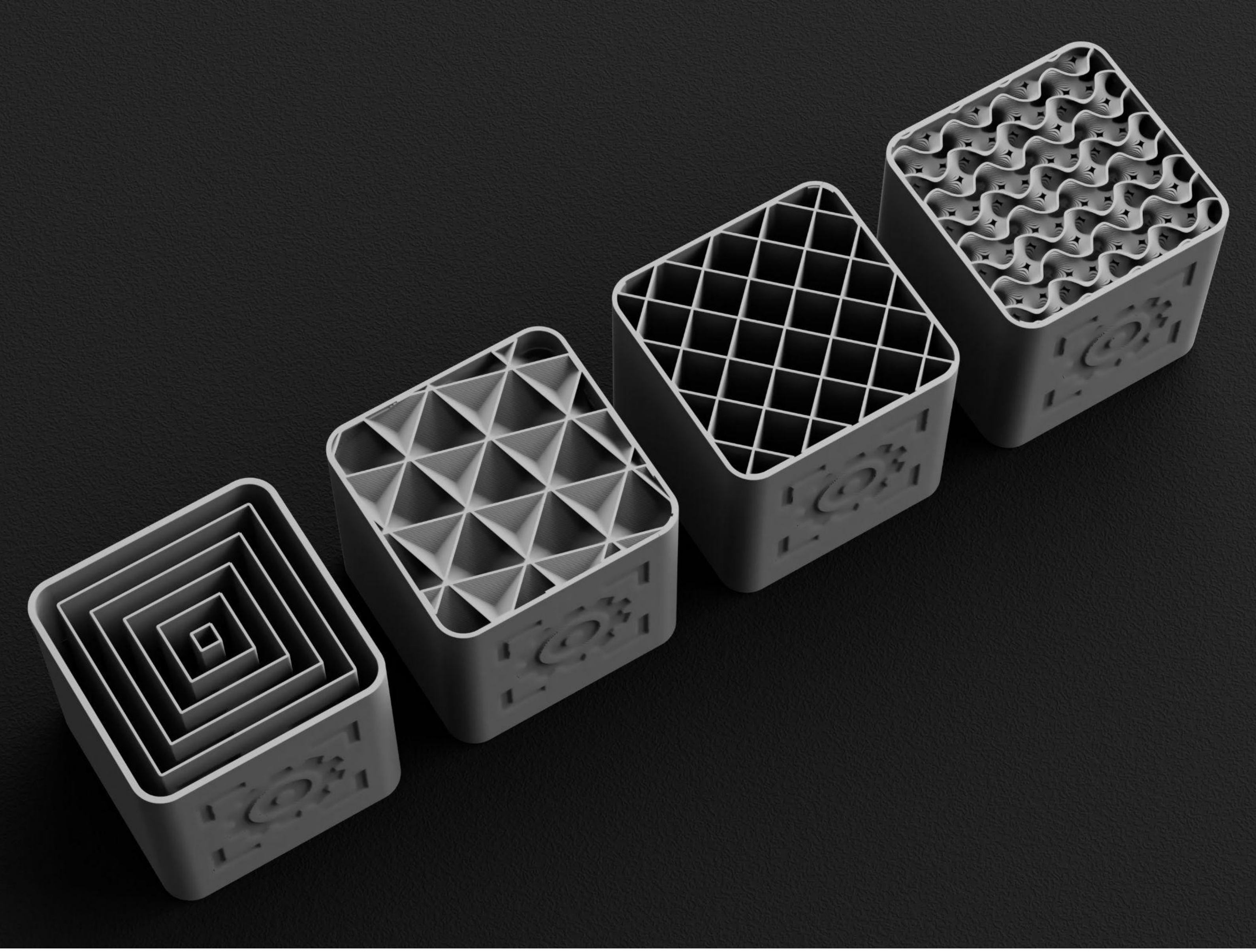


BCN3D x Flowalistik | eBook

# BCN3D Slicing Guide

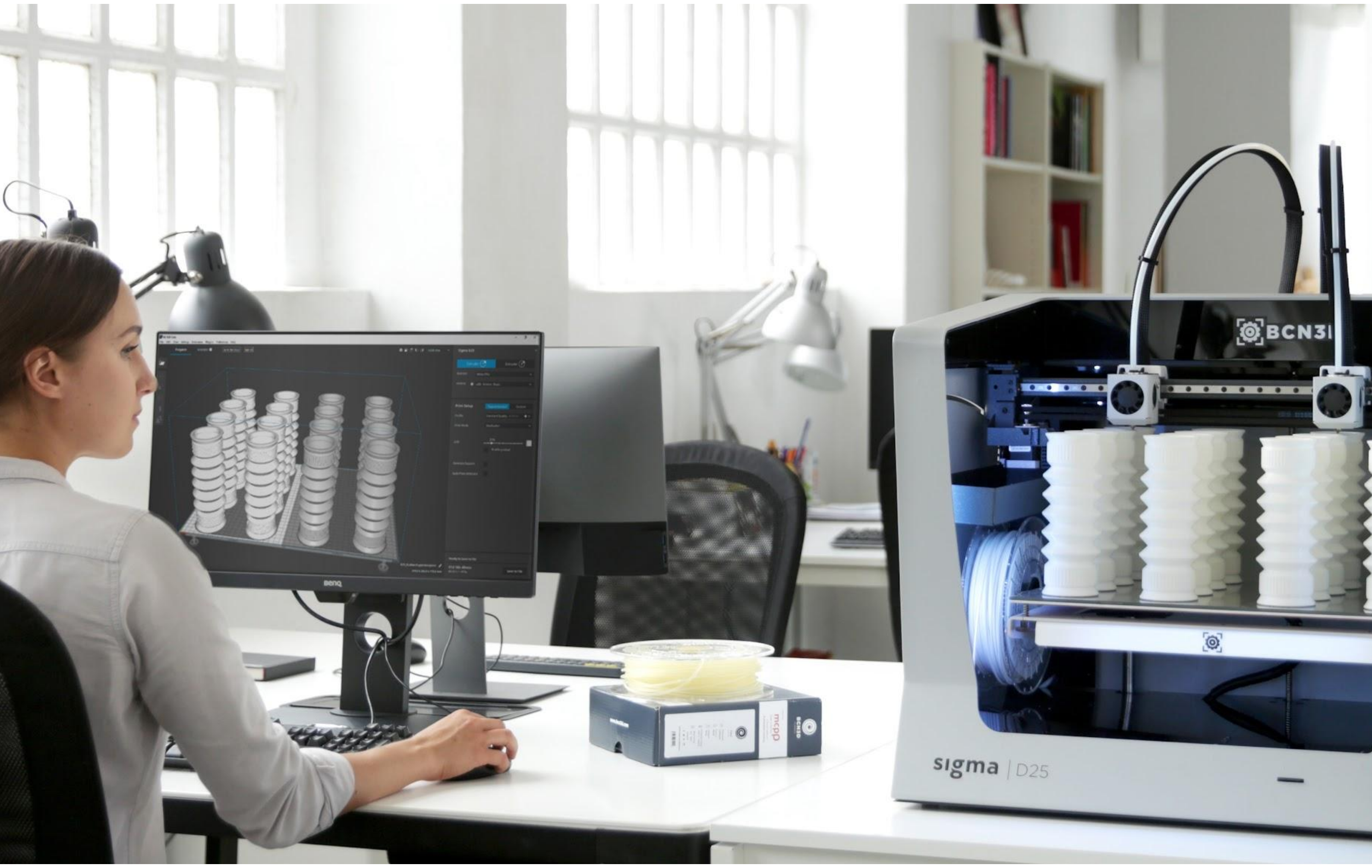
## Your all-in-one ebook

www.bcn3d.com

BCN3D

# Table of Contents

# 1. Layer height



# Introduction

At BCN3D, they know that a great 3D printing experience requires more than just making great hardware. For this reason, they've built an ecosystem where hardware, software, and materials work together, enabling innovators to create the future.

BCN3D Stratos is their advanced slicing software, and it allows you to make the best out of the BCN3D 3D printer and material family. This free and easy-to-use 3D printing software prepares your model for 3D printing. It provides an intuitive user interface and an improved workflow, both for newcomers and expert users.

In the BCN3D Slicing Guide, we go through the most relevant features you'll need to know when preparing a 3D model for fabrication. Learn with examples, practical cases, and technical information and become a slicing expert!

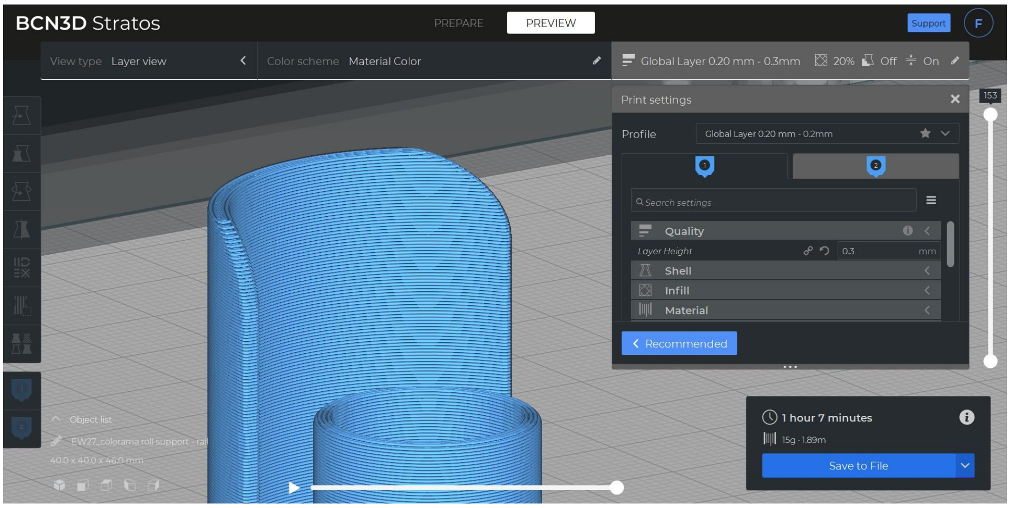
# What is layer height?

The layer height represents the exact thickness of each horizontal layer of your print. It's the most relevant setting, as it's heavily related to the additive manufacturing process present in all FDM 3D printers.

This setting has many effects on the 3D printing process, including:

- **Surface finish**: The lower the layer height is, the smoother the surface will be.
- **Print time:** Reducing the layer height by half will double the print time.
- **Mechanical properties:** Low layer heights can offer slightly better mechanical properties.

Layer height is also known as layer thickness or print resolution, and it's represented in millimeters or microns. A layer height of 0.2mm is the same as a layer height of 200 microns.




Layer View on BCN3D Stratos

# Common layer height options

The recommended layer height depends on the nozzle diameter. Taking as reference the standard 0.4mm nozzle diameter, we can say there are three main print quality options:

- **High Quality (0.1mm):** This layer height is recommended for high fidelity prototypes, end-use parts, models with small details, or designs where a smooth surface finish is required.
- **Standard Quality (0.15mm):** As its name says, this layer height offers a standard quality, combining good surface finish and mechanical properties without having a significant impact on print time.
- **Draft Quality (0.2mm):** Recommended for initial prototypes, simple models, and parts where surface finish is not relevant. Its main advantage is the reduced print time compared to the other two print quality options.

Other layer height options are available with the standard 0.4mm nozzle. For example, you could use layer heights of up to 0.32mm with that hotend, although the print quality would be lower. Such large layer heights are only recommended for early prototypes and designs with small overhangs.

Consider that larger hotends allow you to print with larger layer heights. The layer height can usually be up to 80% of the nozzle diameter to guarantee dimensional accuracy in the printed parts.

# How layer height affects the surface finish

## Surface finish

The lower the layer height is, the smoother the surface will be. Use 0.1mm layers if you want to print a model where layers are almost invisible, and 0.2mm layers (or larger) if you don't mind seeing the layers and having a slightly **rough surface**.


Layer height tests: 0.1mm (left), 0.2mm (middle), 0.3mm (right)

If you want to hide the rough surface caused by layers, it's recommended to use composite materials or filaments with matte finishes.

Materials with matte finishes visually hide the layers, but you can still feel them when you hold the item in your hand. If you truly want to hide the layers of your print, using composite materials is the best solution. These materials usually include fibers, powder, or other small particles that generate a unique texture on the extruded material.

## Small details

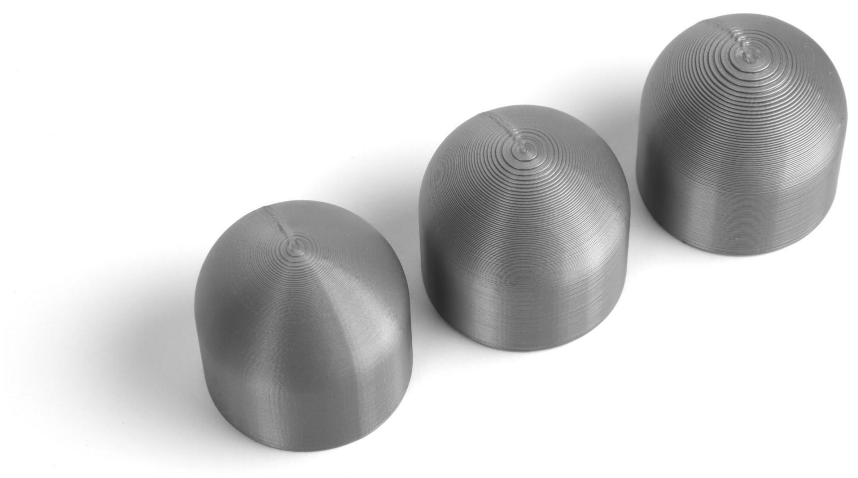
Use a lower layer height if the 3D model includes little details such as **text** or textures. Even though other print settings and the 3D model need to be considered to get the best possible quality, layer height is the most important.


Layer height tests: 0.1mm (top), 0.2mm (middle), 0.3mm (bottom)

As you can see in the image above, the text quality slightly reduces as the layer height is increased. However, the left logo starts losing its contrast as there are smaller elements on its corners.

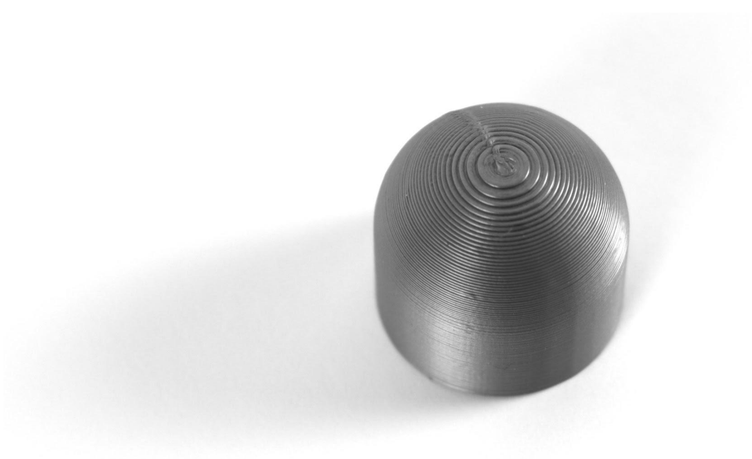
### Large vertical curves

Layers become more visible at the top of curved objects such as **spheres**. Layer height plays a key role when slicing this type of model as it can reduce or increase the number of steps you can see on the printed part.



Layer height and curved surfaces: 0.1mm (left), 0.2mm (middle), 0.3mm (right)

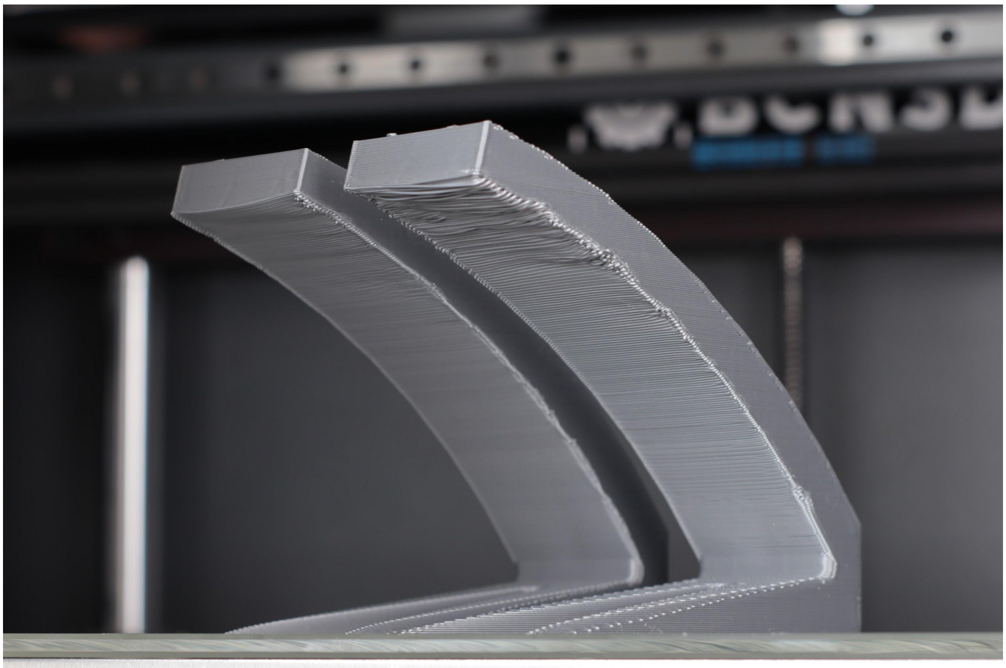
The manufacturing process causes the layer lines, and the best options to hide them are using a low layer height or post-processing the part.



Layer height and curved surfaces: 0.3mm test print

## Overhangs

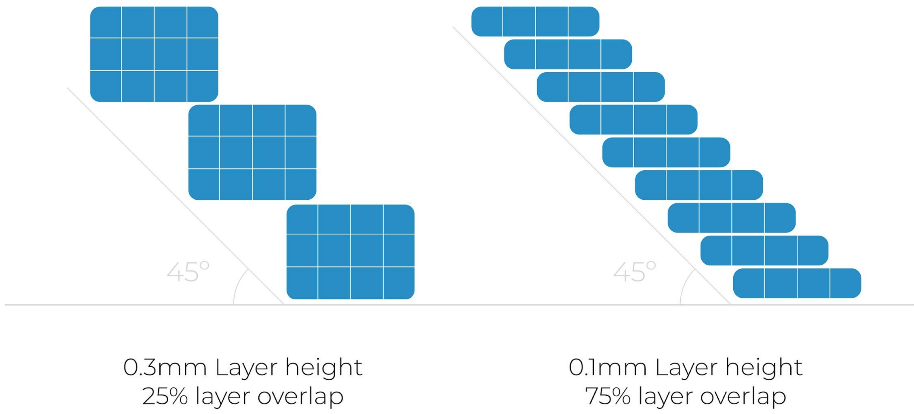
Layer height affects overhang quality as it determines how much the top layer will **overlap** with the one below. Overhang quality is also affected by print orientation, 3D model shape, and part cooling. However, layer height will play an essential role in those parts that combine overhangs with intricate details.



Overhangs and layer height tests: 0.2mm (left), 0.3mm (right)

In the image above, you can notice that overhangs print better when the layer height is lower. The test on the left was printed with a 0.2mm layer height, while the one on the right was printed with a 0.3mm layer height.

When thinking about overhangs and layer height, it's important to mention that the surface finish on overhangs depends on how much the top layer overlaps with the one below.




**Layer Height vs. Overhangs**

For example, if a 3D model has a wall with a 45° overhang, around 25% of the top layer will overlap with the one below if you use a 0.3mm layer height. However, if you reduce the layer height to 0.1mm, you'll notice that the overlap % increases to 75%. The more the top layer overlaps, the better.

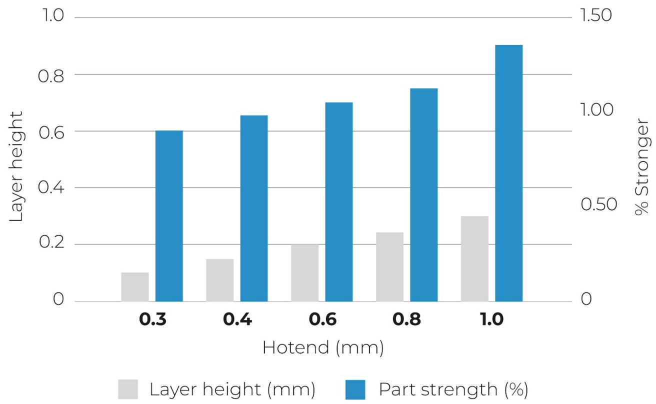
## Post-processing

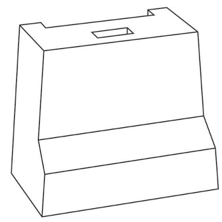
If you're going to sand, prime, or paint the 3D printed parts, you may want to consider using a lower layer height to save you time and improve the surface finish.

- **Sanding**: Eliminate your 3D printed models' layer lines in preparation for painting, silicone molding, or vacuum forming. Consider using thinner layer lines when printing, even if it increases the print time. In exchange, it will save you a lot of time sanding the part, no matter which tool you use.
- **Priming & Painting**: If you 3D print your model with a high layer height, you'll need to apply multiple paint coats to get a smooth surface. In these cases, using a low layer height will save you time as you won't need to repeat the prime - dry - paint - dry process over and over again.

# Layer height and mechanical properties

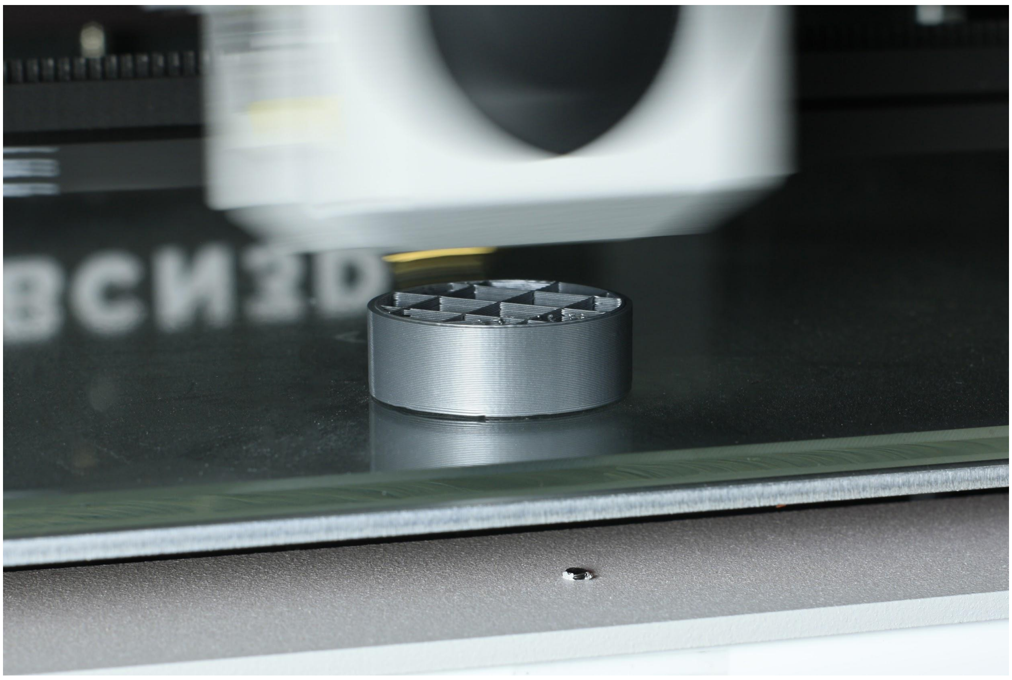
The size of the hotend has direct effects, not only on the printing speed but also on the strength of the 3D printed part. That's why choosing the right hotend size for each application is essential to manufacture high-quality parts within the shortest possible time.



Test part

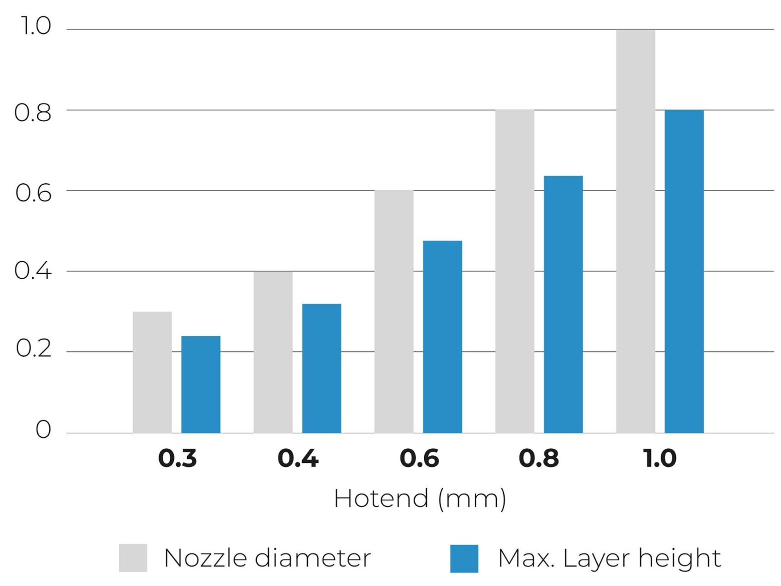
As you can see in the graph above, the combination of larger hotends allows thicker layers, which leads to stronger prints when using similar print settings.



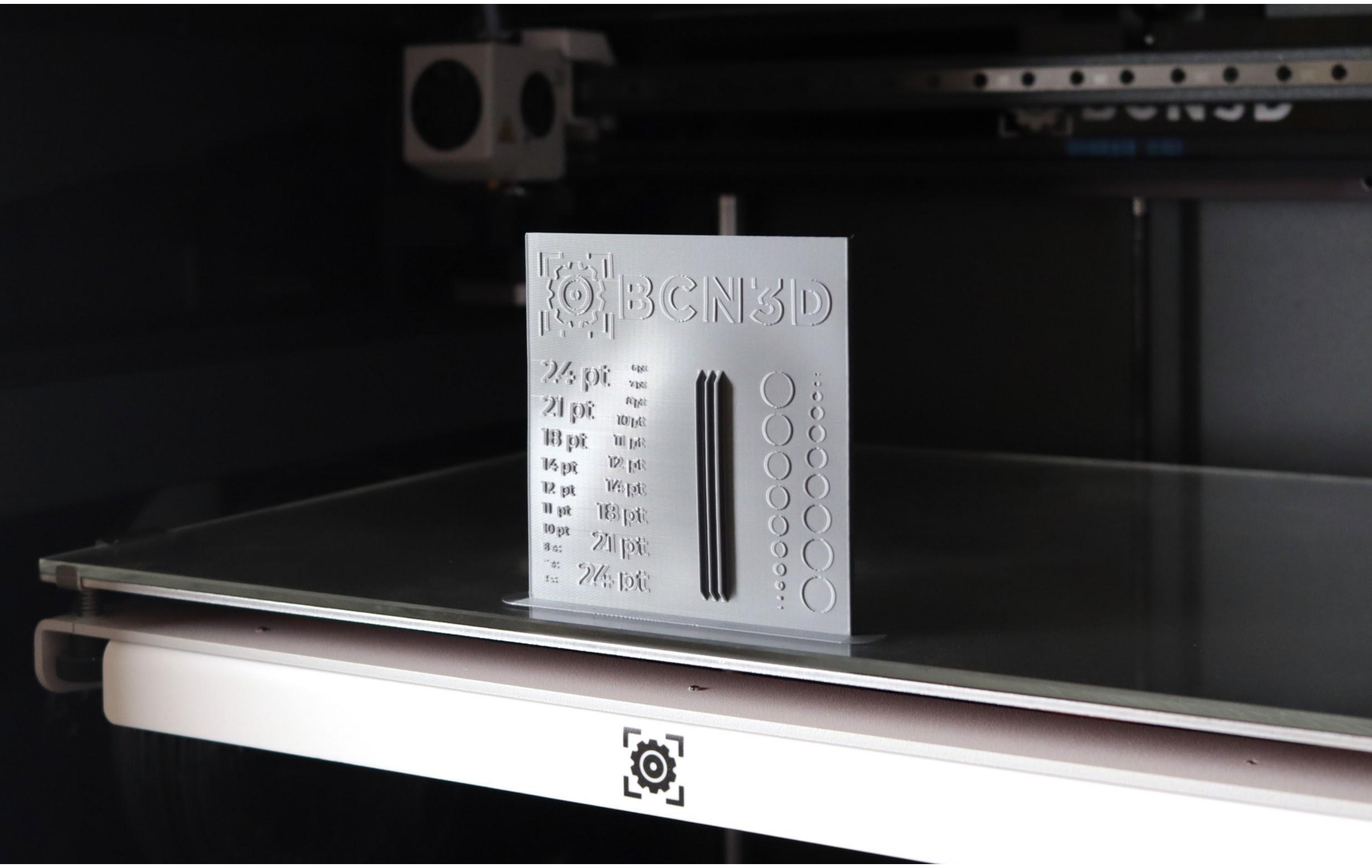
# Slicing considerations

Different hotends are available for the Epsilon and Sigma product families. Each hotend has a different nozzle diameter, from 0.4mm up to 1mm.

The layer height can usually be up to 80% of the nozzle diameter to guarantee dimensional accuracy in the printed parts. This means that the layer height shouldn't be larger than 0.32mm when using a 0.4mm nozzle (standard). With a 0.6mm nozzle, the maximum layer height is increased to 0.48mm.



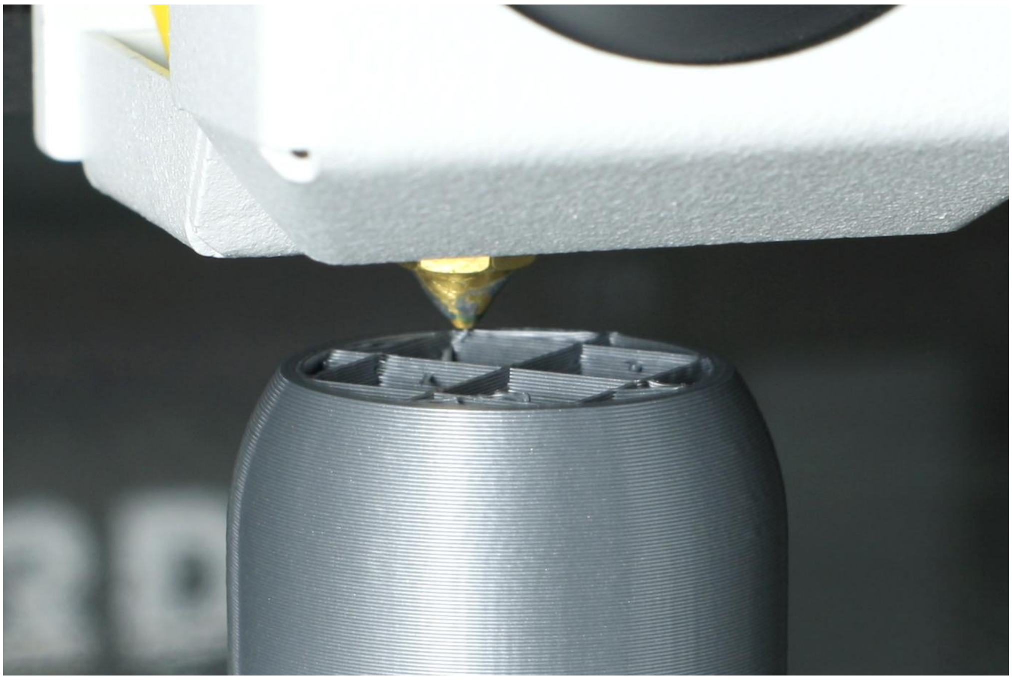
# 2. Line Width



# What is Line Width?

The Line Width setting represents how wide the filament lines are extruded.

This setting is usually the same as the nozzle diameter. However, it's possible to extrude thinner and thicker lines by playing with this setting. For example, if you're using a standard 0.4mm nozzle, you could set a line width of 0.3mm or 0.5mm.



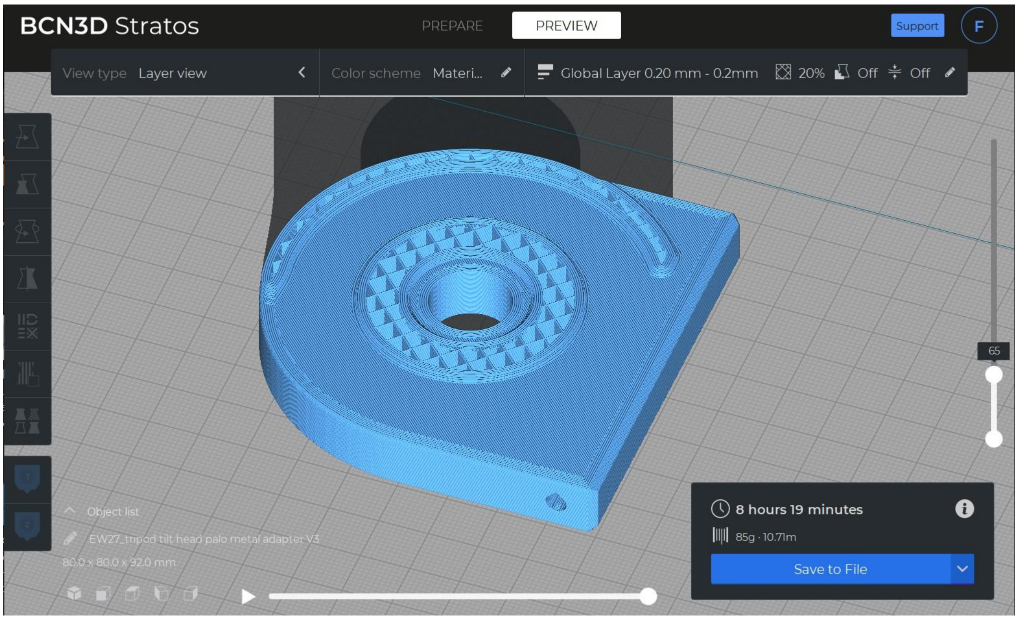
This setting has many effects on the 3D printing process, including:

- **Surface details:** You can improve the surface details' quality, especially on text and geometric shapes.
- **Bed adhesion:** An increased line width multiplier on the first layer improves bed adhesion.
- **Mechanical properties:** A larger infill line width can increase the part strength while

# Line Width options

In BCN3D Stratos, you can individually set the line width of each type of line, including:

- **Line width:** Width of a single line.
    - **Wall Line Width:** Width of a single wall line.
        - **Outer Wall Line Width:** Width of the outermost wall line. By lowering this value, higher levels of detail can be printed.
        - **Inner Wall(s) Line Width:** Width of a single wall line for all wall lines except the outermost one.
    - **Top/Bottom Line Width:** Width of a single top/bottom line.
    - **Infill Line Width:** Width of a single infill line.
    - **Skirt/Brim Line Width:** Width of a single skirt or brim line.
    - **Support Line Width:** Width of a single support structure line,
    - **Initial Layer Line Width:** Multiplier of the line width on the first layer. A larger initial layer line width could improve bed adhesion.




Layer View on BCN3D Stratos

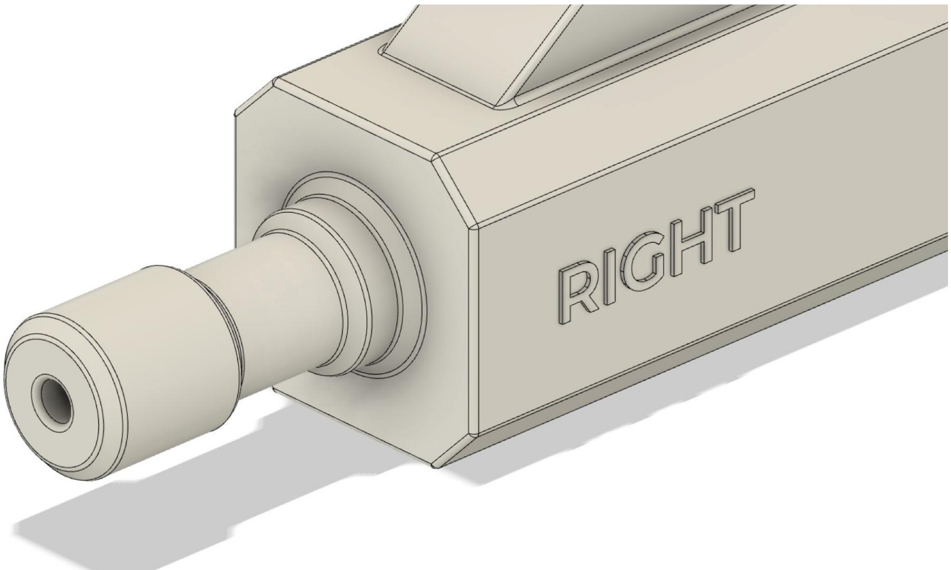
If you don't see all these settings in your BCN3D Stratos, you can activate them with the Setting Visibility menu you can find on Preferences > Configure BCN3D Stratos > Settings.

# Line Width applications

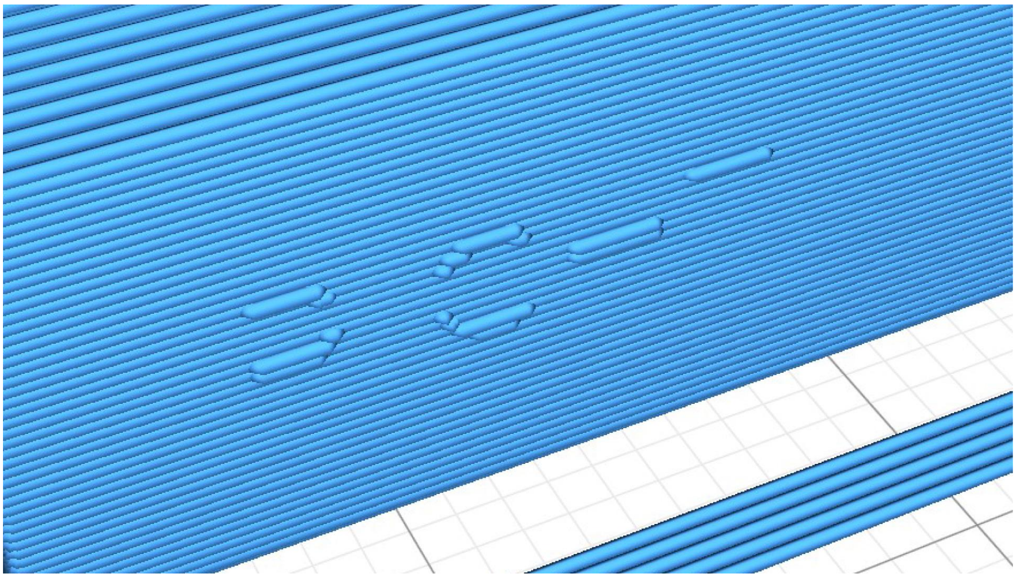
## Surface details

When your 3D model includes surface details that are smaller than the hotend size, BCN3D Stratos ignores them. However, if you slightly reduce the **Outer Wall Line Width**, you may print those features successfully.

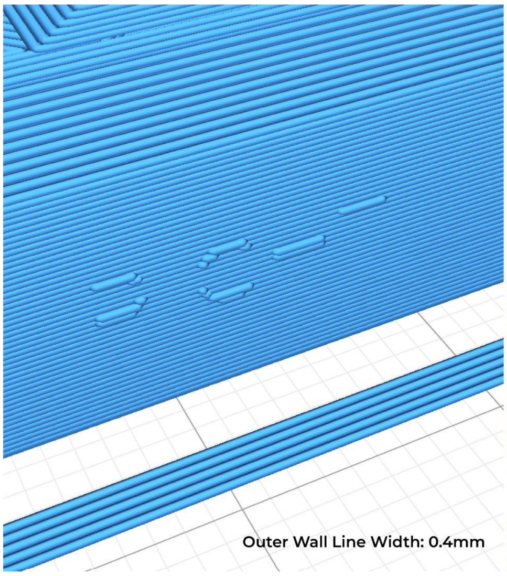
For example, many 3D models include text that offers indications when using the product. The product from the image below has two almost identical components. One must go on the right and the other on the left.

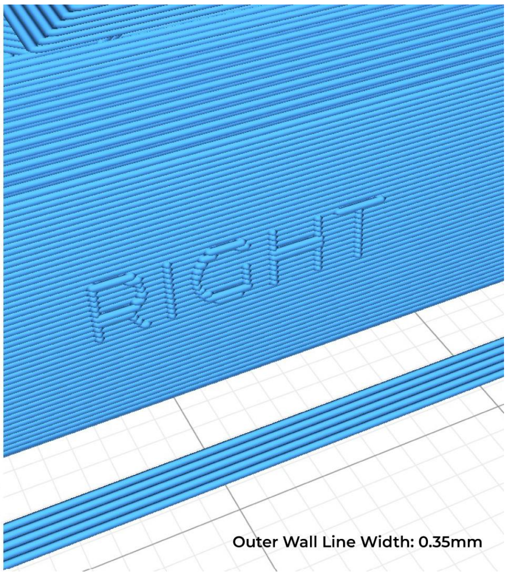



The design is optimized for 3D printing, and the text (*RIGHT*) uses a sans serif font that increases readability. It's also extruded 0.4mm, which corresponds to the hotend size. However, when slicing the model in BCN3D Stratos, the text disappears as the stroke thickness isn't large enough.


Layer View of test with 0.4mm Line Width on BCN3D Stratos

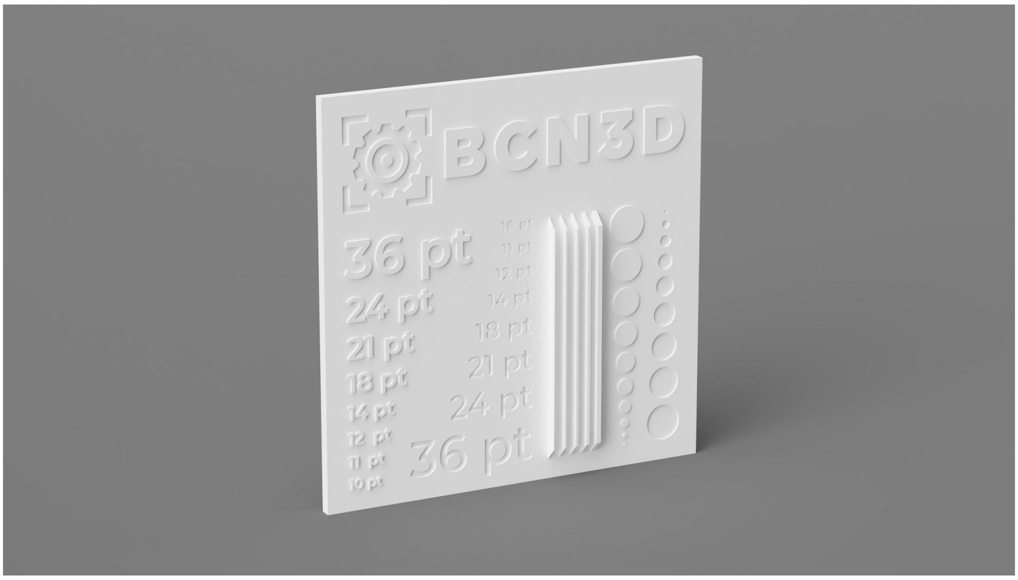
This is a prevalent issue with text features, as the stroke thickness should always be larger than the Line Width. In this case, the problem is solved by slightly reducing the Outer Wall Line Width from 0.4mm to 0.35mm.



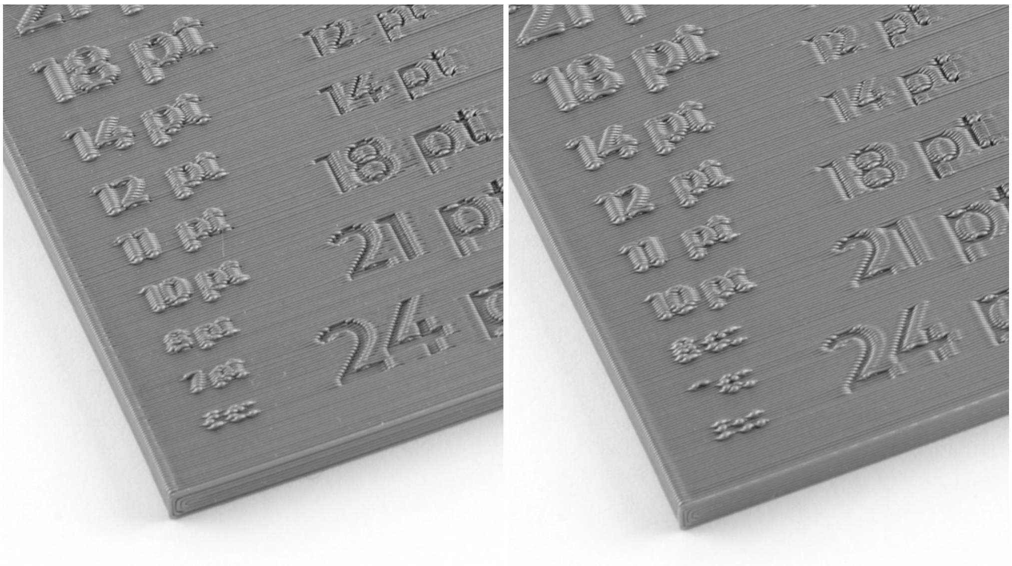



Knowing the smallest feature size and designing considering it is essential when 3D printing. It will help you have everything under control. Yes, even the smallest details.

If your models usually include intricate details, you can design and 3D print a custom-made test part to identify the feature sizes that work best for you. Print one version with the standard Line Width and another one with an adjusted Outer Wall Line Width.



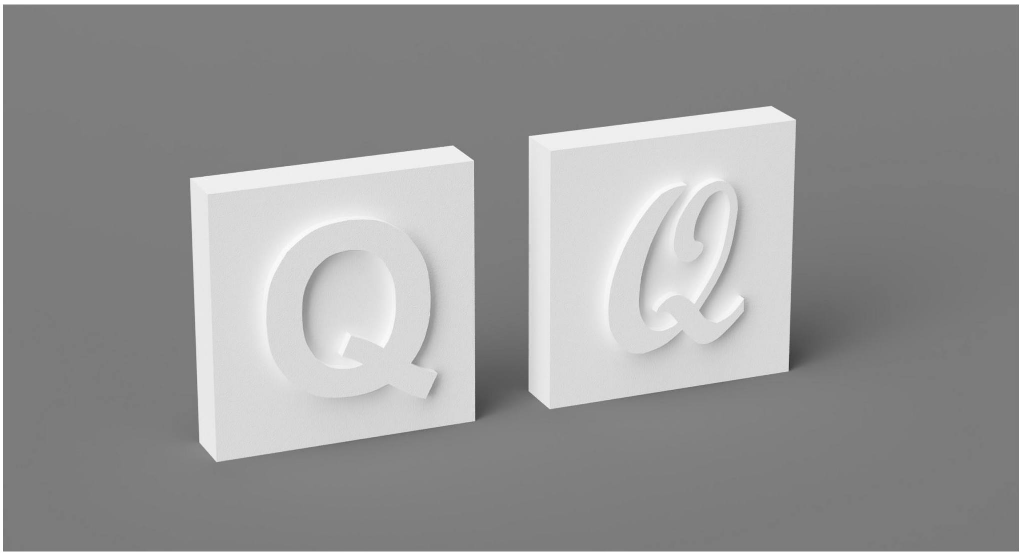

Using test models to identify the best feature size for your designs will help you improve your design skills and understand how to make the best out of your 3D printer.



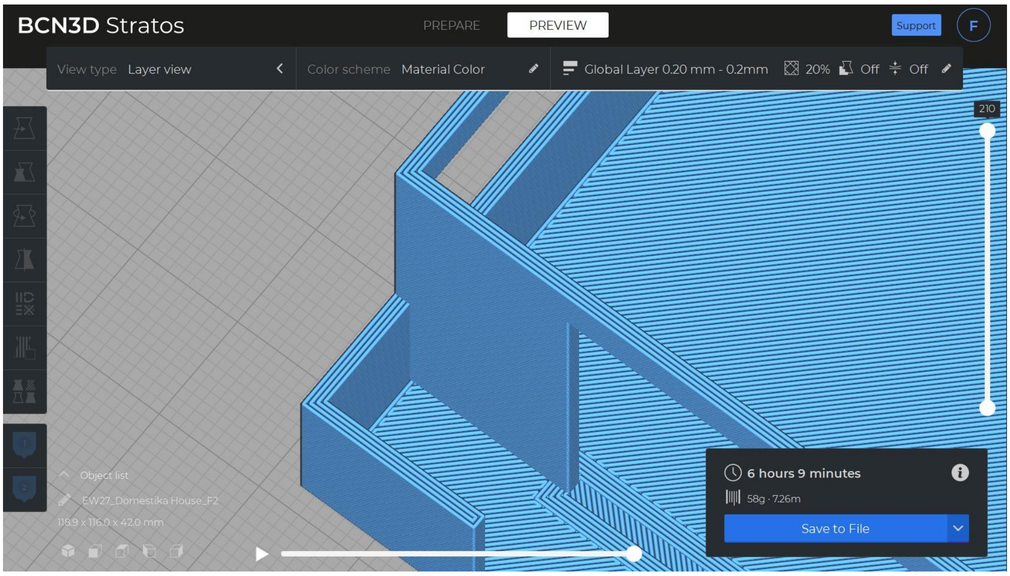

Outer Wall Line Width test comparison: 0.35mm (left), 0.4mm (right)

In the example above you can see how small text sizes such as 7 pt are only printed when the Outer Wall Line Width is reduced to 0.35mm.

If your 3D models include text, use sans serif fonts, bold text, and large font sizes. Handwritten fonts usually require a large font size to compensate for the inconsistent stroke width.



The Line Width settings are also helpful in those cases where complex shapes and textures need to be printed as one single part. For example, architectural models usually include vertical walls that have a variable thickness. Some of them may be too thin to be 3D printed. In those cases, using a smaller line width can help achieve better overall quality.

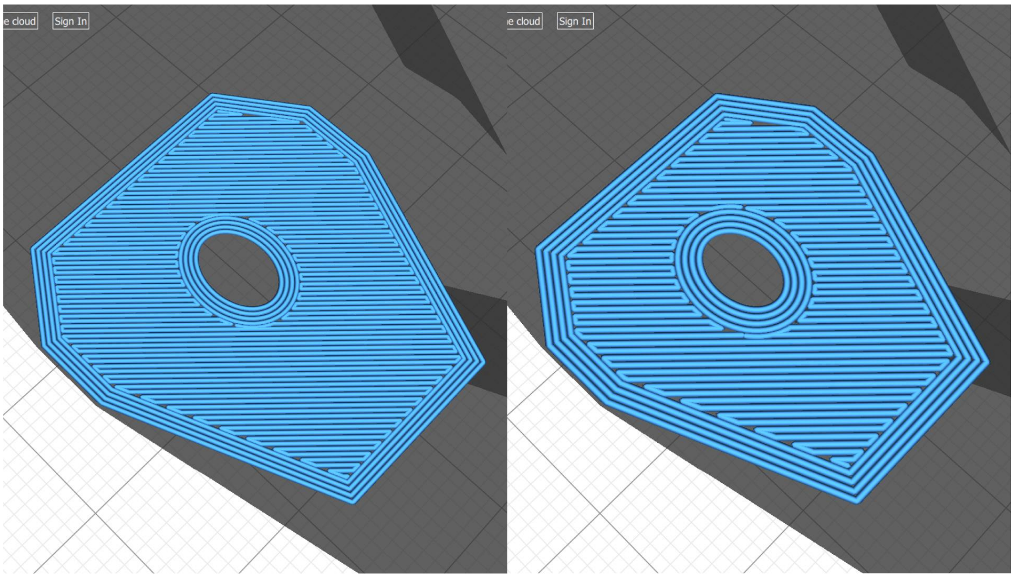


Layer View of an architectural model on BCN3D Stratos

## Bed adhesion

There are many ways to guarantee excellent bed adhesion, including using the right bed temperature, leveling the build plate, or using glue. However, there's one more way to improve bed adhesion: **The Initial Layer Line Width**.

This setting increases the first layer's line width, affecting all the line widths, including walls, supports, infill and skin. All these settings are multiplied by the Initial Layer Line Width.




Initial Layer Line Width comparison: 100% (left), 130% (right)

As you can see in the example above, when the Initial Layer Line Width is increased, all the extruded lines thicken, but the part size remains the same. To avoid over extrusion, when you increase the Initial Layer Line Width, the distance between lines is also increased.

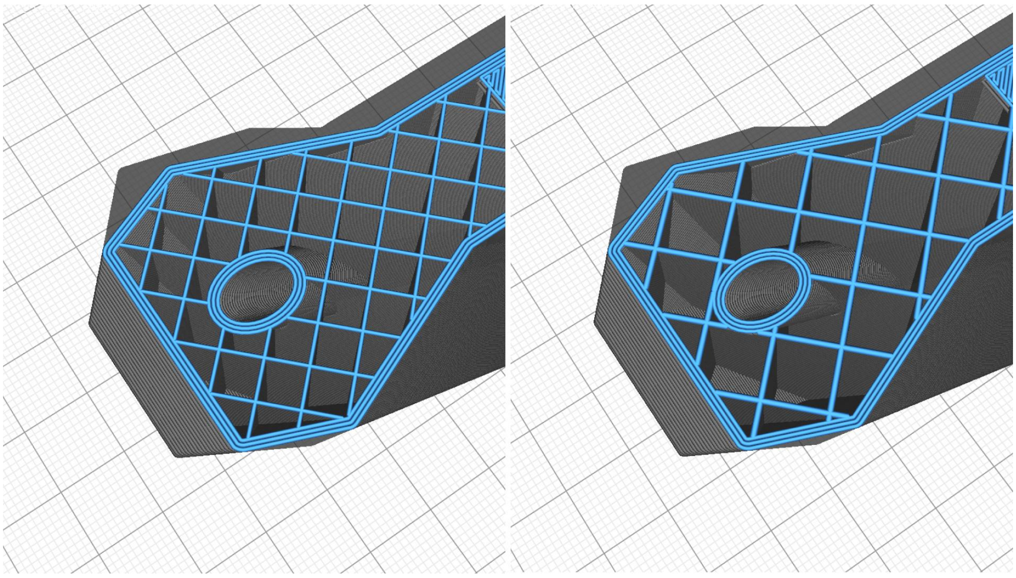
But, how does this improve bed adhesion? As more material is extruded in the first layer, it's pressed harder against the build plate, increasing the adhesion between the printed part and the bed.

## Mechanical properties

If you want to print a stronger part fast, the Line Width is a setting you'll want to consider.

Make slight adjustments to the part strength by increasing the **Infill Line Width** from 0.4mm to 0.6mm instead of increasing the infill percentage. This way, your part's infill will be thicker and more resistant while the print time remains the same.

In the following example, the Infill Line Width was increased from 0.4mm (standard Line Width) to 0.6mm while using a Grid infill pattern.



Infill Line Width comparison: 0.4mm (left), 0.6mm (right)

The number of infill lines is automatically reduced to guarantee an accurate infill percentage. If the infill grid had been identical, the additional material that would be extruded would increase the infill percentage, not by adding more lines but by simply adding more material on the inner part of the model.

The two main benefits when increasing the Infill Line Width are:

- **Reduced print time:** When the Infill Line Width is increased, the number of infill lines reduces, which also reduces the print time. In this example, the print time went from 273 minutes to 243 minutes—an 11% print time reduction.

- **Increased strength:** Using larger hotends leads to an increase in part strength. However, by slightly changing the Line Width, you can get similar results using a smaller hotend.

# Slicing considerations

Even though it's possible to improve the print quality by modifying the Line Width settings, it's recommended to find the right hotend for your print. Different hotends are available for the Epsilon and Sigma product families. Each hotend has a different nozzle diameter, from 0.4mm up to 1mm.


The Line Width can be increased or reduced by up to 20% of the nozzle diameter to guarantee dimensional accuracy. This means that the Line Width could go down to 0.32mm for small surface details and up to 0,5mm for increased strength, all using a standard 0.4mm nozzle.

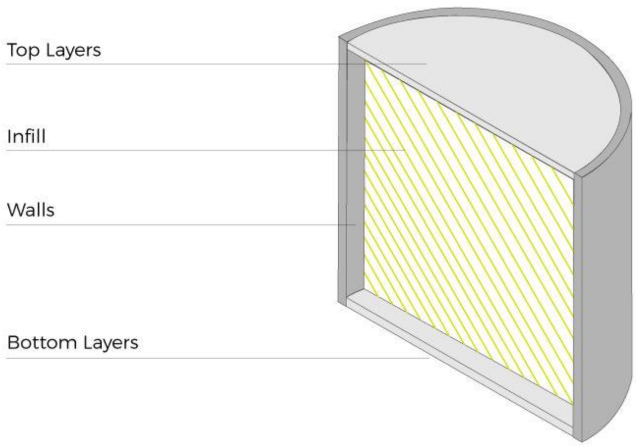
# 3. Wall Thickness

# What is Wall Thickness?

The Wall Thickness setting represents the thickness of your model's walls in the horizontal direction. The Wall Thickness divided by the Wall Line Width defines the number of walls.

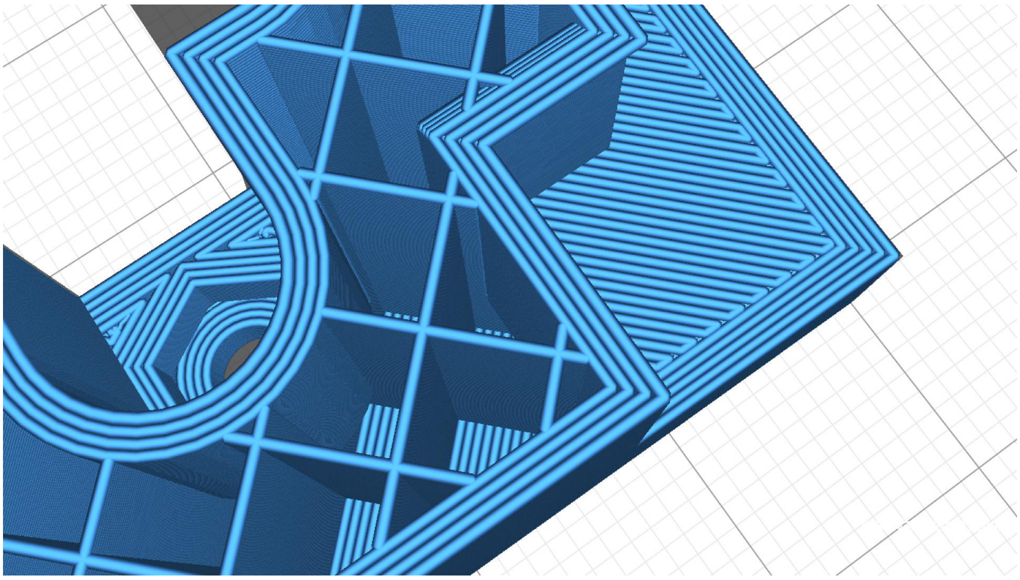
The Wall Thickness is used in combination with the Top/Bottom Thickness, which allows you to control the thickness of the wall in the vertical direction.

You can see the difference between Wall Thickness, Top/Bottom Thickness, and Infill in the image below.




It's common to use the same thickness for your whole model, whether it's the bottom, top, or vertical walls. However, you can individually set the thickness of each one of them.

For example, you could 3D print a model with three **different wall thicknesses**. Instead of using a 1.2mm wall thickness on the three types of walls, you could use a thicker top thickness to prevent issues like pillowing, while the bottom thickness is thinner.

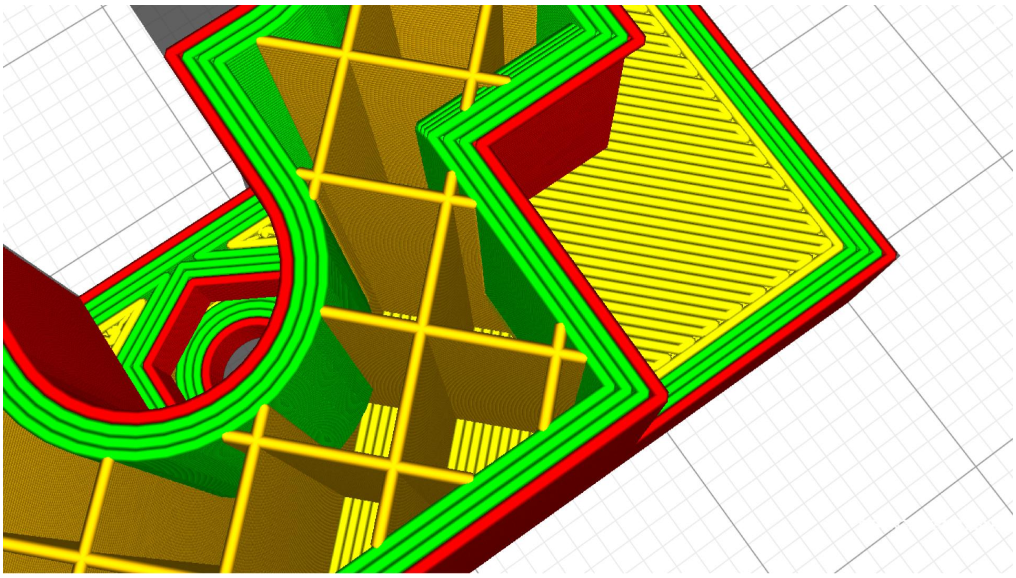


BCN3D Stratos Wall Thickness preview

Notice that the vertical walls of your model are usually divided into two categories:

- **Outer Wall:** The outermost wall line.
- **Inner Walls:** All walls except for the Outer Wall.

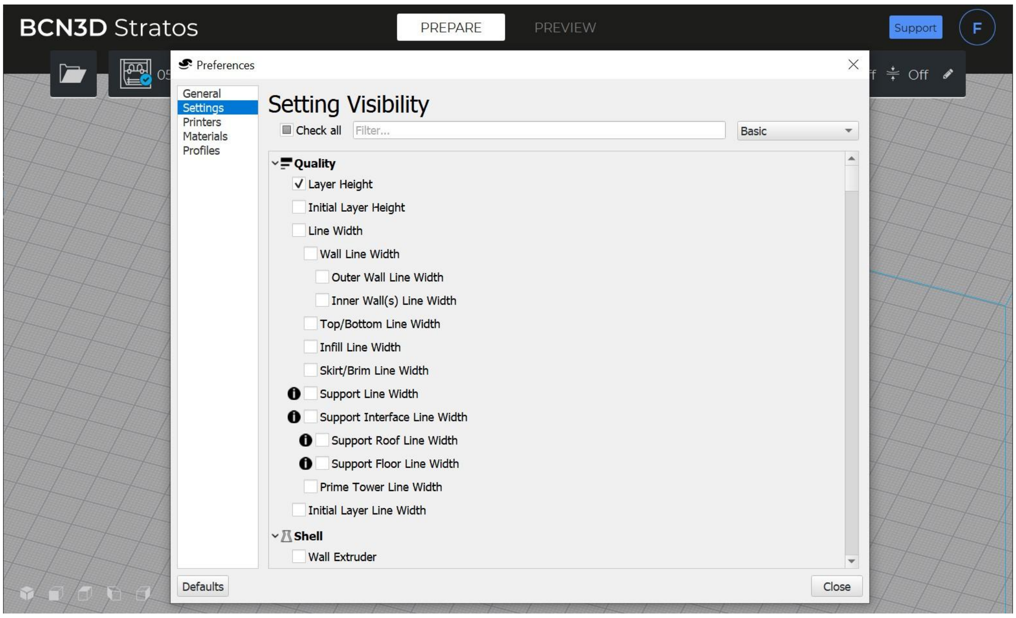
The reason behind this difference is that you may want to print the Outer Wall at a lower speed to increase the surface finish, as it's the only wall you usually see and touch.



BCN3D Stratos Wall Thickness preview: Outer Wall (red), Inner Walls (green)

### Where to find the Wall Thickness settings?

Consider that the Wall Thickness settings are also known as shell thickness, perimeters, or outer shell. In BCN3D Stratos, the Wall Thickness settings can be found inside the **Shell tab**.




# Wall Thickness Settings

In BCN3D Stratos, you'll find different Wall Thickness settings in the Shell tab. These are the most important ones:

- **Wall Thickness:** The thickness of the walls in the horizontal direction. This value divided by the wall line width defines the number of walls.
    - **Wall Line Count:** The number of walls. When calculated by the wall thickness, this value is rounded to a whole number.
- **Top/Bottom Thickness:** The thickness of the top/bottom layers in a printed part. This value divided by the layer height defines the number of top/bottom layers.
    - **Top Thickness**
        - **Top Layers**
    - **Bottom Thickness**
        - **Bottom Layers**
    - **Top/Bottom Pattern:** The pattern of the top/bottom layers.

As you can see, the wall thickness is usually represented by its thickness (mm) or by the resulting number of lines or layers.

## Calculating the Wall Line Count

It's highly recommended to set a Wall Thickness that is a **multiple of your nozzle** diameter.

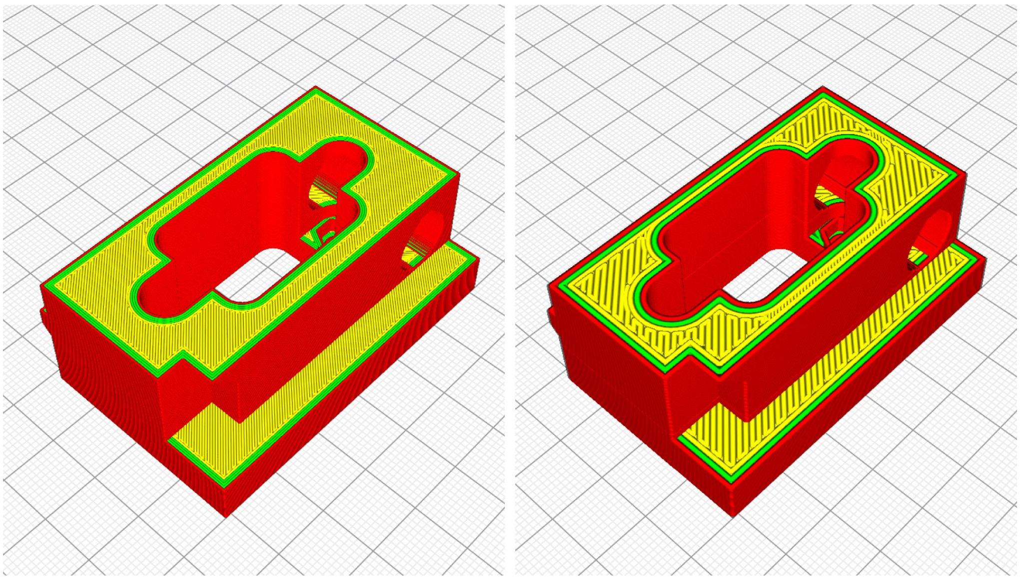
If you're using a standard 0.4mm nozzle, you'll want to set a Wall Thickness of 1.2mm, as it corresponds to exactly 3 Wall Lines. If you set a Wall Thickness of 1mm, it will correspond to 2.5 Wall Lines, resulting in inconsistent print quality.

The more you control the Wall Thickness, the easier it will be to guarantee a specific print quality.

## Nozzle size and Wall Thickness

As mentioned above, it's always recommended to set a Wall Thickness based on the nozzle diameter.

If you want to 3D print a model with 2mm walls, you could either use a 0.4mm nozzle and print five walls or use a 1mm nozzle and print just two walls.



BCN3D Stratos Wall Thickness vs. Nozzle Diameter comparison: 0.4mm (left), 1mm (right)

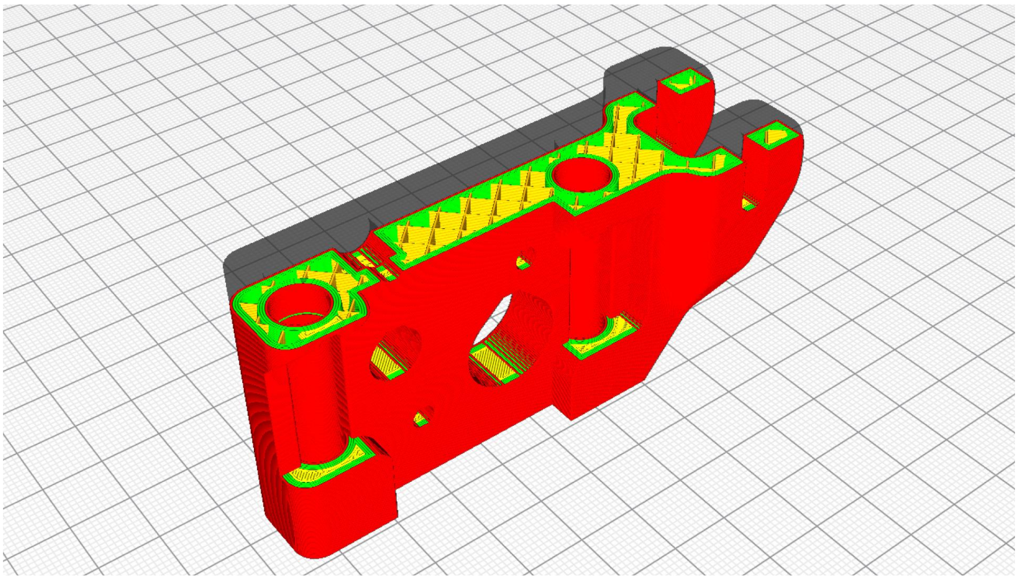
Using larger nozzles results in **reduced print time**, and in some cases, it can increase the overall part strength.

It's important to consider that the nozzle diameter affects surface finish as it rounds sharp corners, and it may **reduce the print resolution** in those cases where small details such as text are included.

# How to get the perfect wall thickness

Finding the perfect Wall Thickness for your model depends entirely on the mechanical properties and surface finish you want to get. Before you slice your 3D model, you need to **understand the part's purpose** and desired functionality.

For example, the walls of a decorative part will be thin because they don't need to be thick to offer a clean surface finish. However, the Wall Thickness of a model that needs to support the weight of a vehicle will be wide as you need to make it as strong as you can.



The examples you can find in this whitepaper will show you when to increase or reduce the Wall Thickness, but testing different prototypes is still the best way to understand Wall Thickness.

# How Wall Thickness affects printed parts

Wall Thickness plays a crucial role in print quality and part strength, but finding the correct settings comes down to the part's purpose.

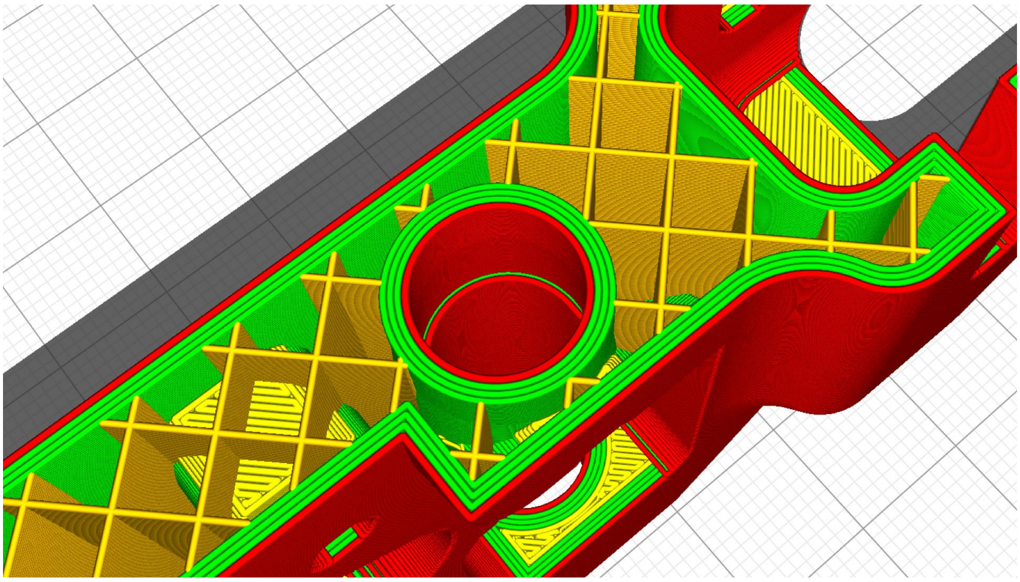
## Mechanical properties

If you need a strong part, increase the Wall Thickness. It's as easy as that.

No matter their purpose, all parts should be printed with a **0.8mm wall thickness or at least twice the nozzle diameter**. The reason behind this number is that parts that are printed with a single wall can deform, affecting part strength and dimensional accuracy.

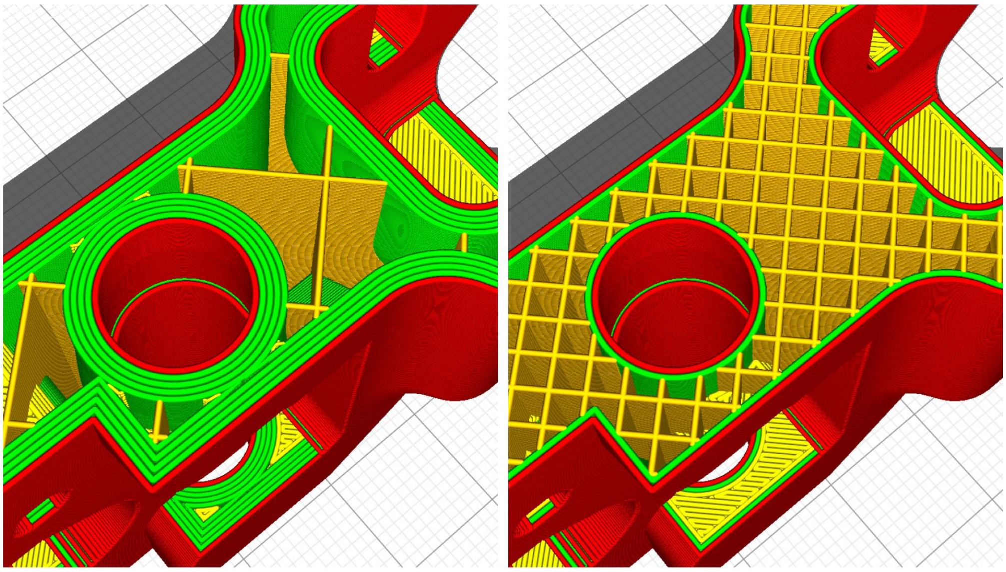

Test prints to find the optimal Wall Thickness

If you want to **increase part strength**, it's recommended to use a Wall Thickness of **at least 1.6mm** (4 walls with 0.4mm nozzle), although you can increase that number depending on the part strength required and the nozzle diameter.



When you want to improve the mechanical properties of your part, you'll usually increase the **Wall Thickness and the Infill percentage**. Notice that it's essential to find the right balance between these two settings.

In some cases, you'll want to use thick walls (2mm) while keeping the infill percentage low (10%), but in other cases, you may prefer to use thinner walls (0.8mm) and a high infill percentage (40%).

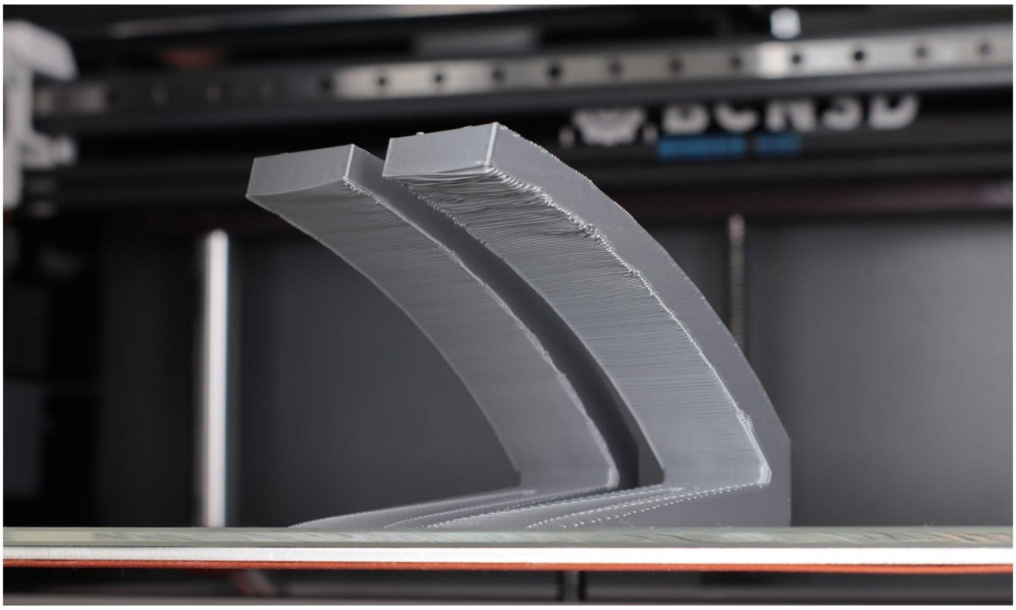


BCN3D Stratos Wall Thickness & Infill 2mm walls + 10% infill (left), 0.8mm wall + 40% infill (right)

## Surface finish

An increased Wall Thickness has a positive impact on overhangs and post-processing.

Models that include overhangs tend to have a low surface quality due to the small overlap between layers. When you increase the Wall Thickness of a model with **overhangs**, you're increasing the overlap between layers, leading to improved surface quality.



Overhang test

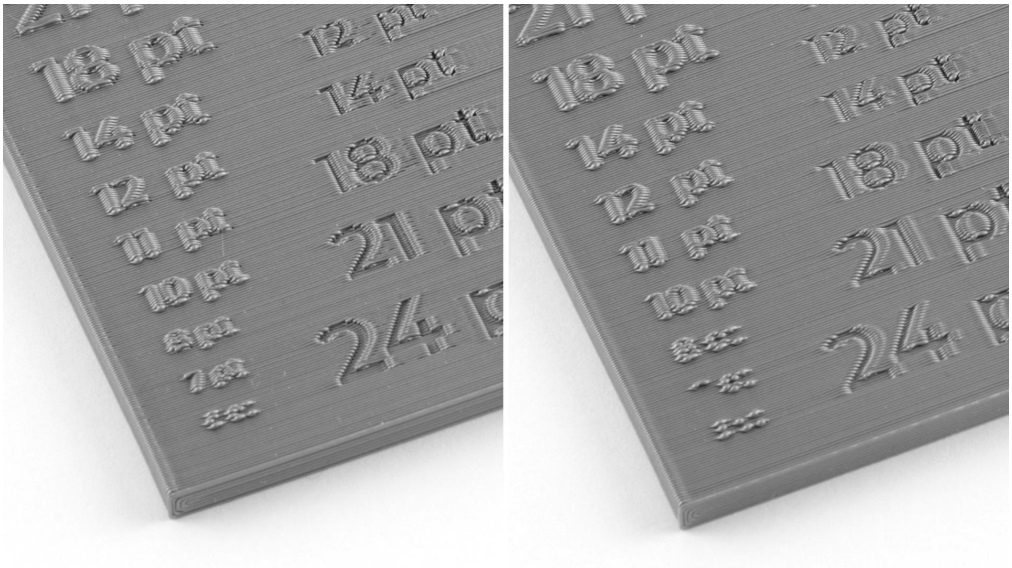
If you want to **post-process** or sand your 3D printed part, it's recommended to increase the Wall Thickness, as sanding usually removes the outermost wall.

This means that if the printed part presents some surface quality issues, you'll have the chance to sand off more materials without compromising the part's integrity.

# Design considerations

To make the best out of your 3D printer, you should consider the best practices when Designing for Additive Manufacturing (DFAM).

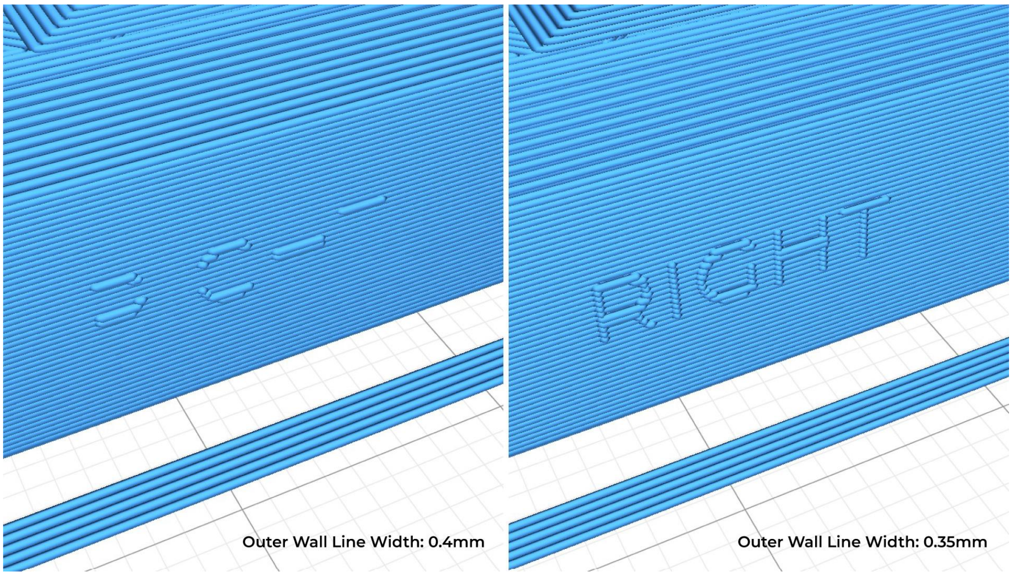
This means that the design you print should be optimized for the manufacturing technology, and the model's feature size is heavily affected by the Wall Thickness settings.




Wall Line Width comparison print test: 0.35mm Line Width (left), 0.4mm Line Width (right)

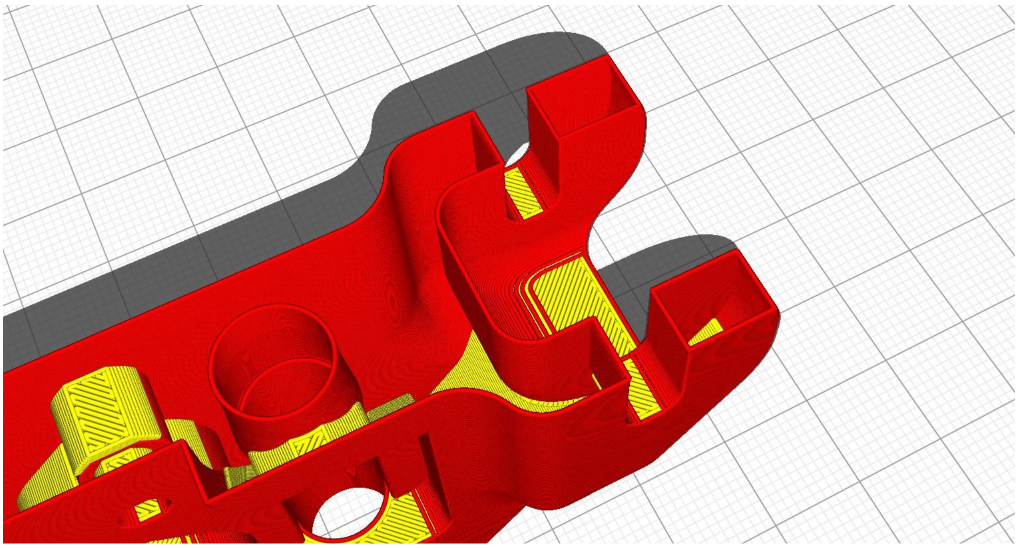
Make sure your design's features are thicker than the minimum wall thickness. It's recommended to have a **feature thickness of at least twice the nozzle diameter** (0,8mm for 0.4mm nozzles).

If the feature size is smaller than the Wall Thickness or the Line Width, BCN3D Stratos will ignore them. That's why using the **Review mode in BCN3D Stratos** is so relevant. That way, you can identify design issues before you print the model.




BCN3D Stratos Preview Mode

Also, consider that the parts are manufactured one layer at a time. This means that if the **Wall Thickness is too small** or a feature too thin, the part may bend or deform on its edges before it cools down. As a result, the following layers would be deformed, ruining the printed model.



BCN3D Stratos Preview Mode: Test model with no infill and one wall line

Related to thin walls and sharp corners, thermal stress accumulates on the edges of the part. This means that when the extruded material cools down, it contracts. This can cause **warping**, which can be easily identified because the corners of the print curl upwards.

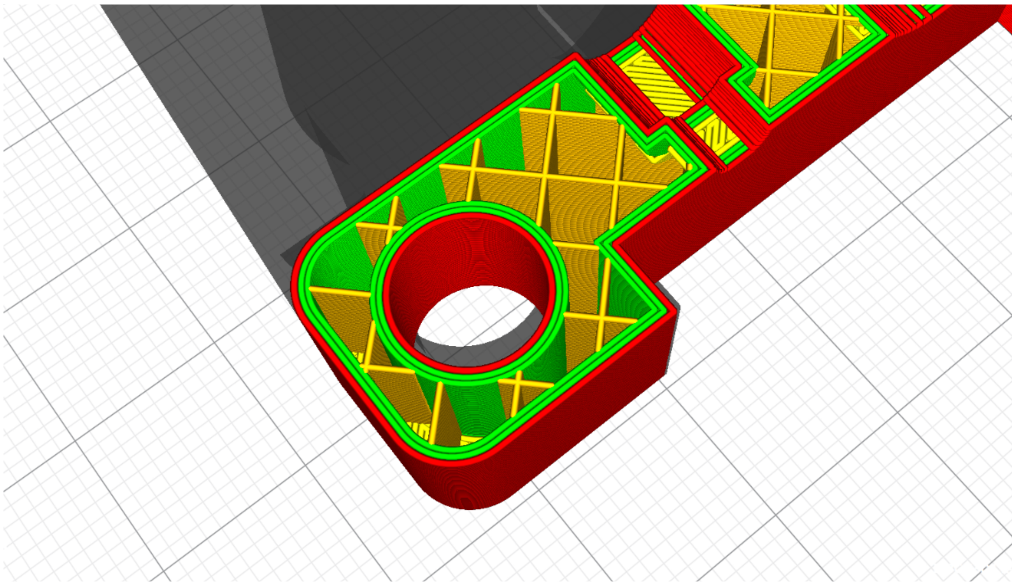
The best way to solve this issue is by **rounding the corners**, as it distributes the stress through the layer. Also, consider that thicker walls may increase the thermal stress of the part, increasing the chances of warping.

# Slicing considerations

## Recommended Wall Thickness settings

The default Wall Thickness value in BCN3D Stratos is 1.2mm, which corresponds to 3 wall lines when using a standard 0.4mm nozzle.

Using **three wall lines** offers a good part strength without increasing print time and using too much material.



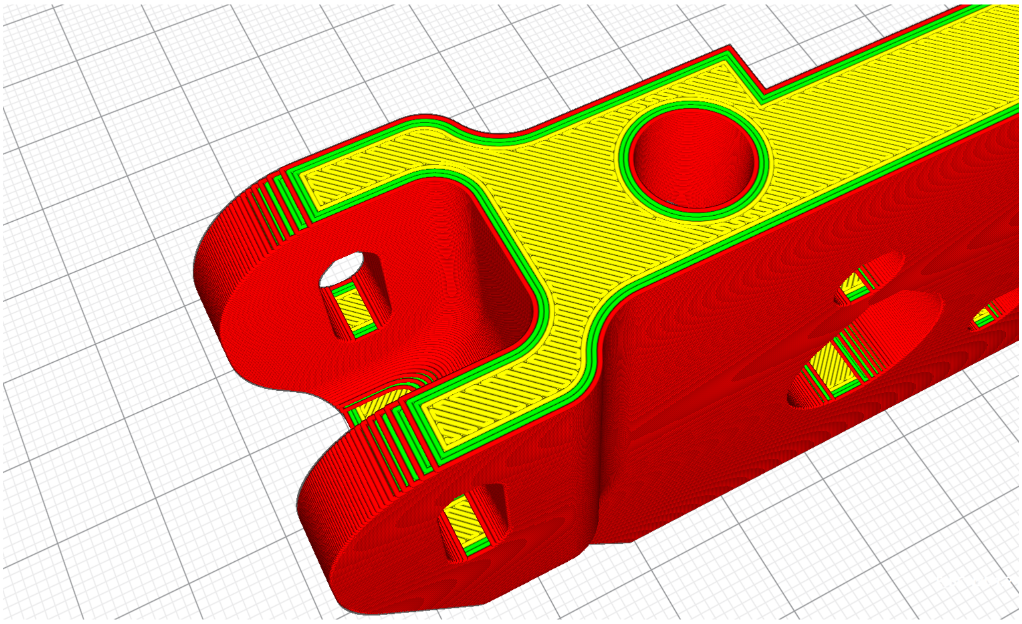
A thinner 0.8mm Wall Thickness can be used for prototypes where dimensional accuracy isn't essential, and 2mm Wall Thickness should be used for parts that require a higher strength.

Be aware that the Wall Thickness should always be at least twice the nozzle diameter.

## Recommended Top/Bottom Thickness

As it happens with the Wall Thickness, the recommended Top/Bottom Thickness is around **1mm-1.5mm** to guarantee similar print results.

If the printed part has a low Infill percentage, there may be pillowing issues on the top of the print. To prevent this, the **Top Thickness can be increased** to up to 2mm, as the additional Top Layers will even the top surface.



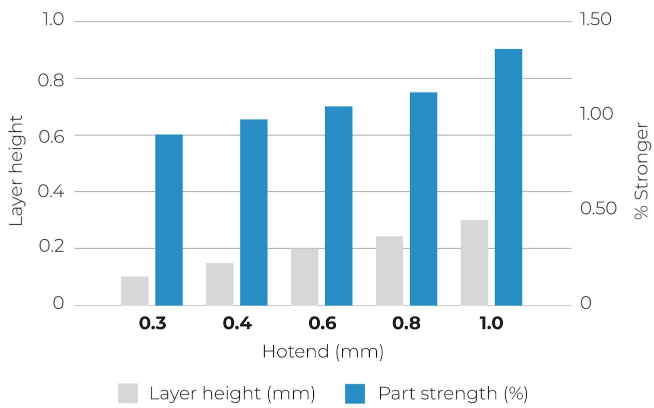
## Common Wall Thickness issues

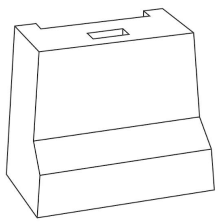
- **The walls are too thick:**
  - Increased material use
  - Increased print time
  - Reduced flexibility of the part (if desired)
  - Increased chances of warping
- **The walls are too thin:**
  - Low structural integrity
  - Lower print success rate
  - Post-processing not possible

# Wall Thickness and Hotend Family

Different hotends are available for the Epsilon and Sigma product families. Each hotend has a different nozzle diameter, from 0.4mm up to 1mm.

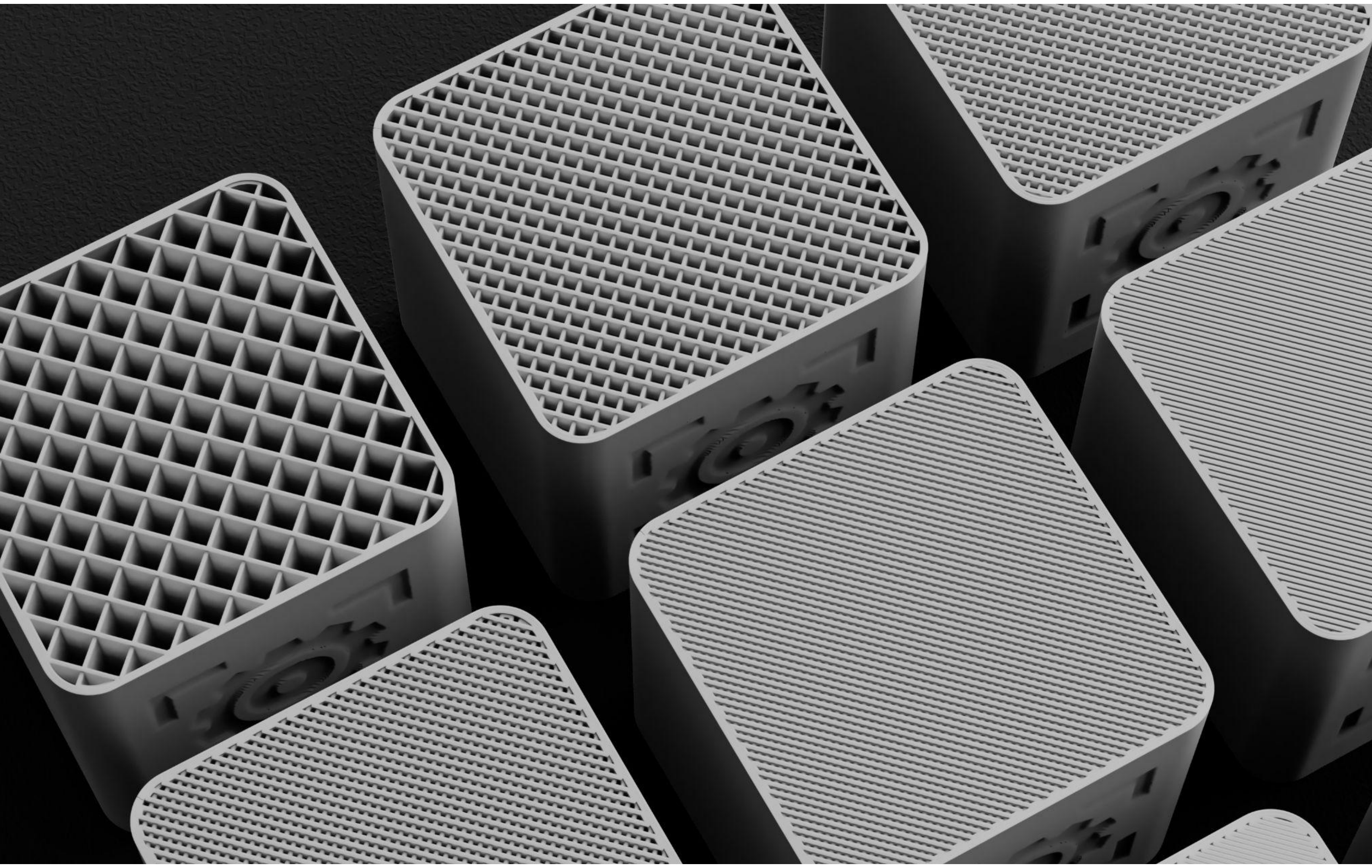

The use of larger nozzle sizes allows you to increase the part strength while you reduce the print time.



Test part

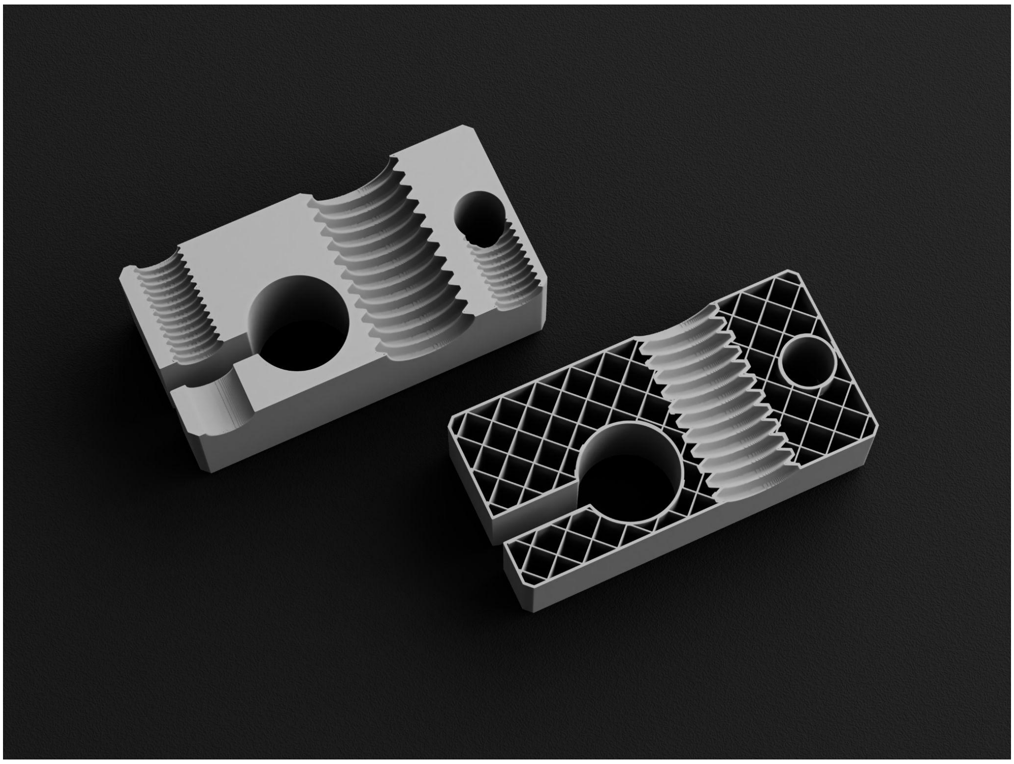
# 4. Infill



# What is Infill?

In the 3D printing industry, the infill refers to the **internal structure that fills the inside of a 3D printed part**. Components made with a 3D printer are rarely solid (100% infill). Instead, a low-density structure is 3D printed, saving manufacturing time and material while offering similar mechanical properties.

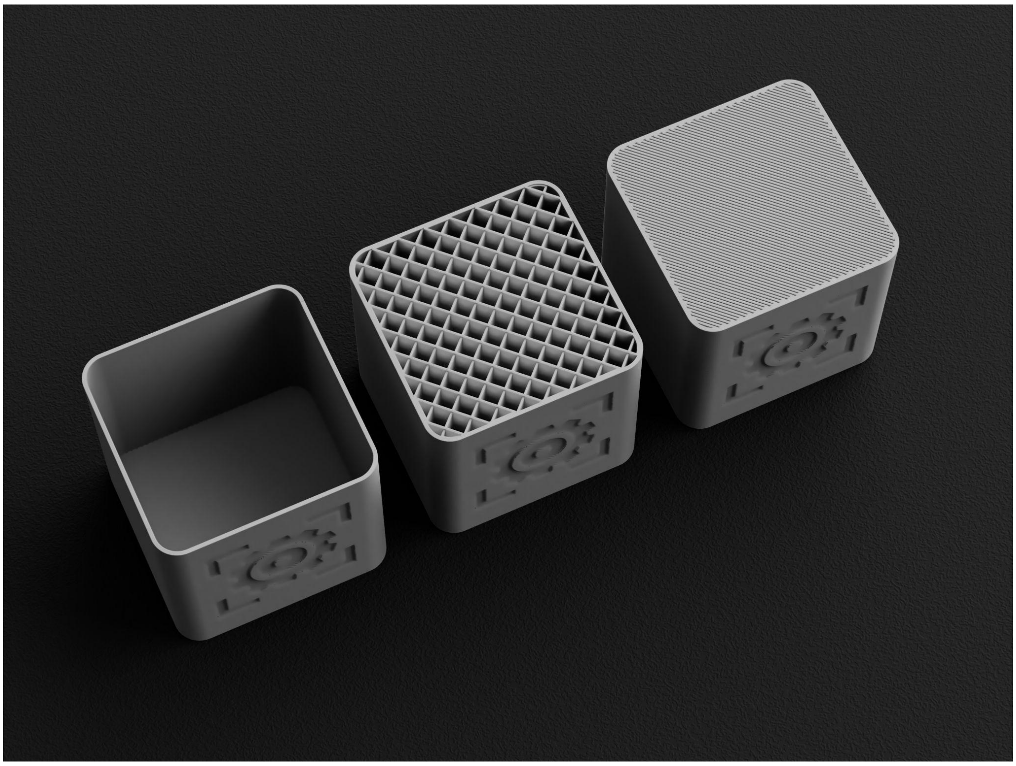
Infill, Layer Height, and Wall Thickness are **one of the most important slicing settings**. These setting groups are responsible for surface quality and part strength.



## Invisible but extremely efficient

Infill is one of the manufacturing features that makes 3D printing unique. The infill settings allow you to **decide the internal structure of the part**, including its density and properties.

In the image below, you can see how different 3D printed parts look. The model on the left is hollow, and it can be 3D printed fast. It's recommended for models with aesthetic purposes. In the middle, there's a part with an average amount of infill that offers infinite possibilities. You can find an almost solid model on the right, which isn't common and is mainly used to test materials.


Infill Density Comparison: 0% (left), 15% (middle), 60% (right)

Traditional manufacturing techniques such as CNC, laser cutting, or injection molding force you to create either solid parts or to manually optimize the internal structure to make it lightweight. That involves additional prototyping and resources.

With the infill settings, you can **modify the part's manufacturing process and its potential applications** in a couple of clicks.

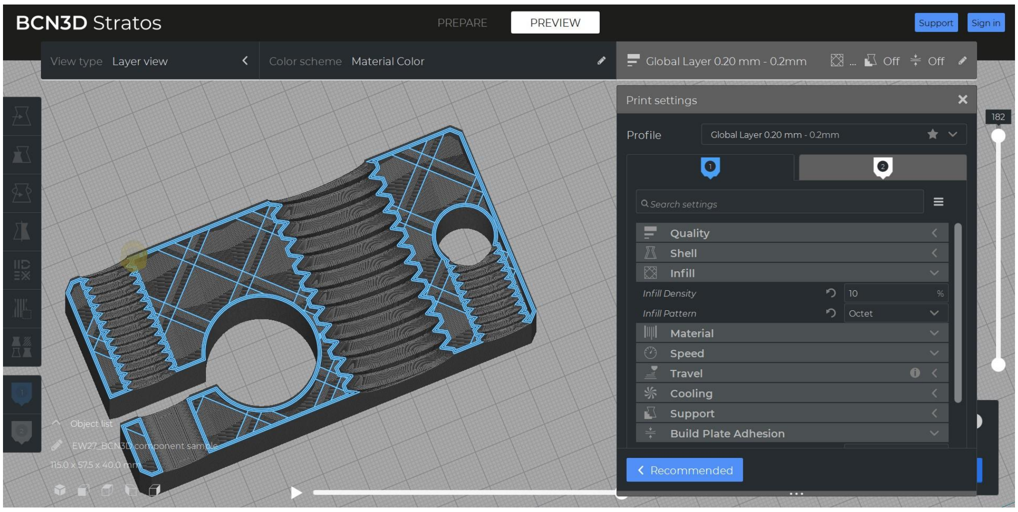
For example, if you're creating a part that must be installed in a race car, you need to **find the right balance between the part weight and mechanical properties**. By tuning the infill settings, you can easily create a hollow part that meets your exact requirements instead of making a solid metal version.

Also, consider that FDM 3D printing (filament) is an additive manufacturing process. This means that the manufacturing process **generates almost no waste**. On the other hand, subtractive manufacturing - including CNC and laser cutting - involve removing material from a large piece of stock, which always generates waste.

# Essential infill settings

Even though you can find dozens of Infill-related settings on BCN3D Stratos, two significantly affect the print quality: Infill Density and Infill Pattern.

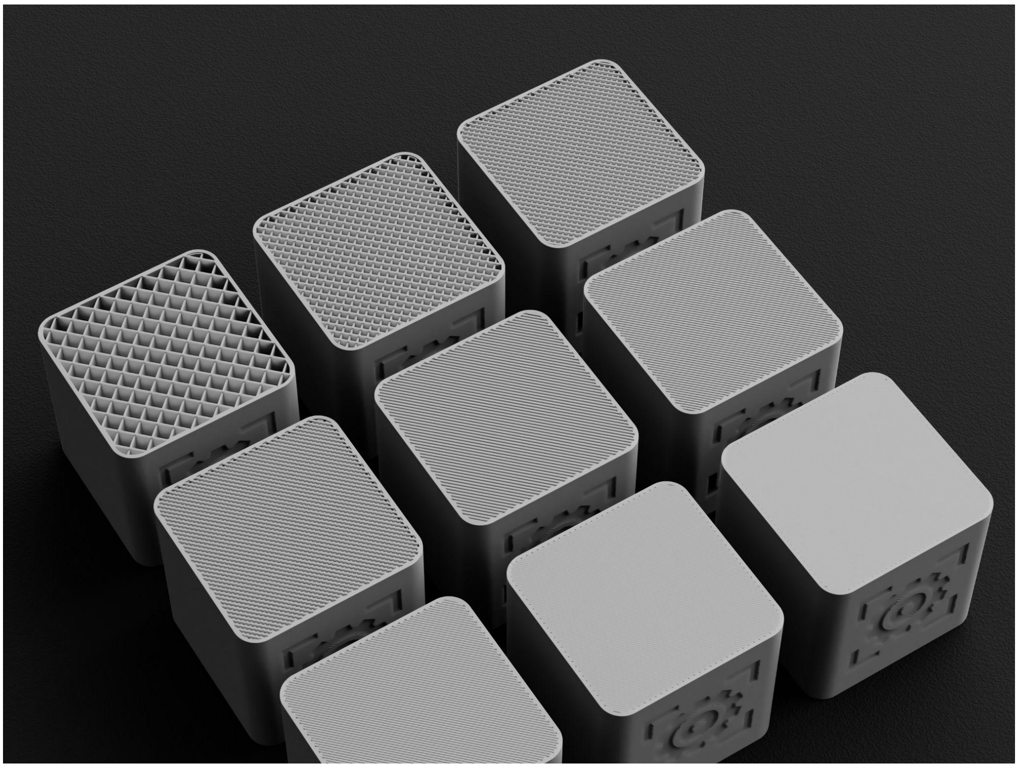
All Infill settings can be found in the Infill tab. The Infill Density and Infill Pattern options will be visible by default.




## Infill Density

This setting determines **how solid the internal structure should be**. It's defined as a percentage, being 0% hollow and 100% solid.

For example, if a part is 3D printed with a 20% Infill Density, 20% of the inside will be filled with plastic, and the other 80% will be empty.


Infill Density Comparison

**Recommended Infill Density settings**

It's essential to **understand the part application** to find the correct infill settings. Most initial prototypes are 3D printed with a low Infill Density, reducing print time and cost. However, final parts are usually made with a higher Infill Density to increase their strength.

- **Aesthetic (5-10%):** Used to make visual parts or early prototypes that don't need to support any loads.
- **Standard (10-20%):** Most common Infill Density. It offers a reduced print time and part weight.
- **Functional (20-30%):** Recommended for functional parts that will be subjected to moderate loads.
- **Advanced (30-50%):** Used to make strong parts that require the best possible mechanical properties.

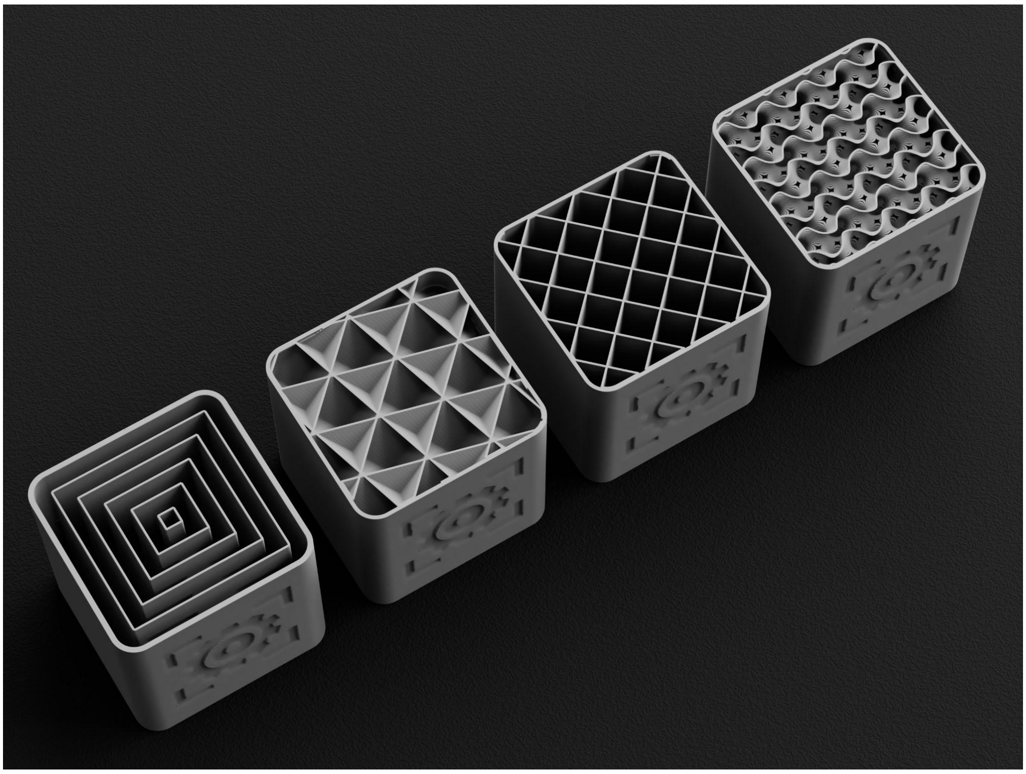
3D printed parts are rarely made using an Infill Density of over 50%. This is because the **optimal mechanical properties are usually achieved at 30-50%** Infill Density rather than 100%. However, testing different settings is always recommended.

Also, parts with 0% infill aren't common as a small Infill Density is always recommended to hold the walls together and improve the surface quality.

## Infill Pattern

The Infill Pattern refers to the **shape of the structure inside of the printed part.**

BCN3D Stratos offers more than ten types of Infill Pattern, ranging from simple lines to complex tridimensional shapes. Even though they all share a basic functionality, each one has unique features that make them appropriate for specific applications.



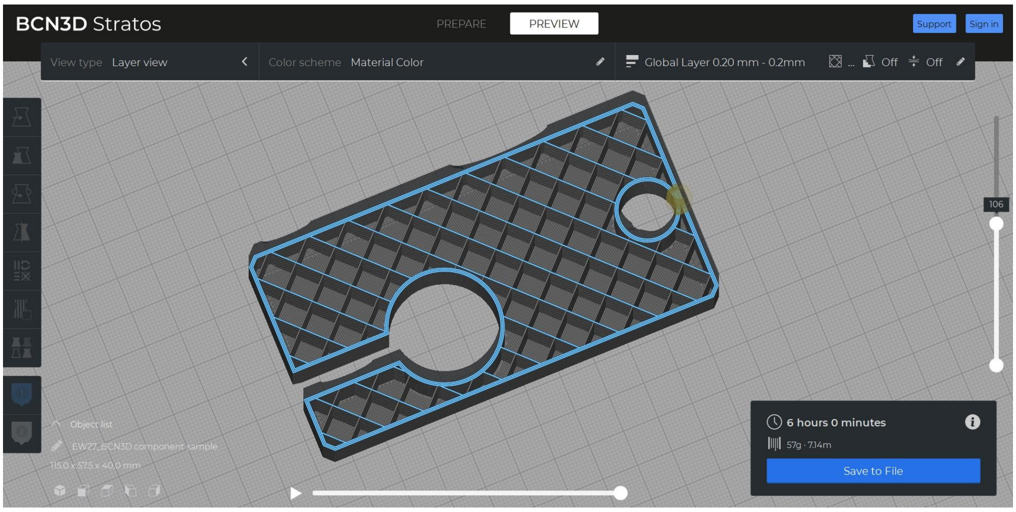
### Recommended Infill Pattern settings

Each pattern is unique, and you should test all of them to find the one that works best for you.

### Line pattern

The line pattern is the most basic type of infill you'll find. It prints simple lines that swap direction (90°) on alternate layers.

It's recommended for basic prints or visual prototypes as it **saves filament** and prints fast.

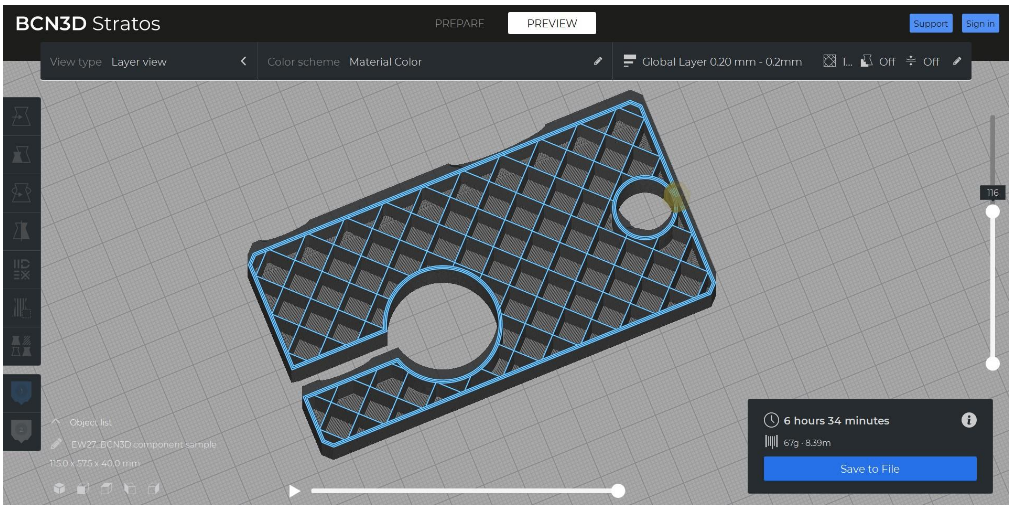



BCN3D Stratos Infill Pattern: Line

## Grid pattern

This is the default infill pattern option. This is because it's one of the fastest infill options and it offers good mechanical properties.

The main difference between the Grid and Line infill patterns is that the Grid pattern is **printed in both directions in each layer**. This means that the part will be more solid, although material may accumulate where the lines intersect.



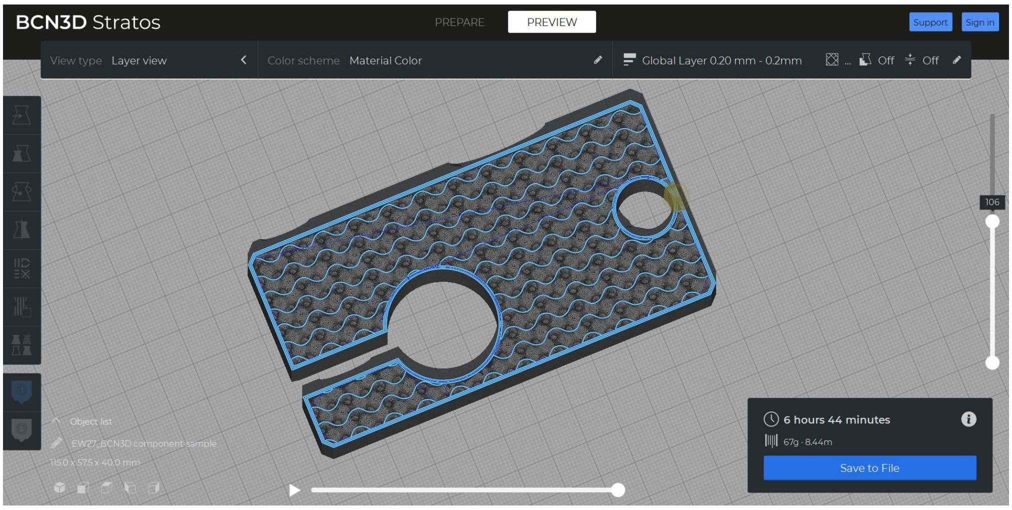

BCN3D Stratos Infill Pattern: Grid

Similar infill types in terms of material usage include triangle, tri-hexagon, cubic, octet, quarter cubic, cross, and concentric patterns.

## Gyroid pattern

Gyroid infill is one of the latest types of infill patterns introduced in 3D printing, and it has quickly become one of the most popular ones.

This tridimensional shape **distributes the strength and support in all directions** while printing relatively fast.




BCN3D Stratos Infill Pattern: Gyroid

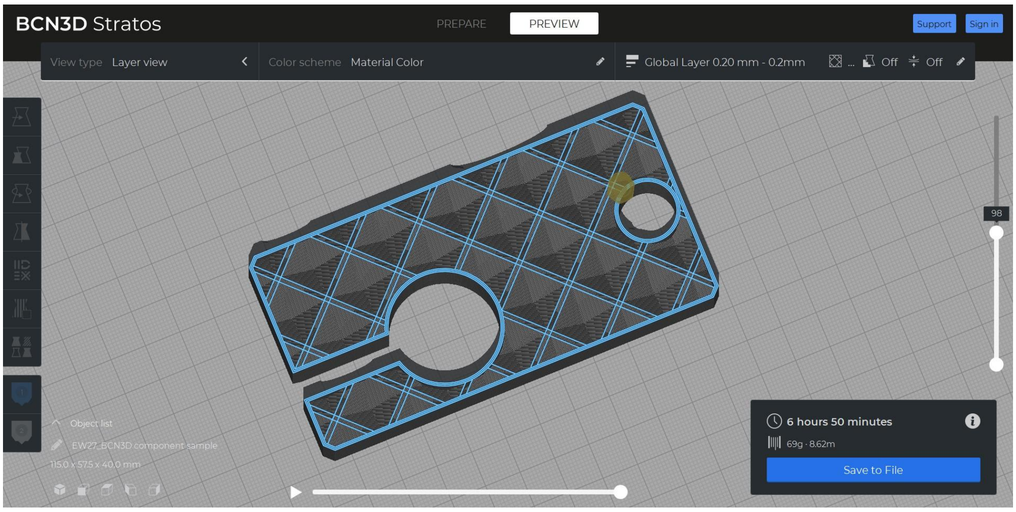
An interesting detail about the gyroid pattern is that all the holes are connected either vertically or horizontally. This allows you to make a small hole in any side of your part and fill it with plaster, cement, or even resin.

Similar infill types in terms of distribution of strength include cubic, quarter cubic, and octet patterns.

### Octet pattern

The octet infill pattern is formed by tilted opposing lines that **form tridimensional shapes** that look like cubes.

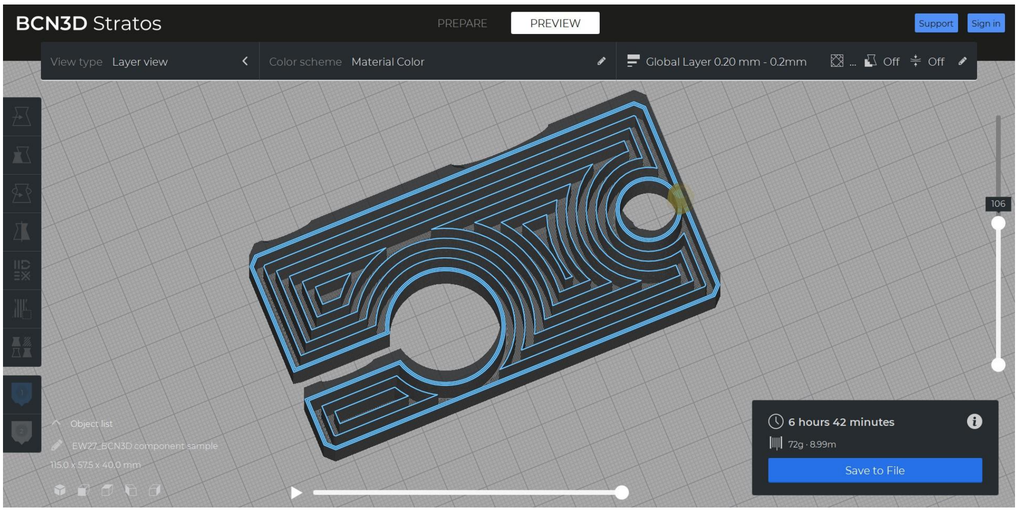
As it happens with the gyroid infill pattern, the Octet pattern forms a tridimensional shape that distributes the strength in all directions.




BCN3D Stratos Infill Pattern: Octet

### Concentric pattern

This type of infill pattern is recommended for parts that are 3D printed using **flexible material**. Its shape is formed by concentric lines that follow the part's external perimeter, allowing for consistent compression from the sides.




BCN3D Stratos Infill Pattern: Concentric

Even though this infill pattern is unique, similar properties may be achieved using a gyroid infill.

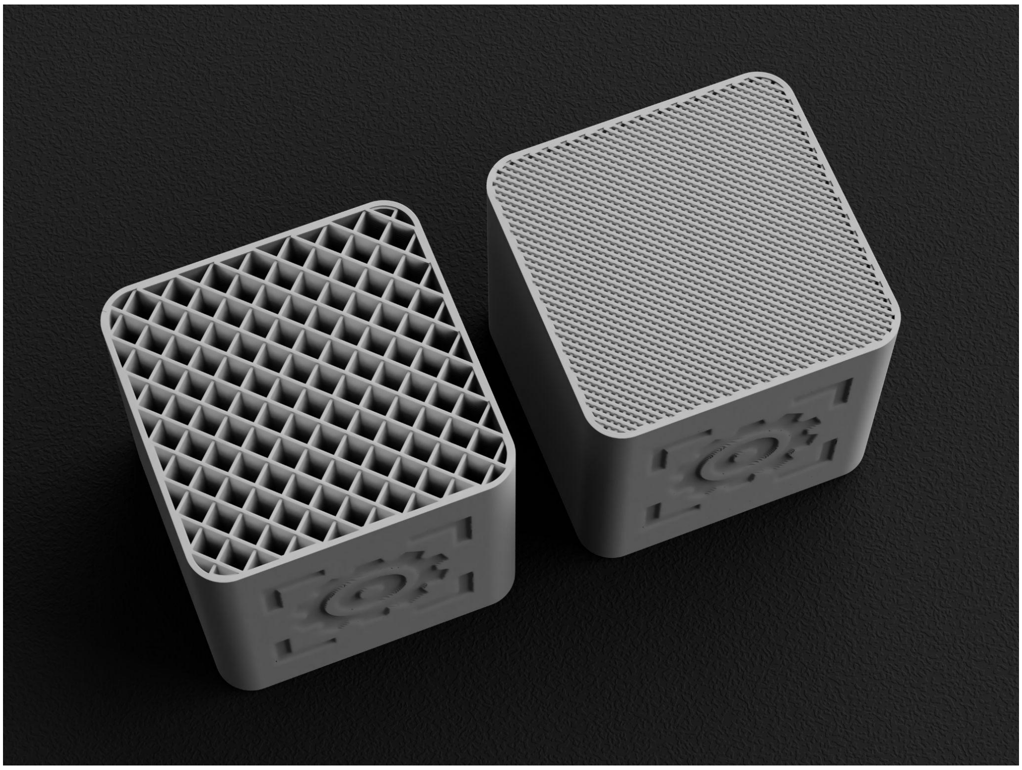
# How infill affects printed parts

Infill plays an essential role in print quality, including how the part feels and how it behaves.

## Part weight

The higher the Infill Density is, the heavier the part will be. It's as simple as that.

For example, the part you can see below weighs 67g when printed with a 10% infill. However, the weight increases to 126g when the infill changes to 40%. This means that a 30% increase in the infill density leads to an 88% weight increase.

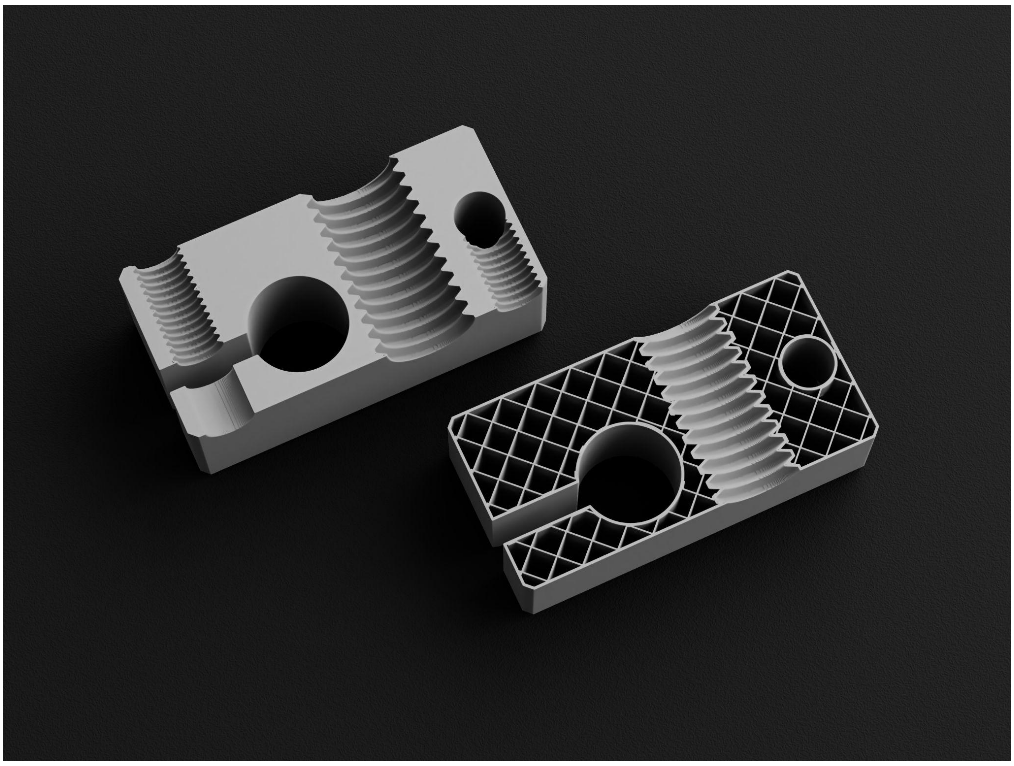


It’s essential to test different prototypes and find the correct infill density, affecting the manufacturing time and cost.

### Part strength

As it happens with part weight, **the higher the Infill Density, the stronger the part**.

It’s important to understand that each part is unique, and its mechanical properties will be affected by many things, including layer height, material, print speed, wall thickness, and infill.

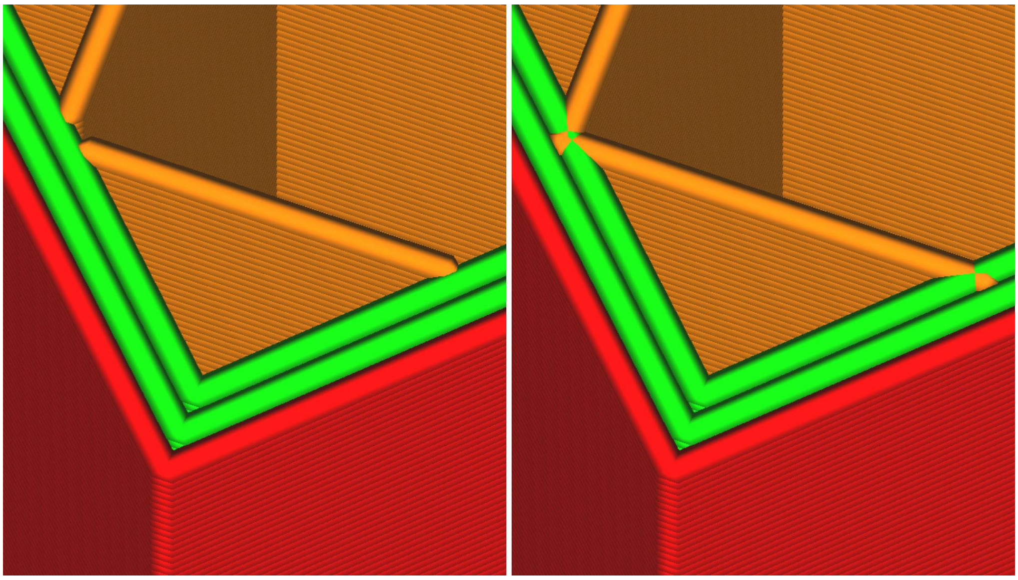


If you’re looking for part strength, you may want to get started with grid, octet, or gyroid infill, although testing will always be required.

## Internal supports

Even though infill is on the inside of the part and isn’t usually visible, it affects the outer layers of the model.

For example, if you increase the *Infill Overlap Percentage* too much, you may see where the infill touches the perimeters on the side of your part. The default overlap value is 15%, and it’s usually high enough.



BCN3D Stratos Infill Overlap Percentage comparison: 0% (left), 75% (right)

The infill also **supports the top layers from the inside**. If your part has a flat top part and the top thickness is too low, you may experience an issue called pillowing. This issue is usually caused by uneven cooling, although a low infill percentage will always make the problem more visible.

The best way to solve pillowing issues is to increase the number of top layers, print slower, and increase the infill density.

# Other slicing considerations

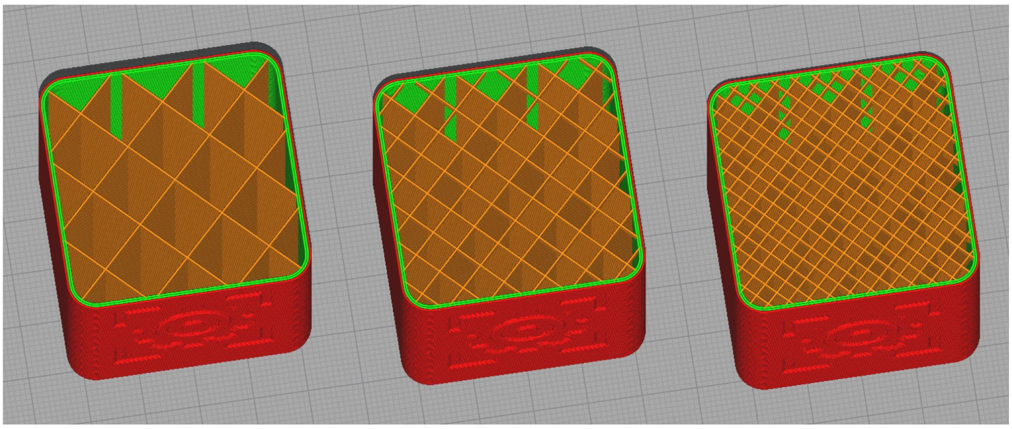
There are many other infill-related settings you should consider. Each one of them can make a significant impact on print quality depending on the part application.

## Gradual infill

BCN3D Stratos allows you to use different infill densities in the same part. The Gradual Infill setting halves the Infill Density of your model the lower it is.

The resulting part will have a high Infill Density on the top layers and a lower one on the bottom. This will reduce the print time while offering a clean top surface.

This setting is recommended if you're printing early prototypes and you need a clean surface finish but not a strong part.

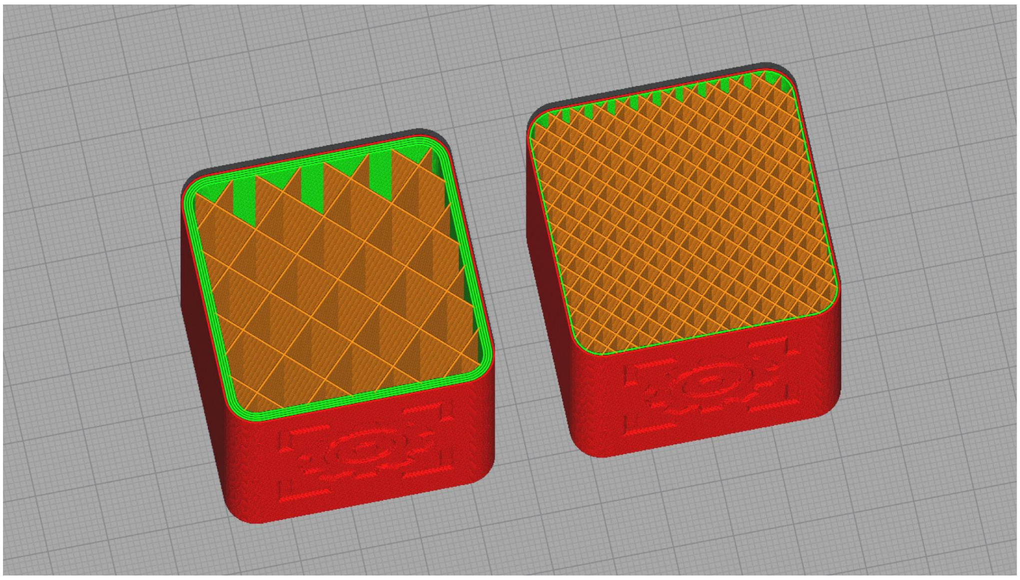


BCN3D Stratos Gradual Infill: layer 200 (left), layer 205 (middle), layer 210 (right)

## Wall thickness vs. Infill

The Wall Thickness and Infill settings are the ones responsible for the part strength.

You can always increase both Wall Thickness and Infill Density to improve the mechanical properties. However, if part weight is a limitation and you need to choose one setting, **increasing the wall thickness will offer slightly better results.**

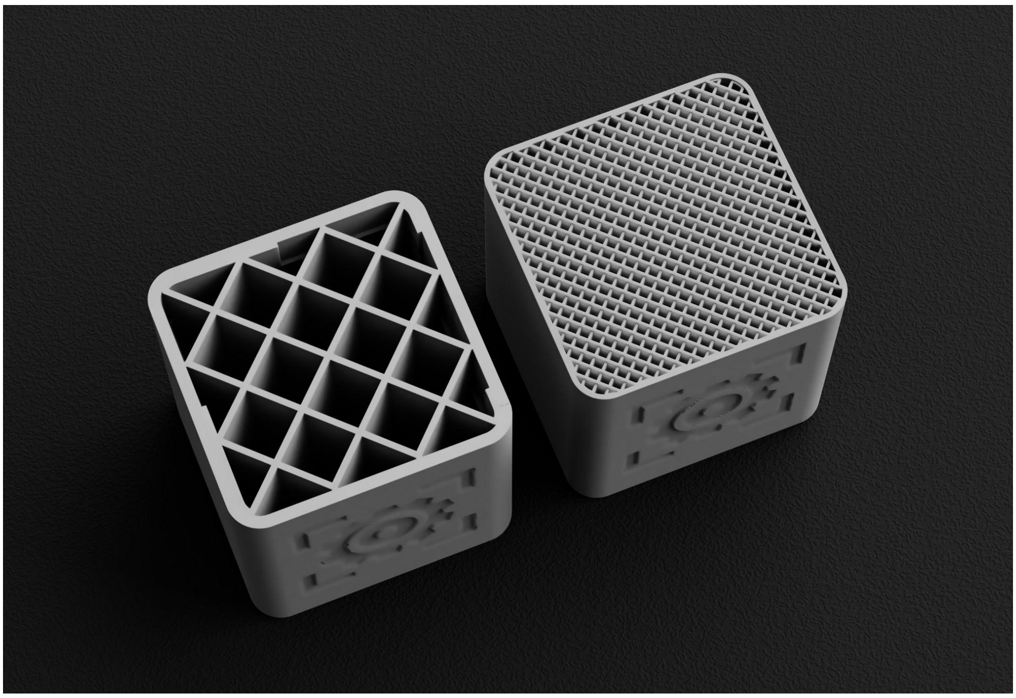


Increased Wall Thickness (left) vs. Increased Infill Density (right)

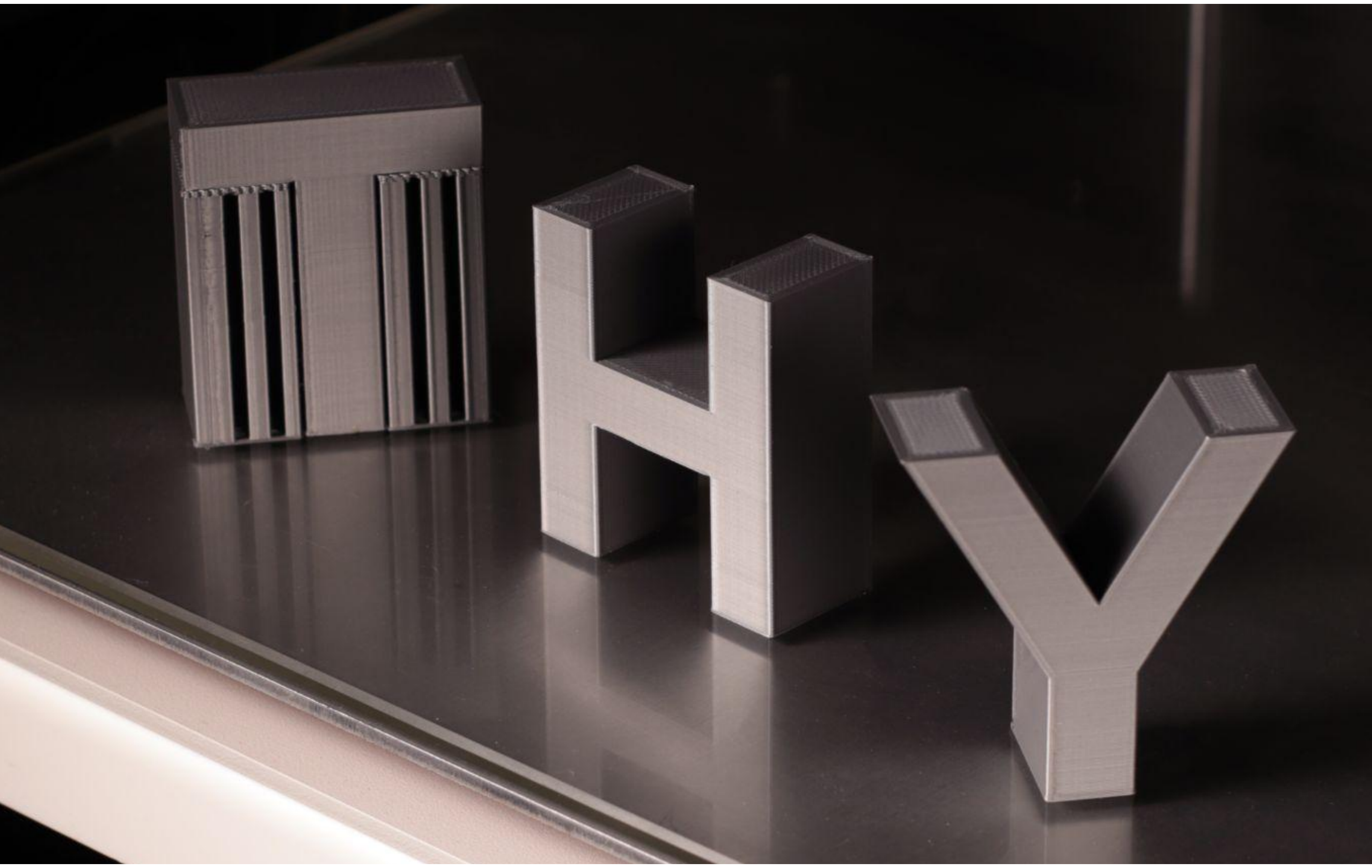
# Infill and Hotend Family

Different hotends are available for the Epsilon and Sigma product families. Each hotend has a different nozzle diameter, from 0.4mm up to 1mm.


The use of larger nozzle sizes allows you to increase the part strength while reducing the print time. For example, in the image below, two models were printed with a 20% Infill Density. However, the model on the left was printed using a 1mm nozzle instead of the standard 0.4mm nozzle. This reduced the print time by more than 30%.



# 5. Support Material



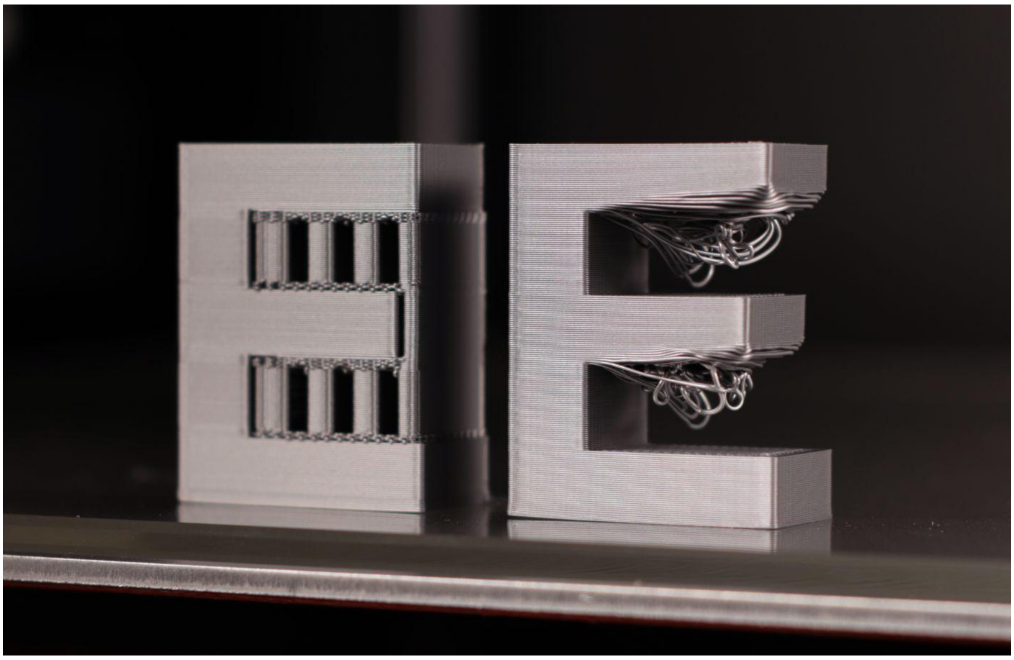
# The importance of support material

FDM 3D printing is an additive manufacturing process where objects are made by depositing material in a predefined way layer by layer. Each manufacturing process has different capabilities and limitations, and FDM 3D printing has one basic rule: Each new layer needs to be supported by the layer below.

But, what happens if your component includes overhangs or features that would be printed on air? Then you'd need to add a support structure.

## Support material explained

The support material refers to the **automatically generated structure that supports the overhangs** and bridges of your model.



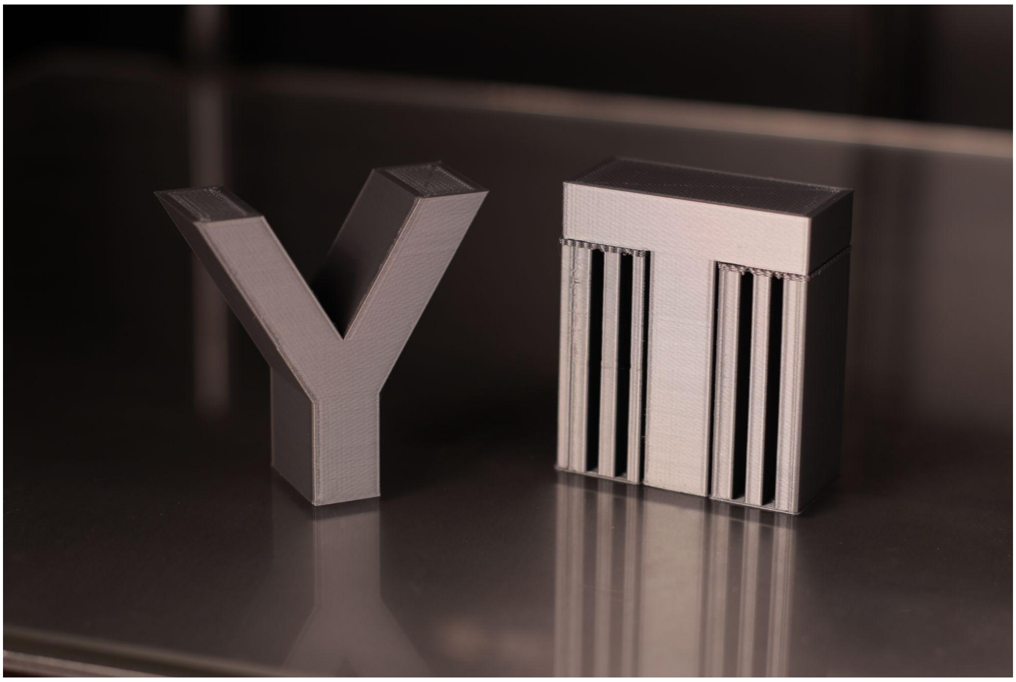
## When do you need support material?

Most models that feature overhangs and bridges require support material to be printed with good visual and mechanical properties. However, there are some exceptions.

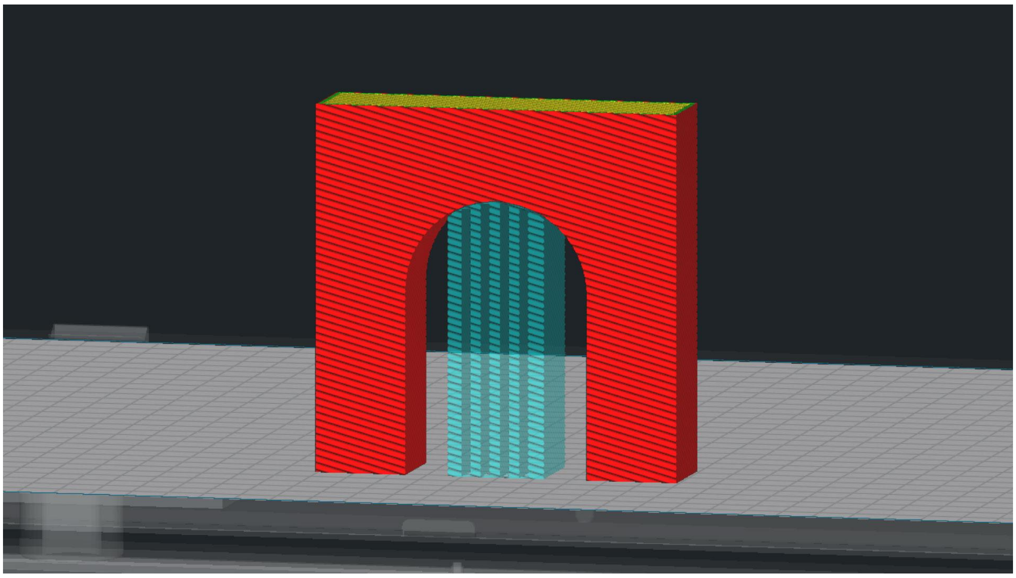
**Overhangs**

All overhangs that are **over 45°** from the vertical should be 3D printed using support structures. The overhang angle could be increased depending on the design, part size and material used, but a 45° inclination is generally accepted as the limit.

The example below shows how the letter T has 90° overhangs on both sides, requiring support material. However, you wouldn't need support material if you printed the letter Y, as the overhang is below 45°.



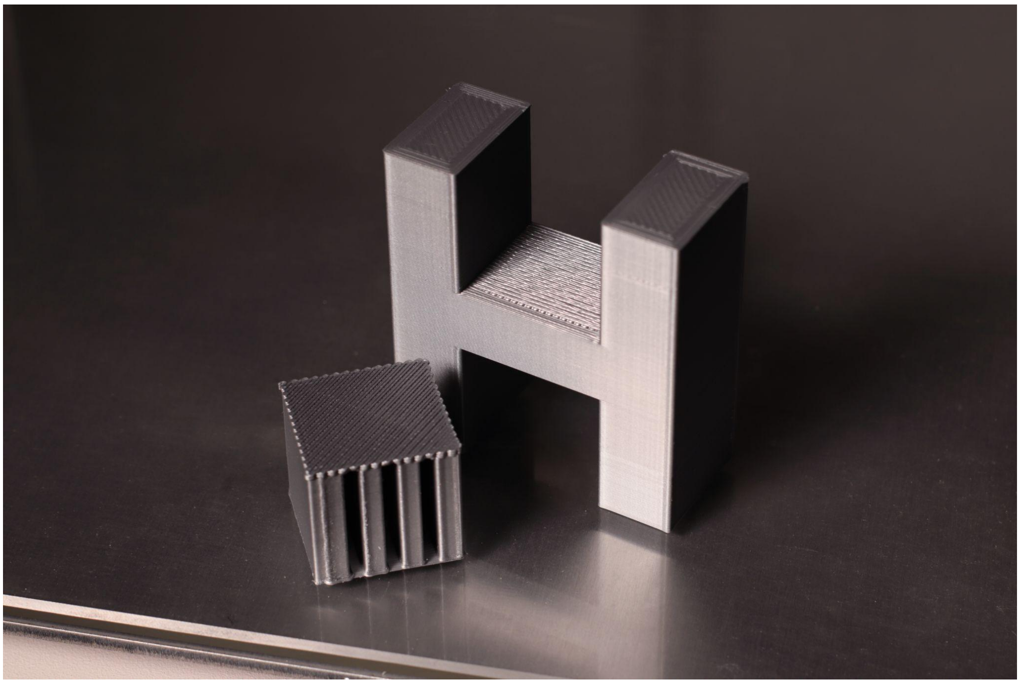
Another great way to understand overhangs is by printing an arch. You will notice that the support structures only start when the overhang exceeds the 45° limit.



**Bridges**

Bridges refer to those horizontal overhangs that connect two points with a straight line.

It's highly recommended to use support material on all bridges, although those under 10mm could be printed with relatively good quality without a support structure below.



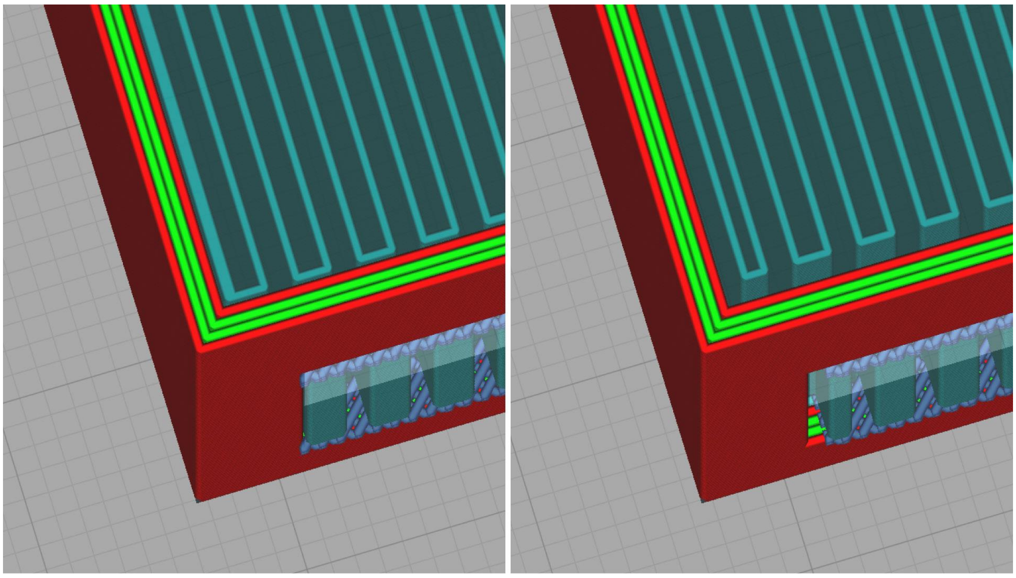
# Essential Support settings

BCN3D Stratos offers hundreds of different slicing settings, and here are the most important ones for support material.

## X/Y distance

This setting affects the distance between the support structure and the walls of the part in the X/Y directions.

If you need a **clean surface finish** on your design, consider increasing the X/Y distance to reduce the chances of getting the support structure stuck to one of the sides of your print.



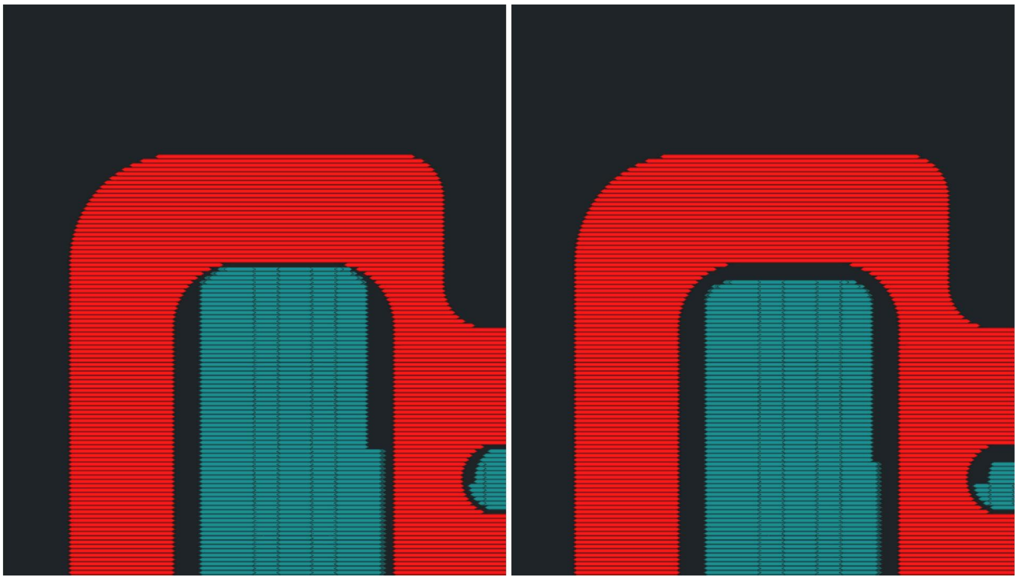
Support XY Distance: 0.1mm (left), 0.8mm (right)

### Z distance

This is probably the **most critical support setting**, and it sets the distance between the last layer of the support structure and the part it needs to support.

The standard Z distance is 0.2mm, although each design and material have unique values. It's highly recommended to test this setting by 3D printing some samples before jumping to the final print.

In the image below, you can see the printed part (in red) and the support material (in blue). If the Z distance is too low, the support structure and the part will fuse, difficulting support removal. On the other hand, if the Z distance is too high, the print will not have enough support and the surface finish and tolerances may be compromised.

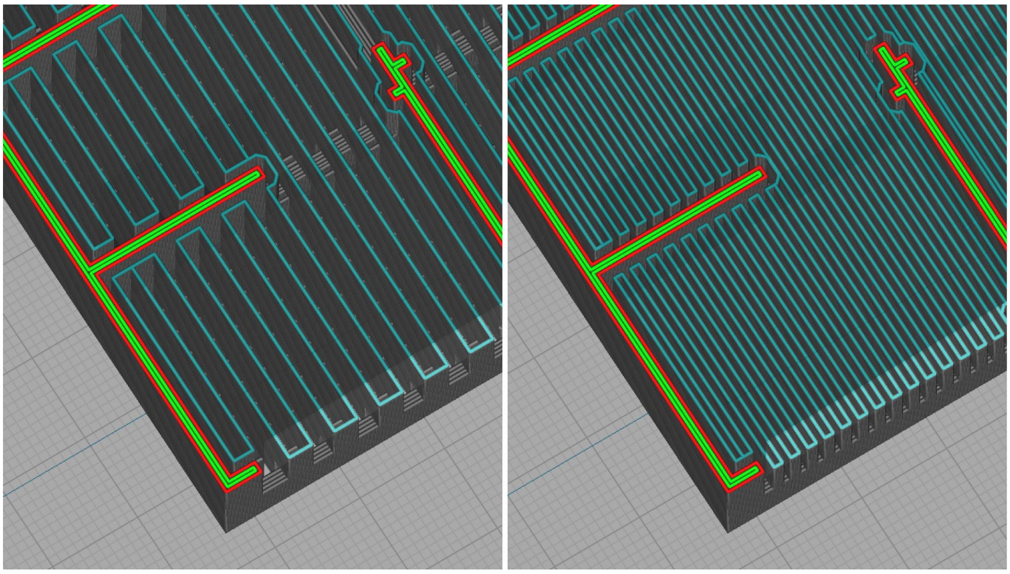


Support Z Distance: 0.1mm (left), 0.8mm (right)

## Support density

The support structure is an automatically generated mesh. As it happens with the Infill, we can adjust the mesh density. The standard infill density is 15%.

A higher infill density will improve the structure's strength, but the supports will be harder to remove and the print time will increase.

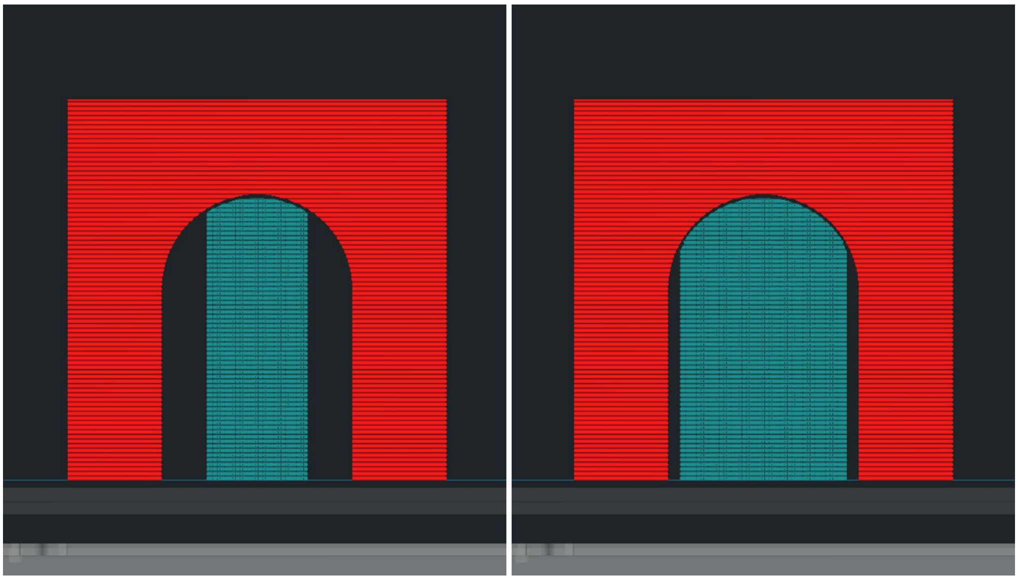


Support Density: 15% (left), 30% (right)

## Support overhang angle

If your model has overhangs at a different angle, you can choose which ones need support with this setting.

At a value of 0°, all overhangs are supported, and at 90°, there won't be any support material. The default value is 60°, although 45° is usually considered a safe value if the material and design haven't been tested.

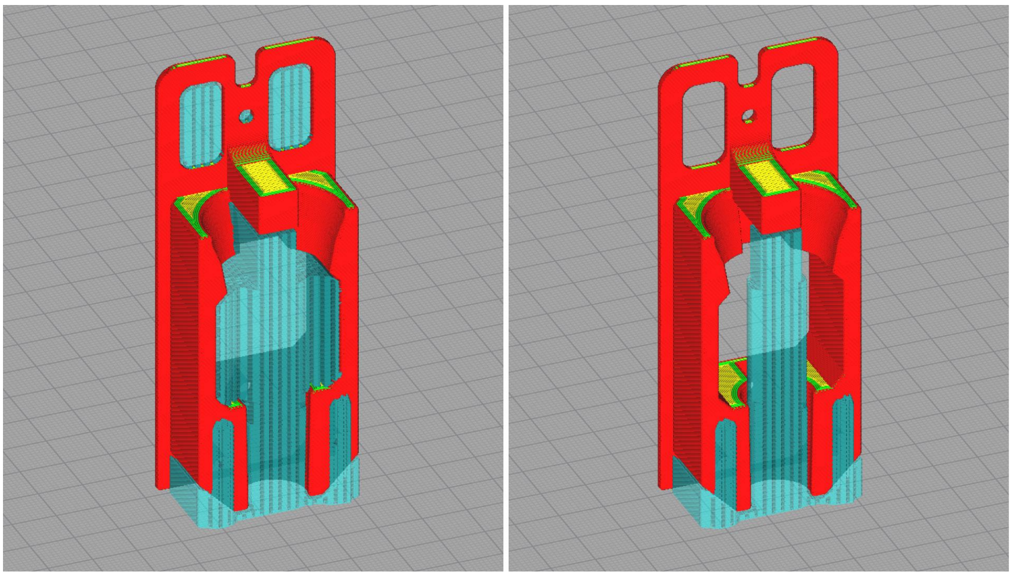


Support overhang angle: 60° (left), 30° (right)

## Support only on build plate

This setting adjusts the placement of the support structures, and they can be placed only on those areas where the structures touch the build plate or everywhere.

If you choose to place support structures everywhere, some of them may have their starting layer on top of the print, affecting the outer shell's quality.



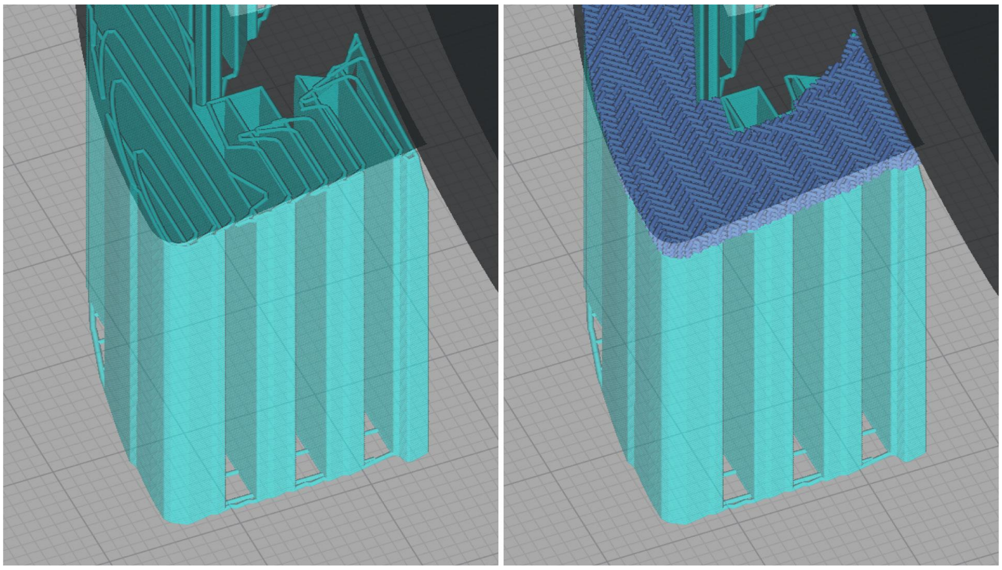
Support placement setting: Everywhere (left), Touching build plate (right)

Support structures are usually more stable when printed on the build plate as the adhesion is generally good. On the other hand, the structures that are printed on the part need to leave a gap (Z distance) when printed, which leads to reduced adhesion.

If your design requires support material on a specific area, consider rotating it so that the support structure is only touching the build plate.

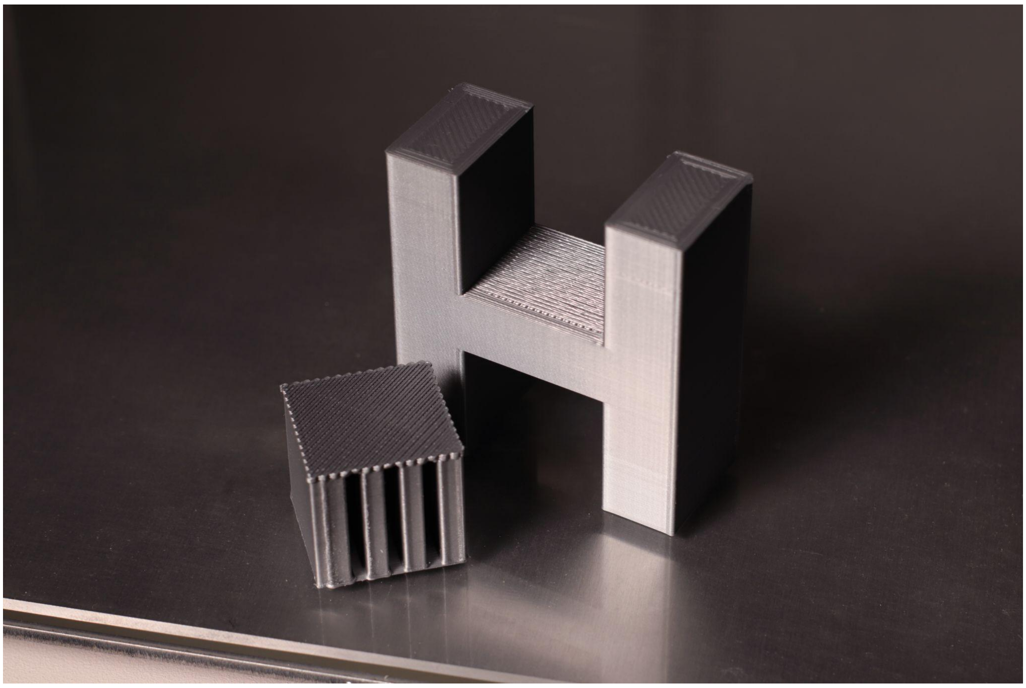
### Support interface

This setting generates a dense interface between the model and the support. It's a **smooth surface at the top of the support structure** on which the model is printed.



Support Interface: Disabled (left), Enabled (right)

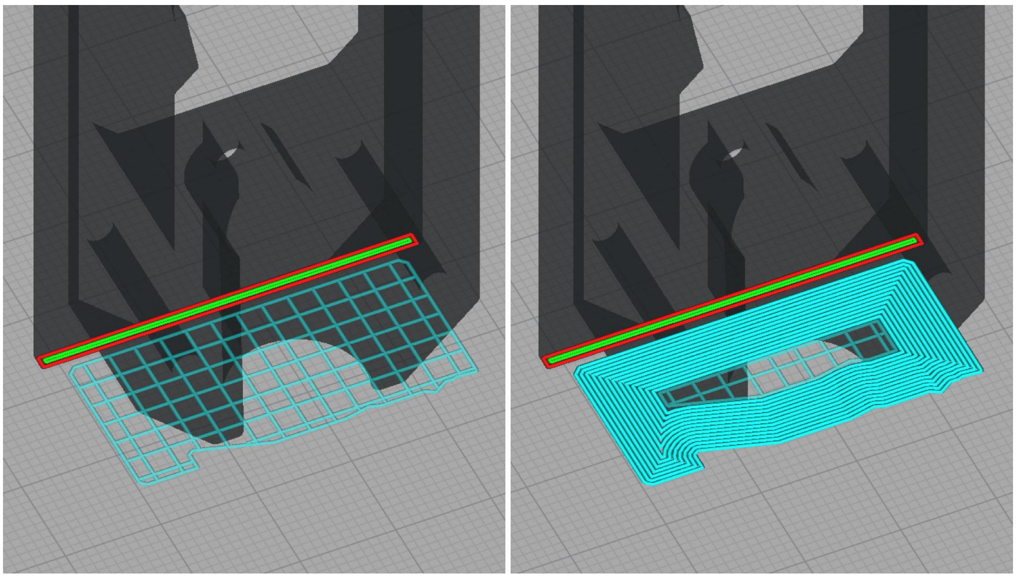
Removing supports structures with an interface is usually easier than removing regular supports, and the **surface finish is generally better**.



## Support brim

This option generates a brim within the support infill regions of the first layer, increasing the support adhesion to the build plate.

It's recommended in those cases where the support density is low and the model has thin parts that need rigid supports.



Support Brim: Disabled (left), Enabled (right)

# Support materials

Thanks to the IDEX technology, you can use different materials in one print, offering new ways to manufacture quality components.

The support settings listed above work for all these materials, but each one of them adds some exciting benefits.

## Same print material

If the print you're manufacturing has a geometric design or doesn't need to be accurate, you can use the same material for both the print and the support material.

- It is recommended for parts with simple or geometric shapes.
- Reduced print time compared to dual material prints.
- Increased part-support bonding as they're the same material.

## PVA

PVA is one of the most popular 3D printing materials. It's exclusively used as support material because it's water-soluble and can be easily combined with other popular materials.

- Water-soluble.
- It is recommended for complex parts that have internal cavities.
- Great adhesion with PLA, PETG, TPU and Nylon.

Learn more about PVA with the How to print with BCN3D PVA and Tips and Tricks articles.

## BVOH

BVOH (Butenediol vinyl alcohol copolymer) is a water-soluble thermoplastic optimized for the FFF manufacturing process.

- Water-soluble.
- Compatible with PLA, PETG, ABS, PA and PAHT CF15.
- Fast to dissolve.

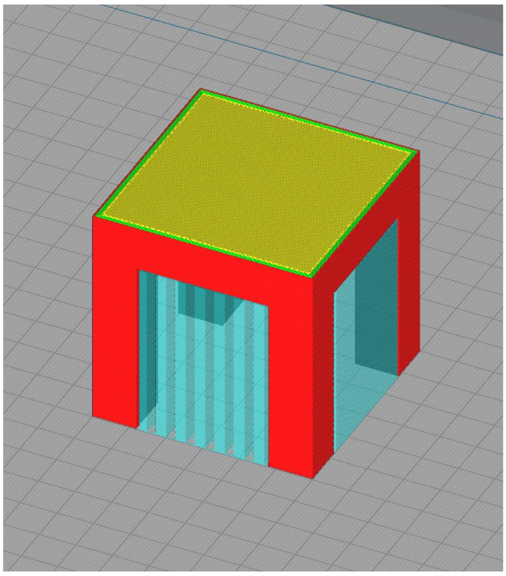
# How to avoid using support material

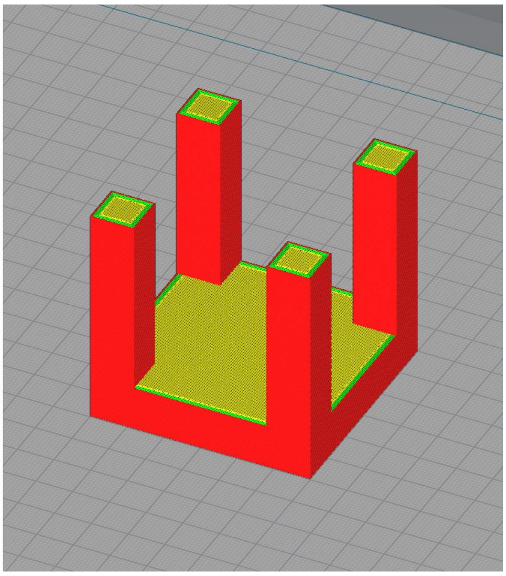
It's always important to consider the manufacturing technique when you design a model. It's not the same to design for FDM 3D printing as for injection molding.

Find below some design tips that will help you reduce the amount of support material your prints need.

## Part orientation

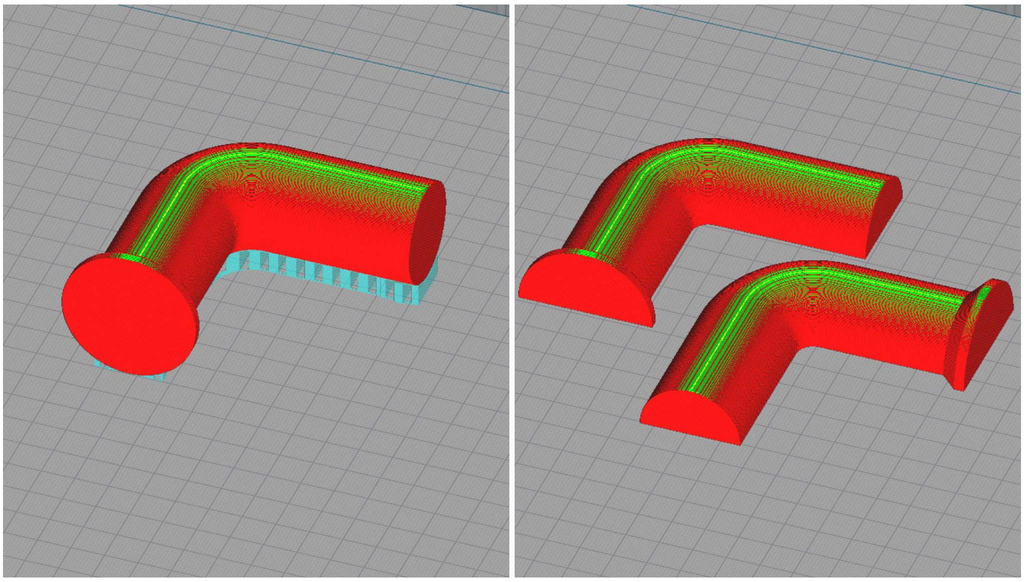
The part orientation influences the number of supports structures that are generated. It's recommended to use the face of the model as a base where most of the details grow in the Z axis direction.





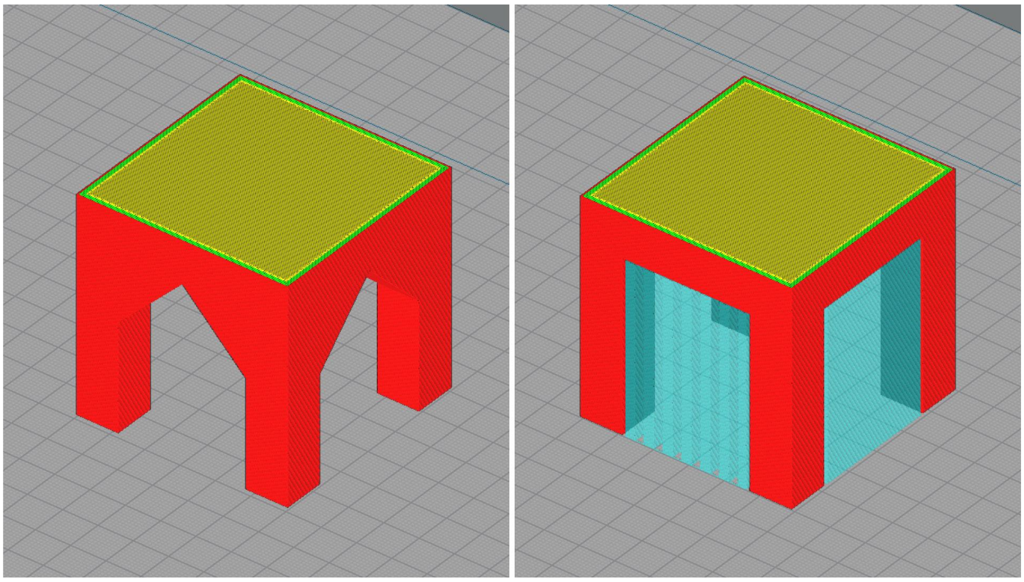
## Model division

Dividing the model into parts that don't require supports to be printed is an excellent solution to save material.



## Add chamfers

Adding chamfers to horizontal parts of the design reduces the need for support structures and also increases the part strength.

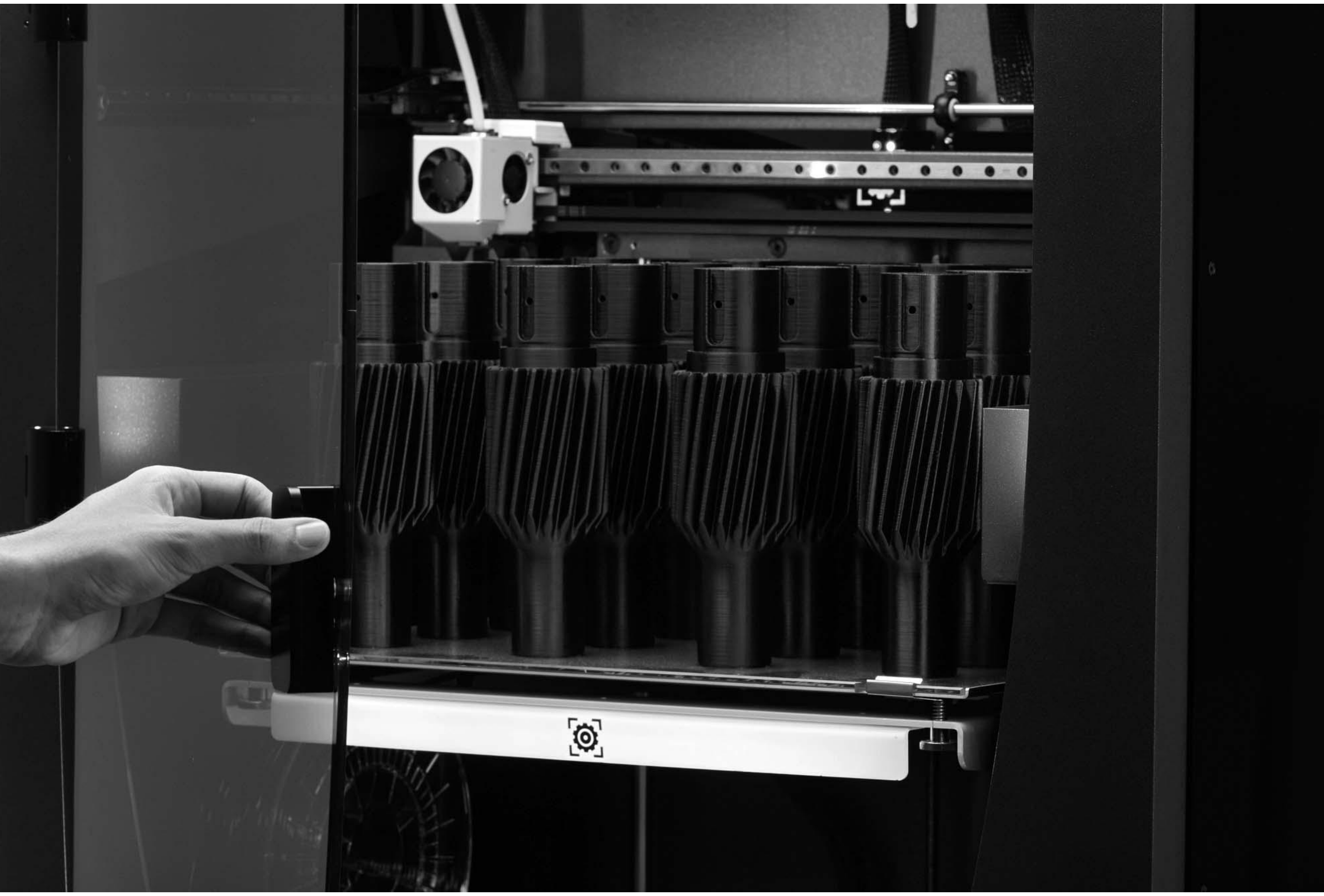


# Support material and Hotend Family

Different hotends are available for the Epsilon and Sigma product families. Each hotend has a different nozzle diameter, from 0.4mm up to 1mm.

The use of larger nozzle sizes increases the part and support structure strength while reducing the print time.

# 6. Build Plate Adhesion



# The first step to a successful print

Each additive manufacturing process has different capabilities and limitations, but most technologies share a common rule: The initial layer must be perfect to guarantee a great 3D printing experience.

This white paper will go through the different tips that will help you get a perfect initial layer.

## Initial layer explained

FDM 3D printing is an additive manufacturing process where objects are made by depositing material in a predefined way layer by layer. **The initial layer is the first layer to be 3D printed and the one touching the build plate.**

## What is Build Plate Adhesion?

Build Plate Adhesion - also known as Bed adhesion - refers to the group of settings and 3D printer components whose primary purpose is to guarantee the adhesion of the initial 3D printed layer to the build plate.

All these components and settings should be checked during the slicing process in BCN3D Stratos and in the 3D printer itself.

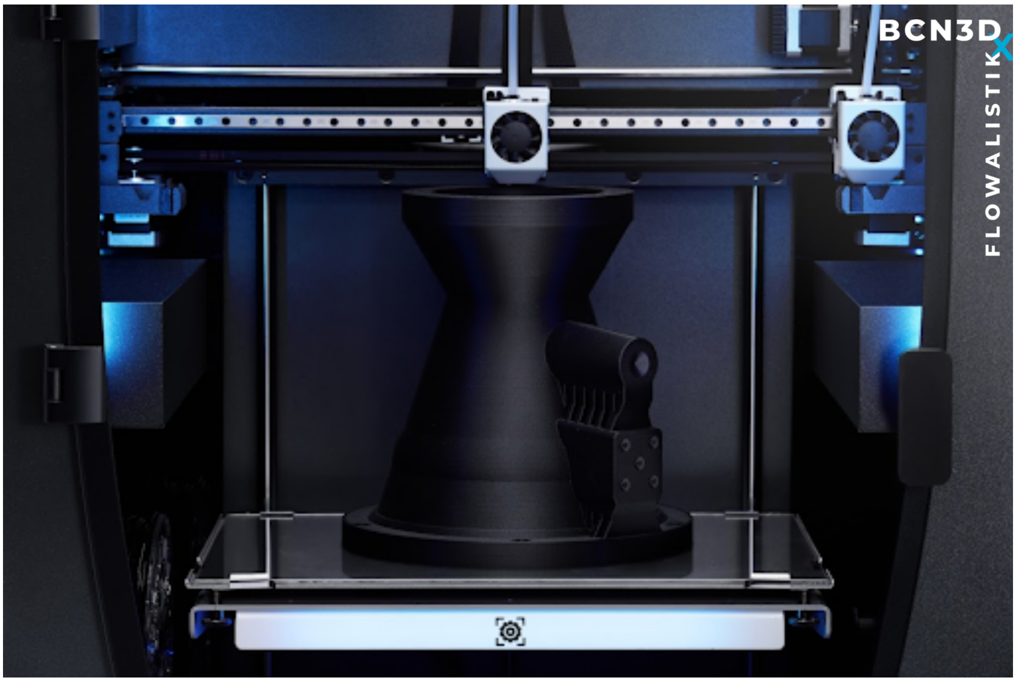
# Common adhesion issues

Build plate adhesion issues can be frustrating. Many different variables affect the 3D printing process, but they usually fall into one of the following categories when it comes to adhesion issues.

## Build plate leveling

If you have build plate adhesion issues, bed leveling is the first thing to check. If the build plate is not level, one side of it will be closer to the nozzle than the other, producing inconsistent build plate adhesion.

The BCN3D product family features a borosilicate glass with a very low thermal expansion coefficient, allowing for reliable bed leveling.




A build plate leveling assistant is also included, featuring a Mesh leveling feature that allows you to control the plate leveling.

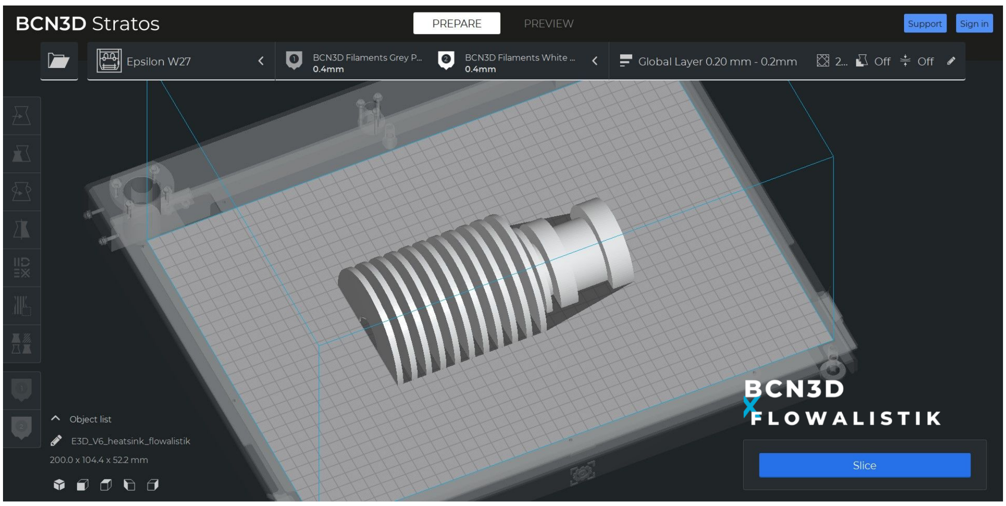

## Unclean build plate surface

An excellent build plate adhesion can only be achieved with a clean surface. Checking the build plate and cleaning it with alcohol and a microfiber cloth after some prints is part of every efficient 3D printing workflow.

Besides a clean build plate, applying a thin layer of adhesive is always recommended to improve the 3D printing experience.

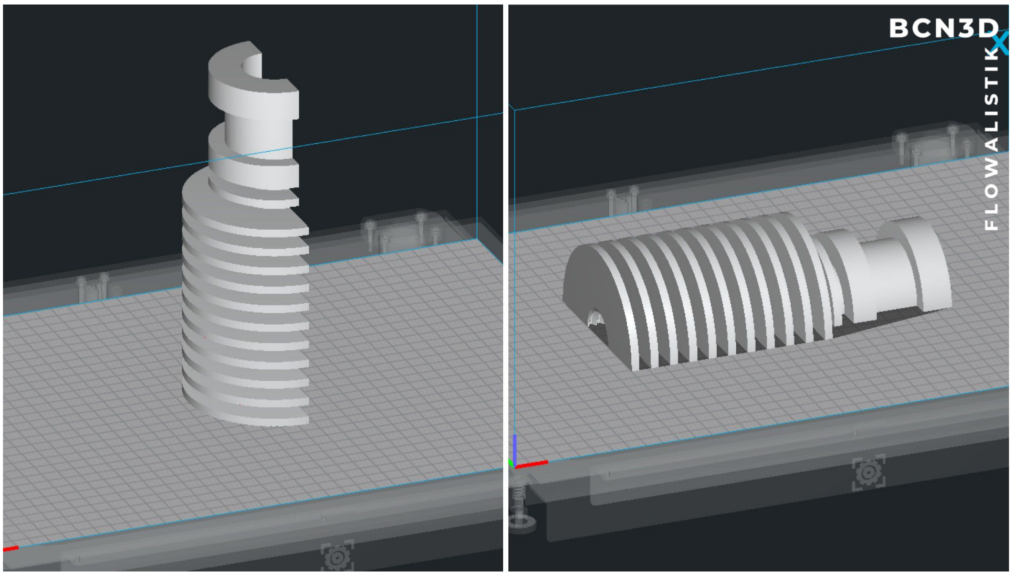
## Wrong slicing settings

Different slicing settings affect build plate adhesion, including print speed, temperature control and even line width. The most important ones are explained below in the white paper.



### Part orientation

The more surface area the initial layer has, the better. When possible, it's always recommended to orient the parts to maximize the initial layer surface area.




# Types of Build Plate Adhesion

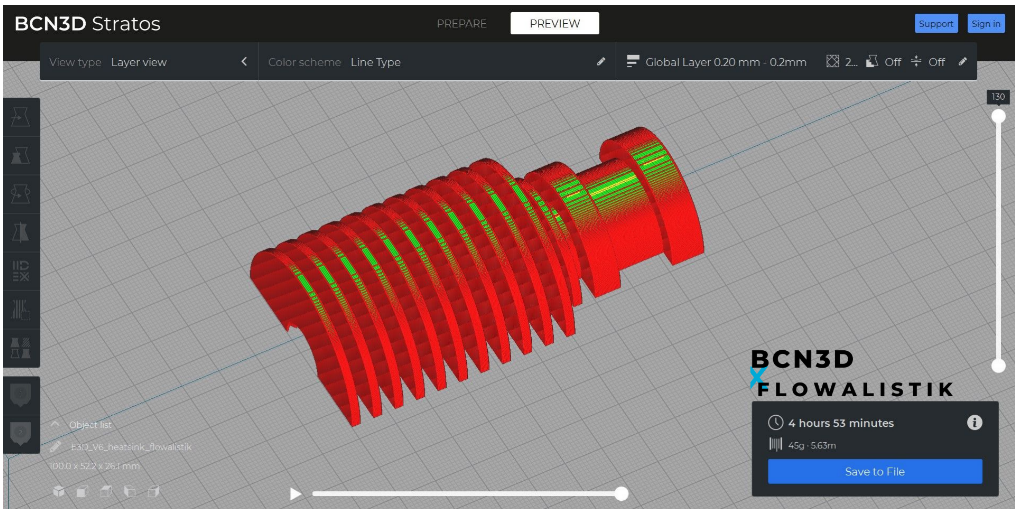
Within all the adhesion-related settings, there's one that stands out from the rest: Build Plate Adhesion Type.

In BCN3D Stratos, you can find four different options that can help you improve both priming the extrusion and the build plate adhesion.

**None**

When this option is selected, the 3D printing process starts as soon as the temperature is reached and the material is purged.

This option is not recommended unless the 3D printing process is entirely under control. If you want to reduce the print time or the amount of material used, the Skirt feature is the best option.

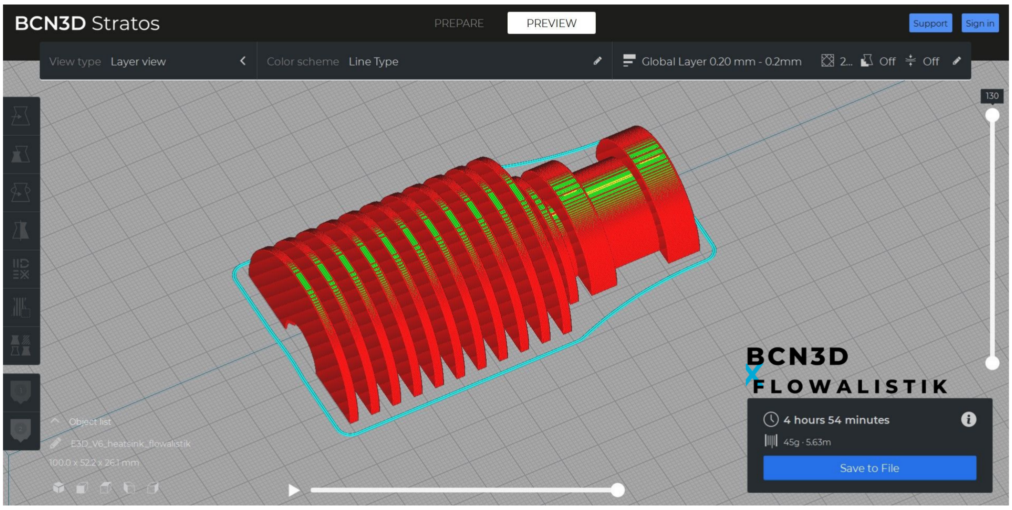



## Skirt

The skirt is a line printed around the model but not connected to it.

Even though this option doesn't directly affect build plate adhesion, it can prevent issues in those cases where insufficient material is purged before the print starts.

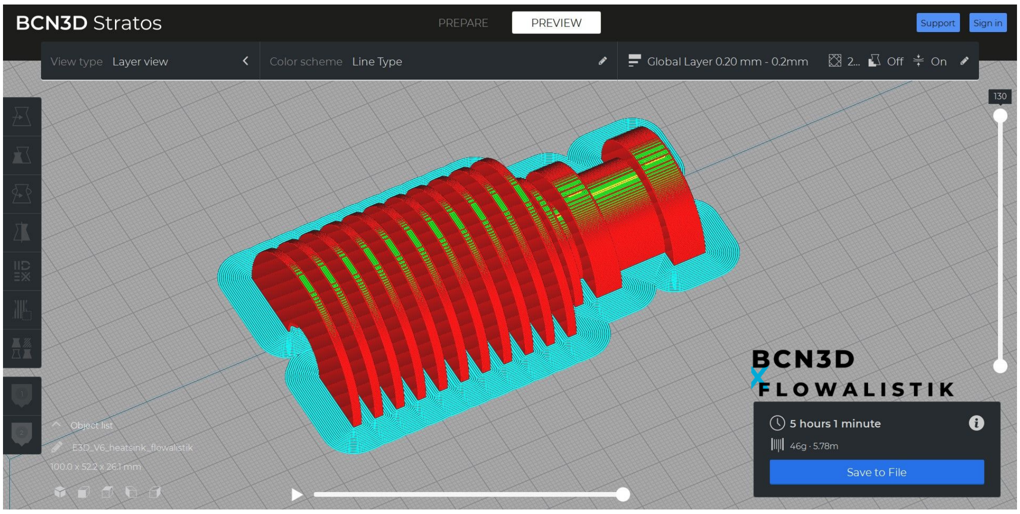
More material is purged by printing a line around the model, guaranteeing that the right amount of material will be extruded when the initial layer starts printing.




## Brim

The brim adds a single layer of flat area around the base of the model, preventing warping.

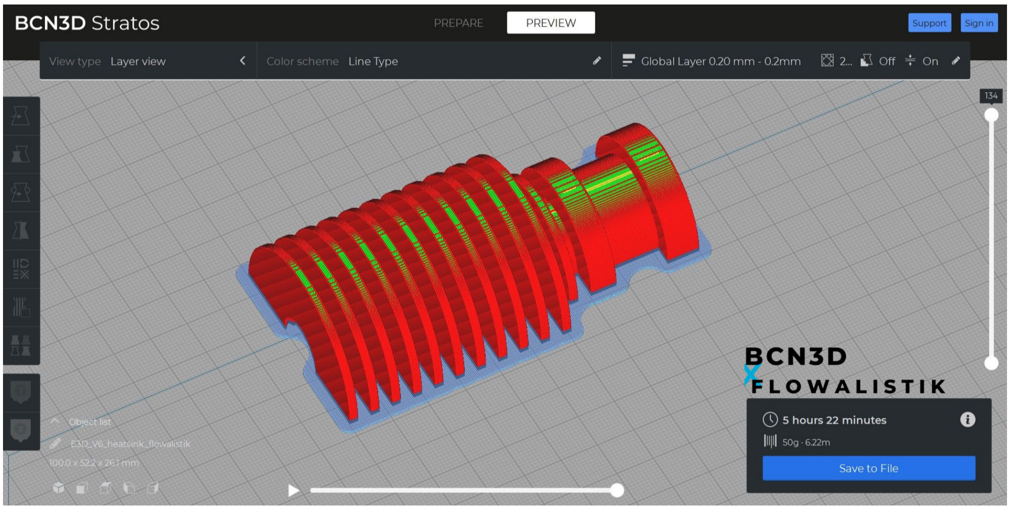
This uses a bit more material, but it's essential when using materials such as ABS that tend to warp. It's also recommended when manufacturing parts that have a small surface area.




## Raft

The raft adds a thick grid with a roof below the model, which guarantees a good build plate adhesion.

This option is only recommended if the model geometry and material present apparent build plate adhesion issues. It consumes a relatively large amount of material and generates a textured bottom layer as it doesn't touch the smooth build plate.



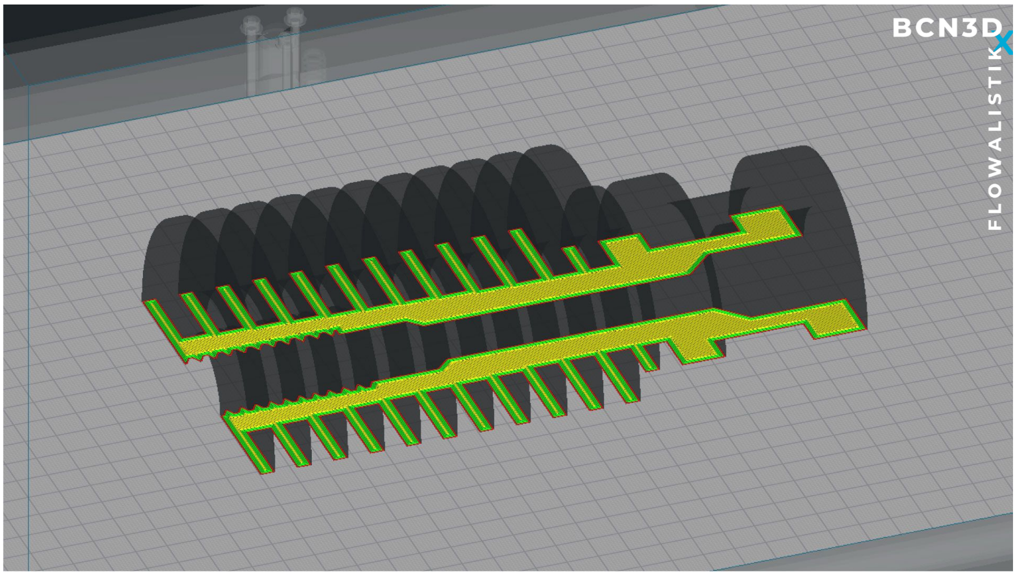
# Essential build plate adhesion settings

BCN3D Stratos includes many different features that affect the build plate adhesion. Find the most relevant ones below.

## Initial Layer Height

A thicker Initial Layer usually makes adhesion to the build plate easier. However, if the layer is too thick, the nozzle may not be close enough to the build plate.

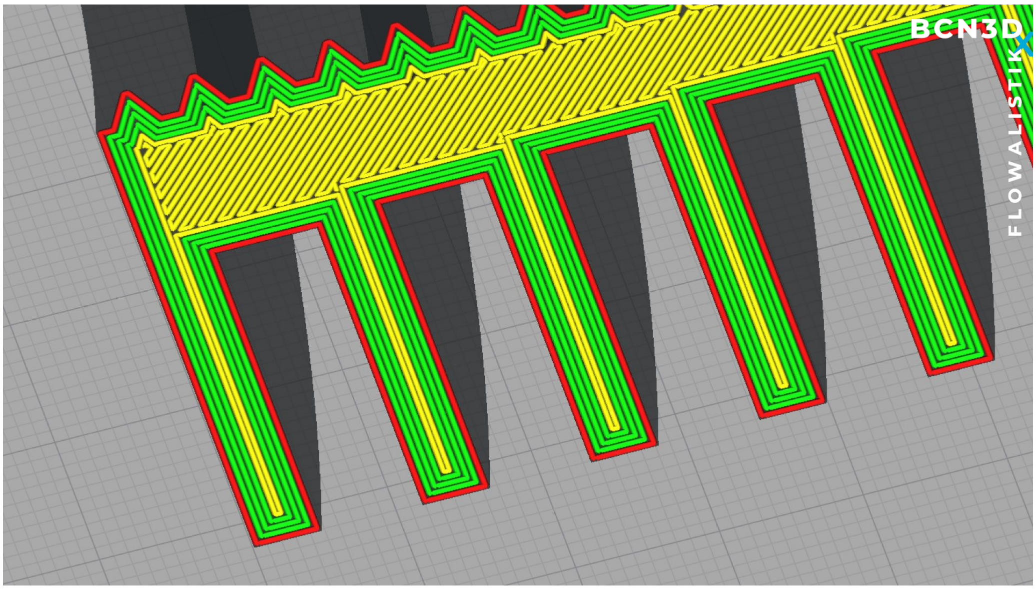
The standard print profiles set an Initial Layer Height of 50% of the nozzle diameter, and this means that a 0.2mm Initial Layer Height is recommended when using a 0.4mm nozzle.




## Initial Layer Line Width

This setting multiplies the standard line width on the first layer. Increasing this setting usually improves build plate adhesion.

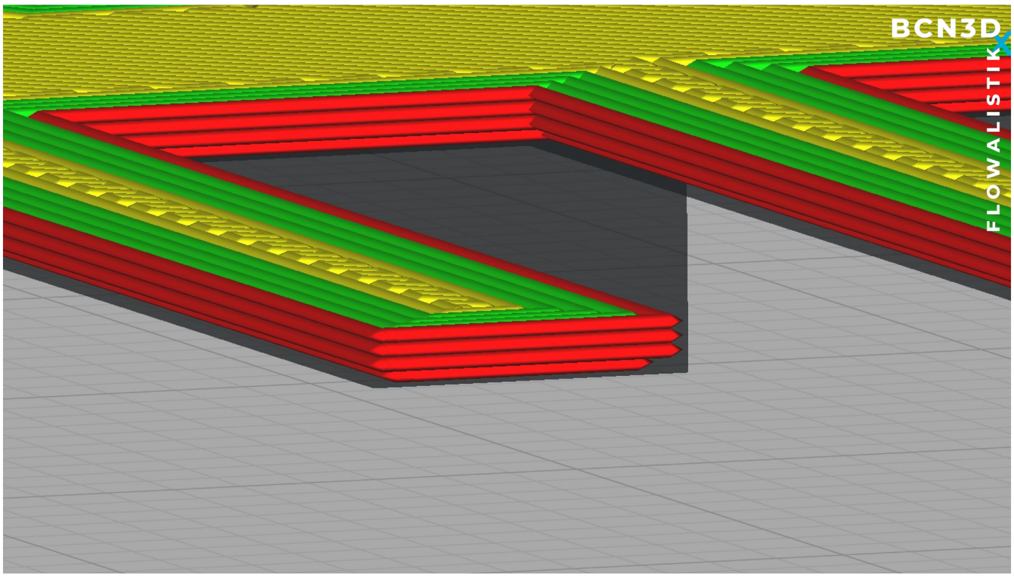
In BCN3D Stratos, the standard Initial Layer Line Width is set to 120%, meaning a standard 0.4mm Line Width will have a 0.48mm width on the initial layer.



## Initial Layer Horizontal Expansion

This setting is handy if the *elephant foot* issue affects your 3D printed parts. This issue expands the initial layer outwards affecting dimensional accuracy, although it increases the build plate adhesion.

It's common to adjust the bed leveling so that the distance between the nozzle and the build plate is too large to avoid the *elephant foot* issue. However, with this setting, you can offset the polygons in the first layer, compensating for squishing the first layer while increasing the build plate adhesion.



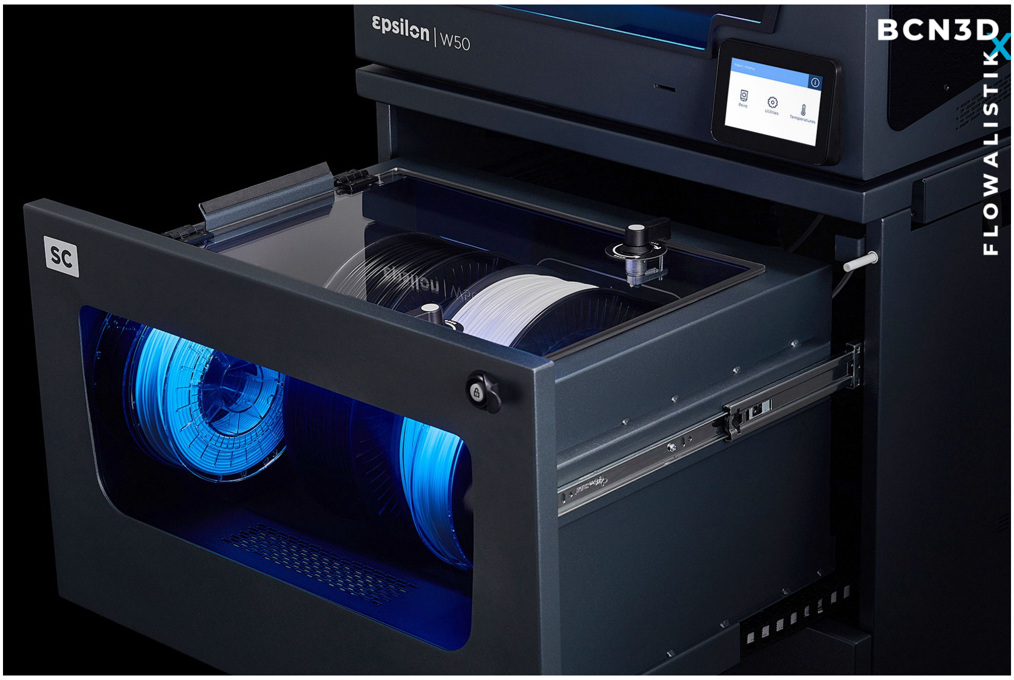
## Initial Layer Print Temperature

In BCN3D Stratos, you can adjust the print temperature of the initial layer. To improve the build plate adhesion, it's recommended to use a slightly higher temperature for the initial layer.

## Build Plate Temperature

Each 3D printing material has a different recommended build plate temperature. It's always recommended to check the recommended print settings and adjust them in BCN3D Stratos.

Build Plate Temperature can be live adjusted during the print process to guarantee the build plate adhesion.

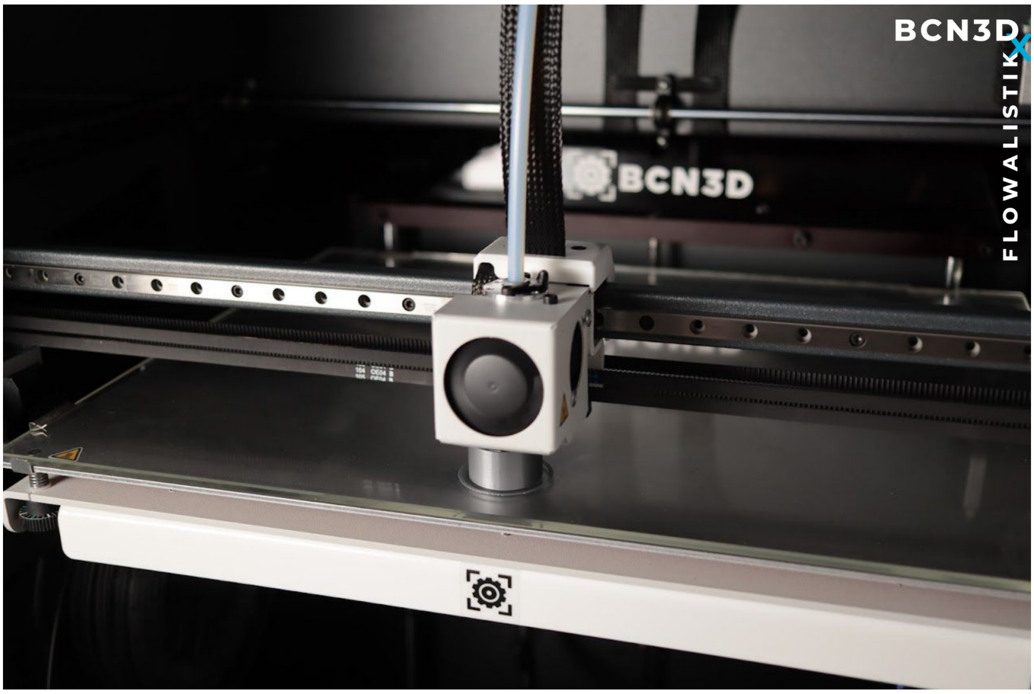



## Initial Layer Speed

A lower value is advised to improve adhesion to the build plate. For example, the recommended Initial Layer Speed is at least 50% slower than the standard print speed.

## Cooling Layer

As it happens with the Build Plate Temperature, each material has a recommended Print Cooling setting. However, turning off the cooling fans during the initial layer is always recommended to increase the build plate adhesion.




# Use Of Adhesives

Even though all the tips shared above help prevent build plate adhesion issues, some advanced 3D printing materials such as Polyamide or Polypropylene require the use of adhesives to guarantee a great 3D printing experience.

BCN3D offers a wide range of Magigoo adhesives you can use to make sure the prints stick firmly when the build plate is hot and that the part is easy to remove once it cools.

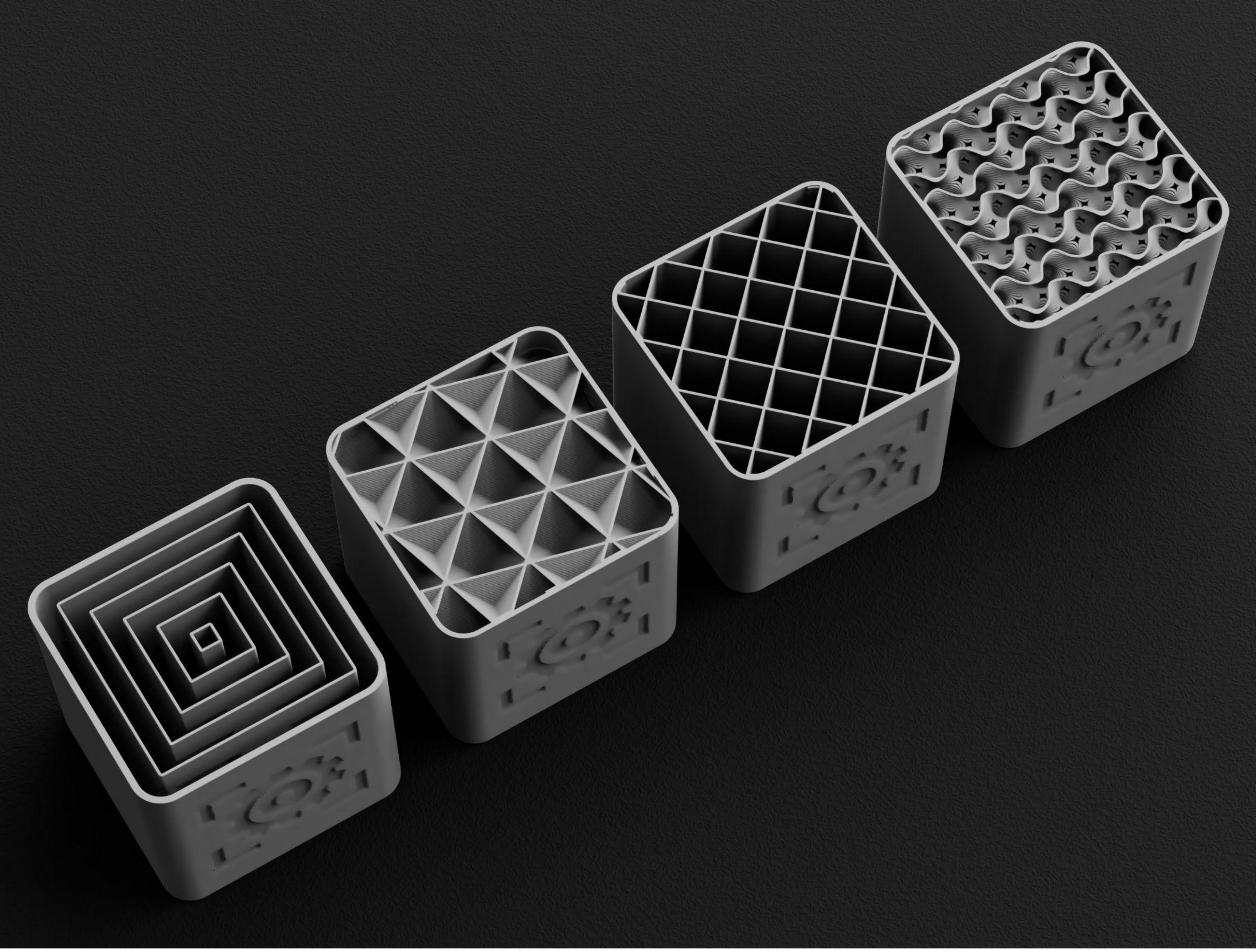
# Learn more

If you are considering implementing 3D printing technologies into your workflow, schedule a customized live demo with one of BCN3D's 3D printing specialists. You will get an overview of all of the BCN3D solutions, see the 3D printers in action, and all your questions will be answered. Book a demo >

Explore more about 3D printing. Learn more >

Wondering what's new in the 3D printing world? Success Stories >

Request a quote for a professional desktop 3D printer. Request a quote >



BCN3D Slicing Guide

# BCN3D x Flowalistik

www.bcn3d.com

BCN3D