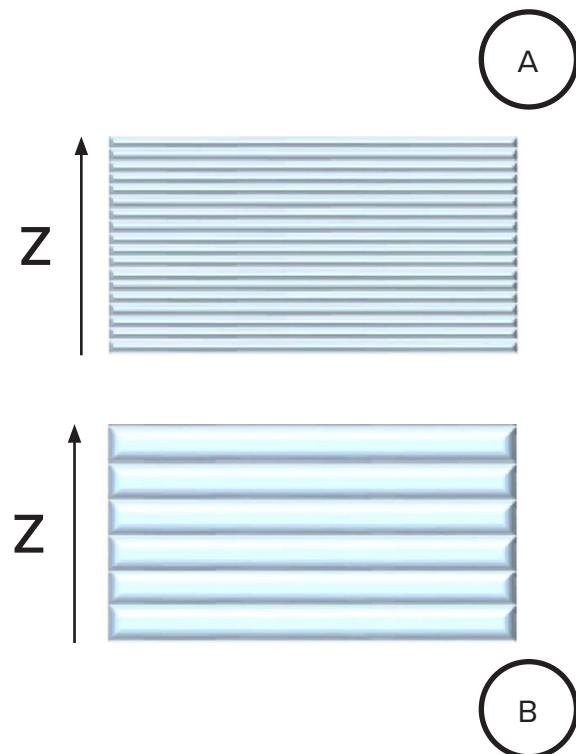
## LAYER HEIGHT

Nearly every **FFF** 3D printer is capable of printing in varying layer heights. This may be represented as a “quality” setting in some 3D printing software. Although many printers can print a variable range of layer heights, when working with an individual machine it is most important to know the difference between parts printed with thin layers (high quality) and thick layers (draft quality).

Parts printed with **thin layers**, as in Figure 3(A), come with several benefits. The most apparent benefit is smoother surface finishes - individual layers may not even be obvious to the naked eye. Parts printed with thin layers also have the added benefit of added strength due to greater fusing between layers. Finally, these parts are less prone to leaking when used as a vessel for liquids, also due to the greater fusing between layers. The primary drawback of printing parts with thin layers is the associated increase in printing time, due to the greater number of layers needed.

Conversely, parts printed with **thick layers**, as in Figure 3(B), have the primary benefit of a quicker printing time. However, they suffer from rougher surface finishes and poorer fusing between layers. These parts will typically serve as suitable visual models, but may not stand up to the requirements of a functional prototype or an end-use part.

Selecting the proper layer height for a given application is thus a balancing act with part strength and surface finish on one side, and print time on the other.



**Figure 3.** (A) is a side-view representation of an FFF part printed with thin layers. (B) is a side-view representation of a part printed with thick layers.

## OVERHANGS AND BRIDGES

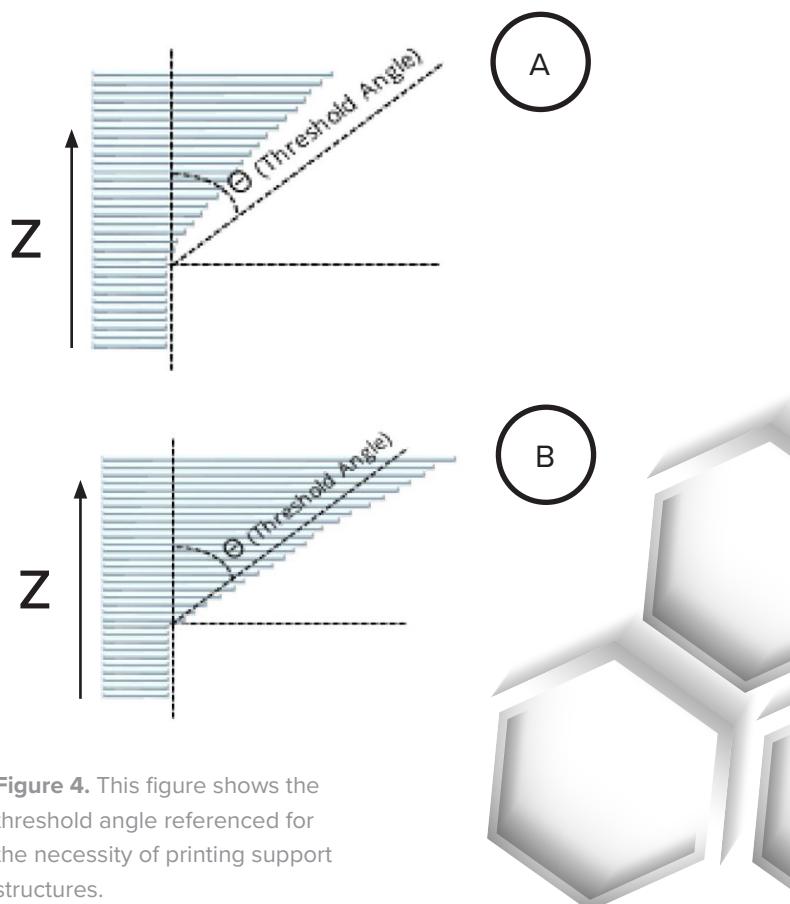
The FFF process relies on the ability to print a flat layer of molten plastic and then another and then yet another on top. But what if the next layer is bigger than the previous layer? Will it print acceptably or will it droop downwards?

Nearly every 3D printer has the capability to utilize scaffold structures in order to print complex geometries. These scaffolds are called support structures.

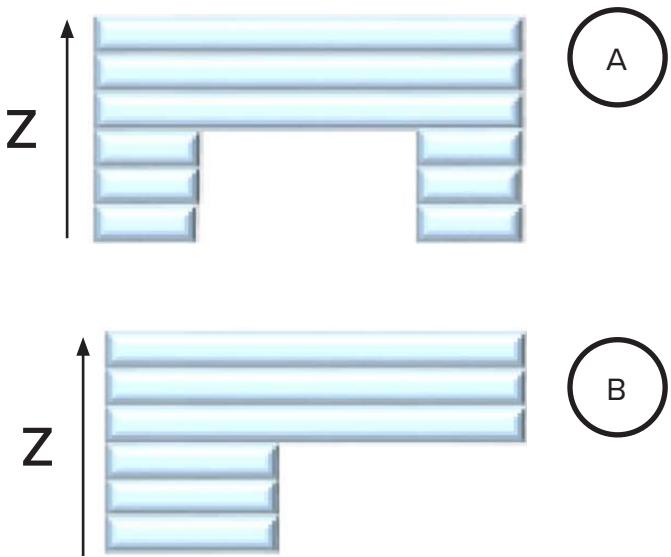
With FFF printers, there is a certain extent to which a layer can be printed larger than the previous layer and form with no issues. Taking advantage of this effect when designing can reduce or eliminate any time spent removing support structures after printing.

To eliminate the necessity of support structures while designing a part, use this simple rule of thumb: ensure that overhangs are closer to the vertical axis than the **threshold angle** of 45 degrees. Every 3D printer and material combination has a specific threshold angle, but assuming a threshold angle of 45 degrees will work with most printers. The threshold angle, shown in Figure 4, is defined as the angle from the vertical (Z axis) at which overhangs can be printed without compromising dimensional accuracy or causing a failed print job. Geometries printed at an angle greater than the threshold angle will require support structures, whereas geometries printed at an angle less than the threshold will not require support structures.

There is an exception to the overhang rule: bridges. Bridges are features which are printed on top of a void, but are supported at both sides. On printers that have active part cooling, bridges can be printed without the use of support structures. Shown in Figure 5(A) is a side-view representation of a bridge feature which would not require support material on a printer using active cooling. Contrast this with Figure 5(B), which shows an overhang that will likely require support structures for a successful print.



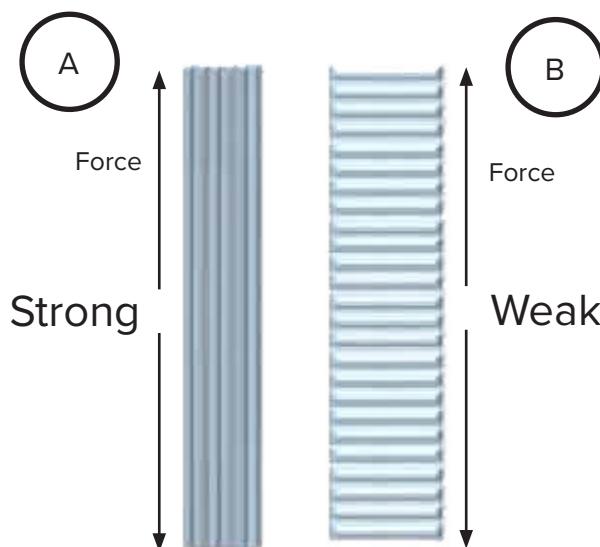
**Figure 4.** This figure shows the threshold angle referenced for the necessity of printing support structures.



**Figure 5.** (A) shows a part with bridge geometry. This part may not require support material with the right printer. (B) shows a part printed with an overhang that will likely require support structures for successful printing.

## PART STRENGTH

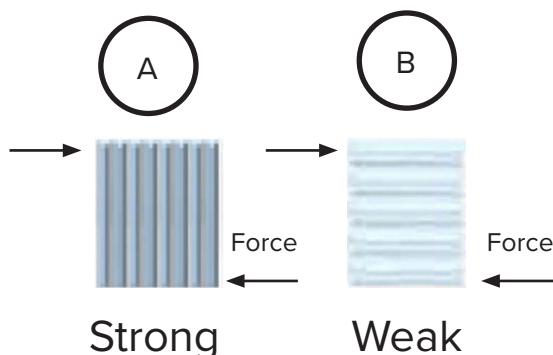
Due to the layer-by-layer nature of the FFF process, there is an asymmetry between the horizontal axes and the vertical axis. This means that print orientation affects bulk and local mechanical properties. Printed parts are strongest in tensile loading along the layers, in the XY plane. The lowest strength is across the layers, in the Z direction. One of the most common failure modes of 3D printed parts is delamination between layers.



### TENSILE STRESS

When loaded in tension, 3D printed parts are strongest along the vector paths of the printed layers. This can yield a result that is as strong as homogeneous plastic. When loaded perpendicular to the printed layers, a printed part can be much weaker. This is due to a layer delamination failure at a lower force than at which a part would normally yield. See Figure 6 for an illustration of this.

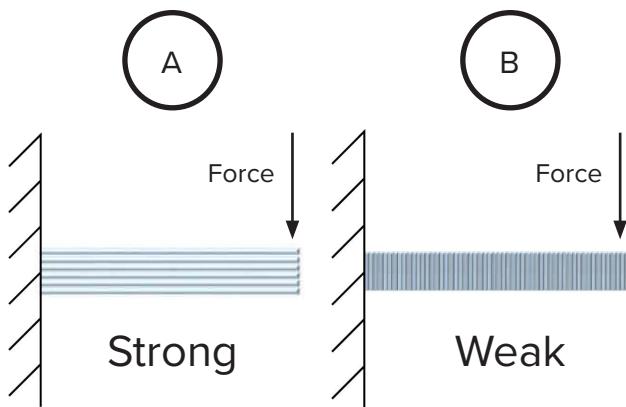
**Figure 6.** (A) represents a part that is loaded with the force applied along the strands of the printed part, an orientation that will perform nearly the same as homogeneous materials. (B) represents a part loaded with the force applied perpendicular to the printed layers, a configuration shown to be much weaker.



### SHEAR STRESS

The effects of shear stress also depend on the orientation in which a part is printed. Parts will typically fail due to shear stress if the forces are applied parallel to the layers, whereas parts loaded with shear stress applied perpendicular to the layers are typically stronger. See Figure 7 for a diagram of these differences.

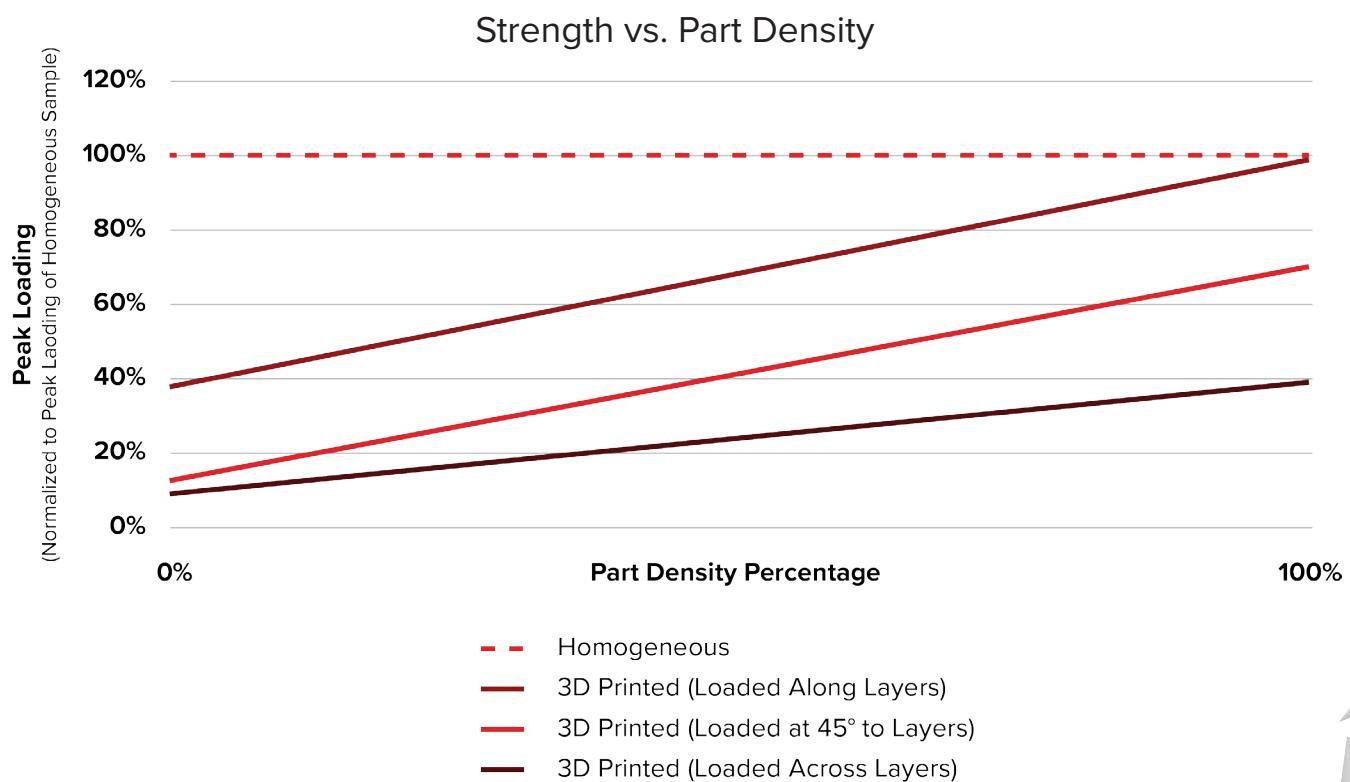
**Figure 7.** (A) represents a part loaded in shear with forces applied perpendicular to the printed layers. This configuration will typically be stronger than the configuration shown in (B) where the forces are applied parallel to the printed layers.



## FLEXURAL STRENGTH

Parts loaded with bending stresses are similarly affected by printing orientation. When bent, parts loaded with the applied force

**Figure 8.** (A) represents a configuration of bending stress in which the printed part will perform best. (B) represents a configuration of bending stress that will cause the part to fail at a relatively low applied force.



## INFILL DENSITY

Each layer of an FFF fabricated part consists of at least one perimeter and a pattern filling in the bulk of the interior, as in Figure 2(A). This filling material is referred to as the infill, which can be printed at varying densities commonly represented as a percentage in 3D printing software. Increased infill density correlates to increased part strength and thus a more durable part, but also increases print time and material usage. Figure 9 communicates the effect of varying infill density for several printing configurations.

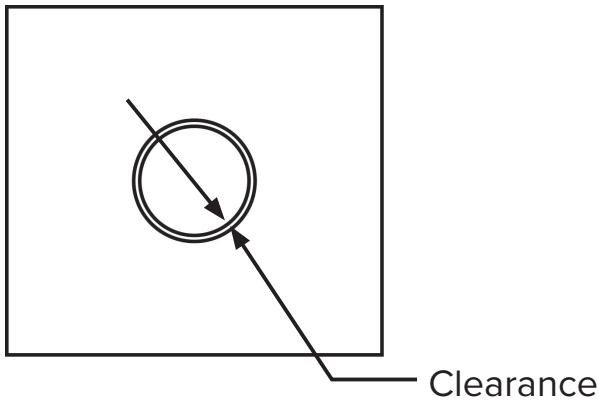
**Figure 9.** Infill density vs. peak tensile loading capacity of PLA samples. Tensile loading data is included for loading along the layers, on a 45 degree slant to the layers, and at 90 degrees across the layers.

## TOLERANCES

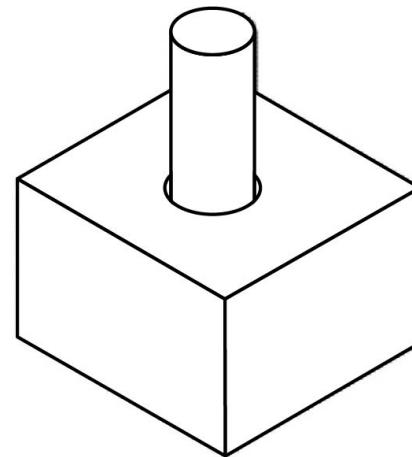
The accuracy of the printer must be considered when designing a part. FFF printers typically have a linear accuracy of 0.2% to 2%. Accuracy considerations are especially important when designing parts that need to fit together. Parts can be designed to give clearance fits, transition fits, or interference fits, just like traditionally produced parts.

Additive parts are often slightly oversized. Some 3D printing software provides control of a parameter known as **geometric compensation**, allowing the printing of parts which fit properly without having to alter CAD models.

**Figure 10.** Altering clearances can allow printing of slip fit, transition fit, and interference fit components.



Top View



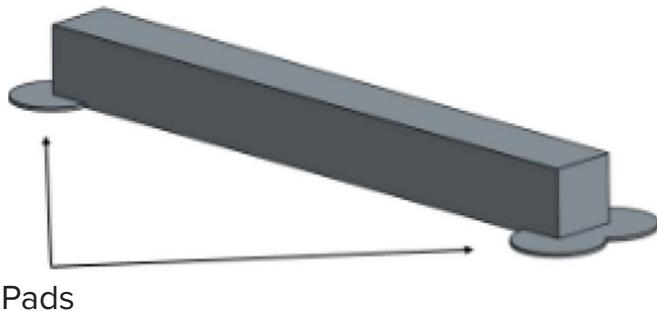
Isometric View

<b>Clearance Fit:</b> shaft rotates or slides freely	Clearance = 0.5mm - 2mm
<b>Transition Fit:</b> shaft is held precisely but can be disassembled	Clearance = 0.25mm - 0.5mm
<b>Interference Fit:</b> shaft requires force for insertion	Interference= 0.0mm - 0.25mm

## THERMAL DISTORTION

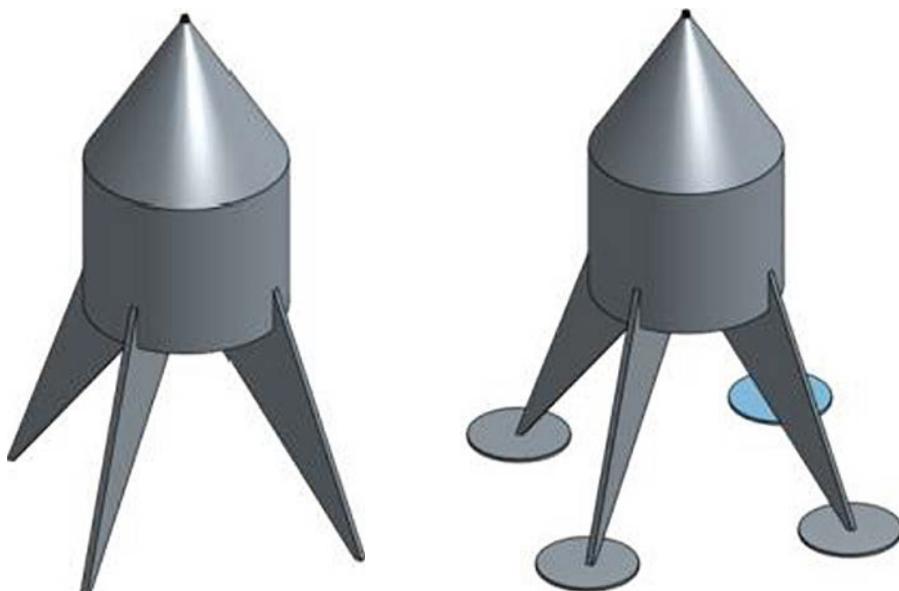
Depending on the printer and material, thermal distortion may become an issue as hot material is laid down upon partially or entirely cooled material. Thermal distortion causes **warping**, which typically manifests itself as the part pulling up and away from the print surface on the corners and edges. This may further lead to a part becoming detached from the surface mid-print, causing a failed print and lost time.

**Pads**, thin circular elements which are easily removed after printing, can help to prevent a part from coming loose mid-print. Pads are commonly added in areas where the part may not have sufficient surface area on the first layer to stay adhered to the print surface throughout the entire print. **Rafts**, or flat rectangular features printed underneath the entire part, can also help to mitigate warping and have the added benefit of compensating for small irregularities in the print surface.

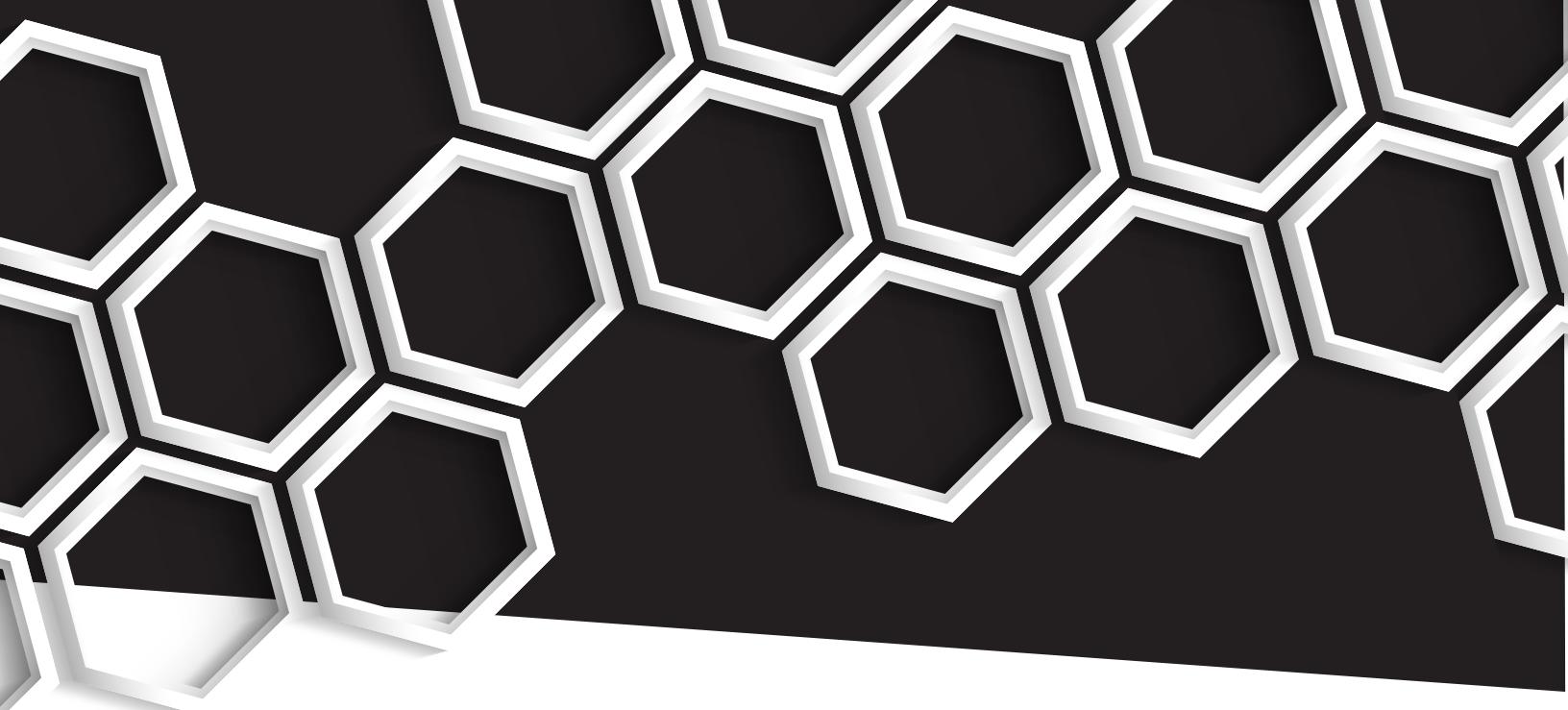


Pads

**Figure 12.** The addition of pads to a design can help to prevent parts with a small first layer surface area to adhere to the print surface and avoid becoming detached mid-print.



**Figure 11.** The addition of pads to a design can help eliminate warping effects. Pads are typically no more than 0.5mm to 1mm thick. In this case, these pads are most likely added to minimize the effects of warping.



## H O L E S

Negative space features such as holes are typically undersized on FFF parts. If very precise hole sizes are required, drilling or reaming is recommended to generate properly sized holes. If the 3D printing software offers geometric compensation, this parameter can be set to a slightly negative value to enlarge holes. This comes at the cost of shrinking positive space features such as pins or external dimensions by the same factor.

## W A L L T H I C K N E S S

The horizontal thickness of the perimeter of each layer is known as the wall thickness. A minimum wall thickness of 0.8mm or greater will typically yield an acceptably strong surface, depending on the printing material. The absolute minimum wall thickness is dictated by the nozzle diameter.

## POST PROCESSING

Most types of additive manufacturing, including FFF, are near-net shape processes. This means that these machines will create parts that are quite close to the desired geometry, but which for certain applications will require a bit of extra finishing work to get the desired results. Common reasons for post-processing include:

- **Removal of support structures** - this can be as simple as picking off support structures by hand or with pliers, or can be done with the aid of a machine such as a grinder or belt sander.
- **Improvement of surface finish** - conventional sanding, machining, or abrasive bead blasting can be used to mechanically smooth the surface. Also, certain print materials allow for the use of chemical smoothing agents. For example, ABS is soluble in acetone, so coating, dipping, or brushing an ABS printed part with acetone will cause the surface molecules to dissolve in the acetone and precipitate and re-bond with the part as the acetone evaporates, significantly smoothing the part.
- **Sealing or coating the part** - this can range from simple painting and dipping to epoxy infiltration and electroplating with metal.
- **Assembly of multiple sections** - this can be done by bonding or gluing sections together, by using conventional fasteners, or by designing parts with built-in assembly features such as threads or pins/holes.





7420 Kilby Road  
Harrison, OH 45030  
(513) 367-7100 | [info@e-ci.com](mailto:info@e-ci.com) | [www.e-ci.com](http://www.e-ci.com)

**CINCINNATI®**