3D Printing Reference Guide

Fall 2025

The Creative School
Design + Technology LAB

Table of Contents

3D Printing General Overview

Welcome to the wonderful world of 3D printing! In this document, you will find information about **3D file preparation**, **printing specifications** and **best practices**.

What is 3D Printing?

3D printing is an additive manufacturing process that creates physical, three-dimensional objects using digital 3D models. These objects are often made with thermosoftening plastics or UV sensitive resins. The material is added to a platform layer by layer, resulting in the form of the object.

There are 4 Basic Steps with all 3D Printing Processes:

1. **Get an .STL Model**

Most 3D Prints are modelled using Computer Aided Design (CAD) software; there are many software options for all levels of modelling experience. Alternatively, you can use open source models from online libraries like Thingiverse or Printables. Once an object has been modelled with all design considerations, **export** your 3D model as an .STL file. Submit to the D+TL 3D Printing Services Google Form for processing.

2. **Select a Printing Process**

A model's structural properties, intended use, print time and material preference often determines the appropriate print process.

The Design + Technology LAB supports three different 3D printing processes:

FDM (thermosoftening filament extrusion) printing uses spools of PLA plastic filament. A spool of filament is fed through a heated nozzle and strategically layered to form the shape of the model.

Example of an FDM-printed dinosaur

SLA (vat polymerization) 3D printing uses resin which is cured with UV-light. A tray of UV-sensitive resin is hardened layer by layer onto a platform using a laser.

Example of an SLA-printed dinosaur

SLS (nylon powder fusion) printing uses a laser to fuse together nylon powder to create a solid 3D print without the need for supports.

Example of an SLS-printed dinosaur

3. Assign Print Settings *(performed by LAB technicians)*

Assign print settings to your STL with a separate “slicing” software, such as Bambu Studio or Cura for FDM prints, or Preform for SLA and SLS prints. **Slicing** refers to the conversion of an .stl file to .gcode. An STL file holds structural properties of the three-dimensional model, while .gcode translates these properties into strategic pathways for a printer to execute. These pathways are aligned with the printer's parameters and are formatted using a combination of X, Y and Z coordinates. While this is a step that is done by technicians in the D+TL, we recommend familiarizing yourself with slicing software.

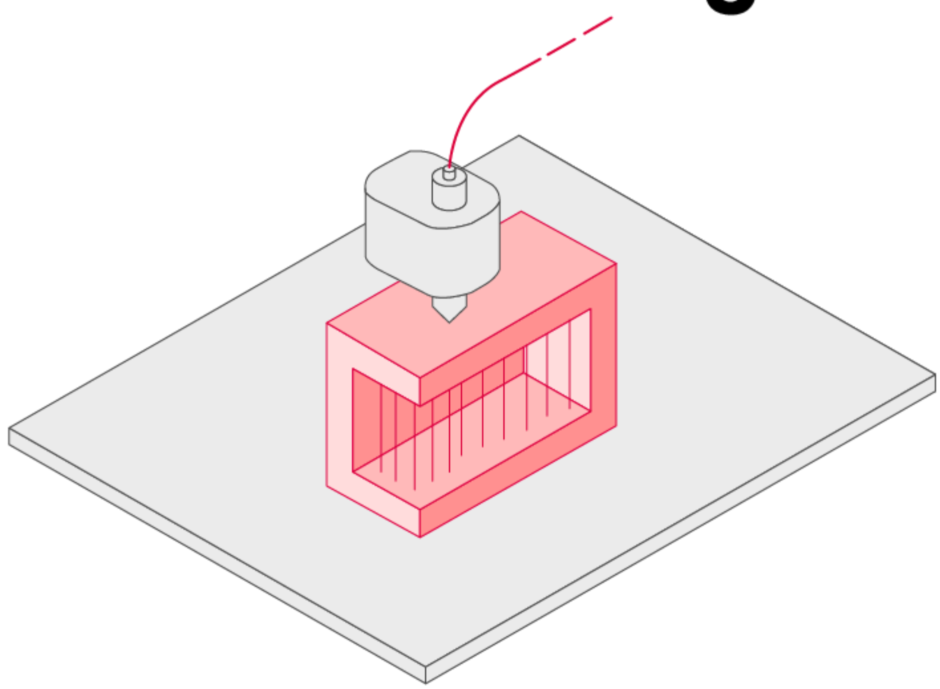
4. Printing the Model *(performed by LAB technicians)*

After a model has been prepared, sliced and approved by a D+TL Technician, it is assigned to one of our printers.

Take a look at the LAB’s 3D printing detailed video of file setup for 3D printing.

A 3D Printed Articulated Kuka Robot arm.

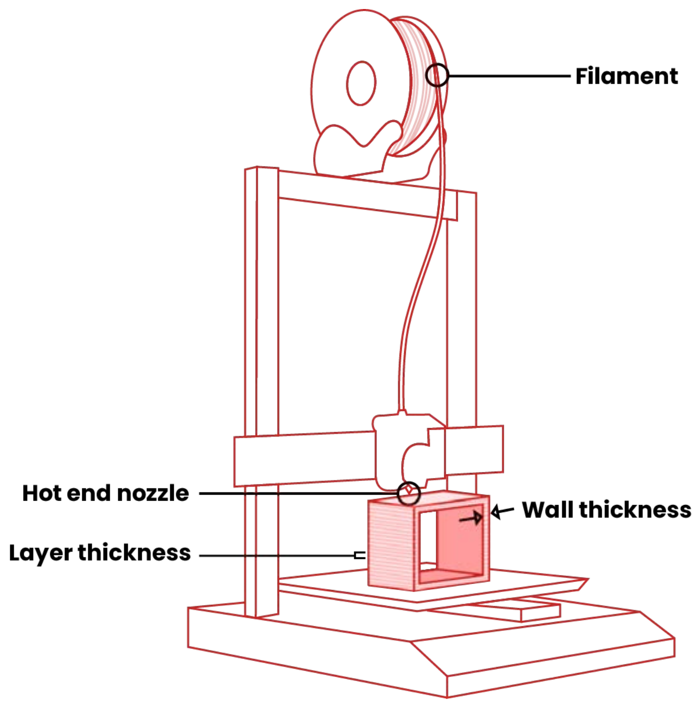
FDM Printing



How do FDM Printers work?

Fused Deposition Modelling (FDM) printing works by building up melted, extruded plastic, layer by layer. Though many kinds of 3D printing filament exist, **the Service Bureau only prints in PLA**. PLA is known as a user-friendly filament, due to its high print success rate & relatively low temperature requirements. If you'd like to experiment with other types of filament, please reach out to the LAB.

Simplified FDM 3D Printer Diagram



Simplified diagram of an FDM 3D printer

Filament: The PLA plastic filament used for printing.

Hot end nozzle: The hot end nozzles at the LAB have a diameter of 0.4mm. This dictates the thickest possible layer height.

Layer thickness: The layer thickness or layer height refers to the height of each layer in mm. For more info, see: Layer Height.

Wall thickness: The wall thickness refers to the thickness of the surface of the printed object. For more info, see: Wall Thickness.

FDM Slicing Software

FDM Printing is typically favored for its affordability, speed, and ease of color customization. This makes it ideal for both prototyping and final pieces.

In order to print your models, 3D printers need to follow specific directional instructions called *toolpaths*. **3D slicing software** is the tool which provides these directions to the printer by interpreting the array of polygons from your .STL file into printable, layered toolpaths.
They can also provide estimates for the **duration of 3D printing time** and **the amount of material** that will be used.

Bambu Studio Logo
(in D+TL Red)

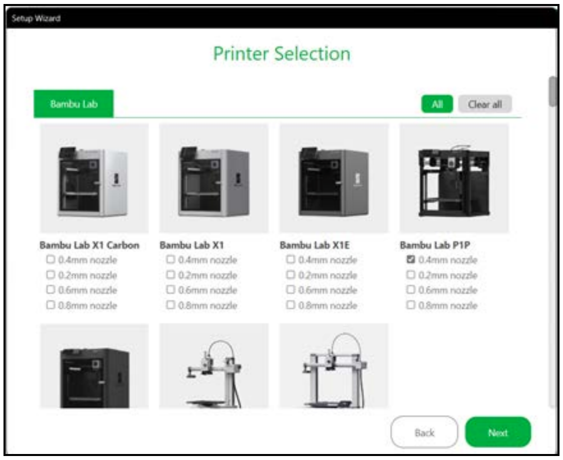
The Design + Technology LAB's recommended slicer for FDM printing is **Bambu Studio** (it's free, so you can download it, too!). The D+TL previously used Ultimaker Cura (also free) and you can still find an archive of those instructions at the end of this guide; however, they will no longer be updated.

Previewing your prints with a slicer allows you to confirm:

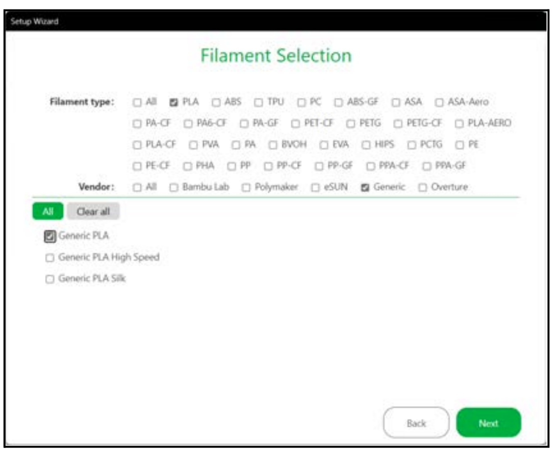
- **Overall Printability:** Receive helpful tips & notifications for various common errors you may encounter, such as exceeding the bed dimensions, non-manifold edges, and more. Importing your model into slicing software will also allow you to preview your model and ensure that the resulting print matches your intended design. This includes seeing if small details will successfully print.
- **Print Orientation:** Plan & preview your model's orientation for a more successful result. This includes placing flat areas flush with the print bed, optimizing for less support considering how the orientation can impact the overall strength of your print (for more info, see: Layer Orientation).
- **Scale:** Ensure that your model is at the correct size and within our printers' max. dimensions of **256mm x 256mm x 256mm**. You'll be able to resize/cut the object in the slicer, if necessary.
- **Detail:** The overall resolution of your print, determined by the thickness of each printed layer.
- **Support Generation:** Slicers automatically generate supports and offer a visualization of where they will be applied, as well as selection of the type of support. This can be helpful when planning for post-processing or when designing the model & optimizing for less required supports.
  Unless otherwise specified, D+TL staff will select the support type that we deem most appropriate on a per-model basis.

- **Time & Weight Estimates:** Slicers will present an estimated print time and material use. While these are not always perfectly accurate, they are useful guides for estimating costs and time to completion.
- **File Format:** Bambu Studio can be used to convert various 3D file formats into .STLs to submit to the Service Bureau.
- **Model Repair*:** Bambu Studio has a built-in model repair service that can very roughly fix a range of model issues. It is recommended that you look closely at the "fixed" model to ensure it is still representative of what you are looking for.
  *only available on Windows devices

Getting Started with Bambu Studio



"Printer Selection" menu. "Bambu Lab P1P" option is selected (step 1).



"Filament Selection" menu. "Generic PLA" is selected & the filters are active (step 2).

Bambu Studio is a free slicer **for FDM prints**. Upon downloading the software, it will prompt you with the Setup Wizard, which will guide you through the following steps:

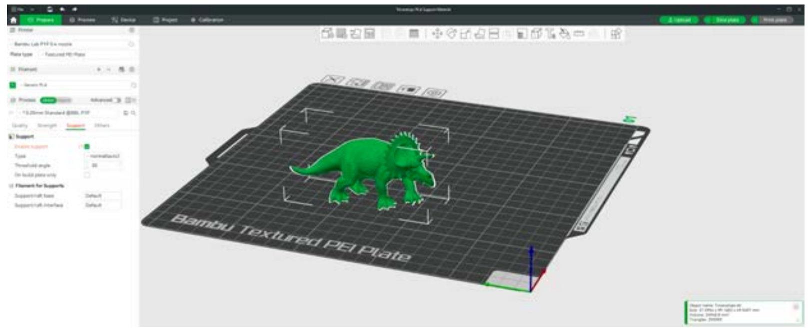
1. On the "Printer Selection" screen, click "Clear all" and select only the **Bambu Lab P1P** w/ the **0.4mm** nozzle.
2. On the next screen, "Filament Selection," click "Clear all" again, and select only "**Generic PLA**".
   TIP: use the filters - deselect "All" &, select "*PLA*" as the Filament Type and "*Generic*" as the Vendor.
3. The Setup wizard will prompt you to install the Bambu Network plug-in - this is not necessary at this time, unless you have your own printer, as this is for remote printer set-up and control.
4. Click "Finish."

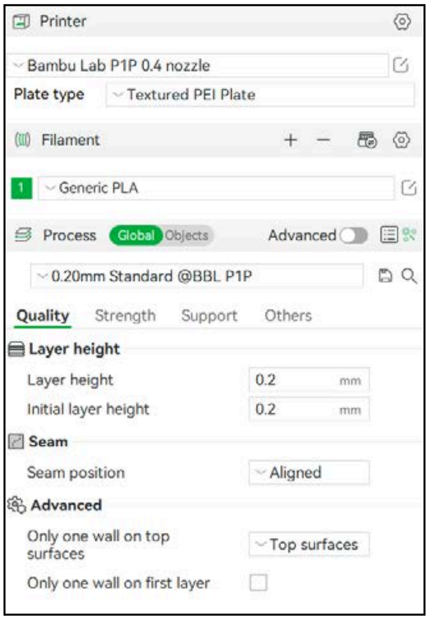
Once you've done this initial set-up, you can begin importing your 3D models for everything from time & cost estimation to model repair & modification.
The printer & filament selection can also be revisited as needed when visualizing your model to add/remove printers & filaments.

If you experience any issues throughout, reach out to the LAB, come visit us during our Open LAB Hours, or consult online resources. Bambu has fortunately built a strong online community of makers who often have answers to both common & rare issues (Bambu Lab Wiki, Bambu Lab Community Forum, r/BambuLab, and more).

File Setup in Bambu Studio

Models can be imported by using the **top menu bar**, the **toolbar**, or the **shortcut** [Ctrl/Cmd + I]). You'll then be able to manipulate it as needed and then slice it for previewing.





Left-hand menu.

In the left-hand menu, there are three sections: **Printer**, **Filament**, and **Process**.

In "**Printer**," you should have "**Bambu Lab P1P 0.4 nozzle**" selected. Plate type can be ignored as this is only a required selection when printing.

In "**Filament**," you should have "**Generic PLA**" selected.

If this is not the case, consult the "Getting Started with Bambu Studio" section.

In "**Process**," is where you can make all the necessary adjustments to your printing settings.

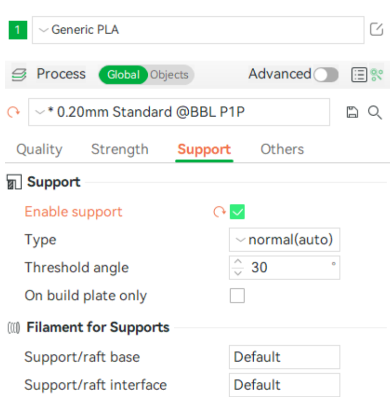
The D+TL's default process setting is: **0.20mm Standard @BBL P1P with supports enabled**.

To select:

1. Navigate to the drop down menu and select **0.20mm Standard**.
2. In the Process tab, click on "Support", and check the box beside "**Enable Support**." *normal(auto)* is the default selection, but you can also choose *tree(auto)*.

   The different types of support structures each have their appropriate use cases, and D+TL staff select the most effective support type for your project, unless specified otherwise. Be advised that certain models may not print successfully using certain kinds of supports.

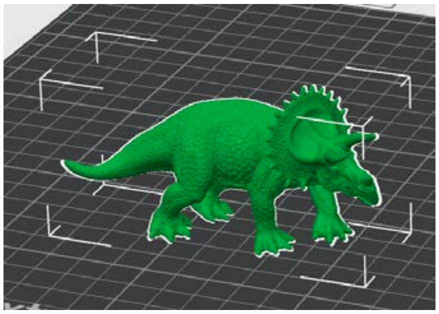
   To learn more about supports, see: Support Structures or visit the Bambu Wiki.



"Process" tab with 0.20mm Standard preset and normal supports enabled.

Estimating Costs & Previewing Your Print

Once you have the proper settings selected, click on "**Slice Plate**" in the top right bar. If you'd like to edit your model in Bambu Studio before slicing, proceed to the Beyond Previewing section.

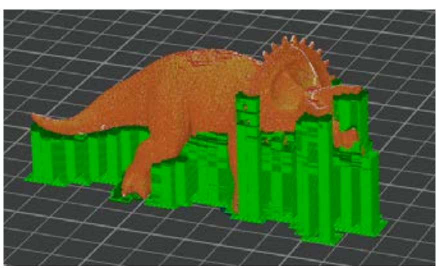


Original model, un-sliced

Bambu Studio will then process your model(s) and generate a preview of the print, including the **total filament** used & the **print time**. For further information and a walkthrough of the slicing information, visit the Bambu Wiki.

Total Estimation		
Total Filament:	7.15 m	21.32 g
Model Filament:	4.18 m	12.48 g
Cost:	0.43	
Prepare time:	5m47s	
Model printing time:	1h23m	
Total time:	1h29m	

*The cost provided by Bambu Studio is not equivalent to the D+TL's cost.

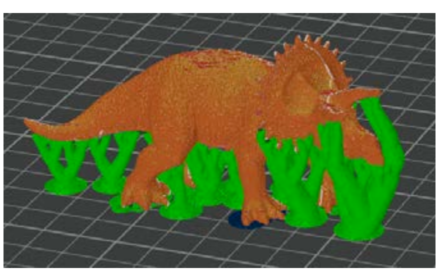


Sliced model with *normal* type supports
21.32g of filament
1h29min print time

The D+TL charges $0.15 per gram of filament, which includes your printed model(s) & any file set-up, consultation, & troubleshooting required for a successful print.

Total Filament in grams x 0.15 = **Print Cost Estimate**

e.g. this model with normal supports: **21.32** x 0.15 = **$3.20**



Sliced model with *tree* type supports
16.74g of filament
1h21min print time

The final weight of your print depends on a variety of factors such as support type, filament type, infill percentage, print resolution, and more. Your final cost will likely vary from the Bambu Studio estimate. It is intended to be used as a guideline.

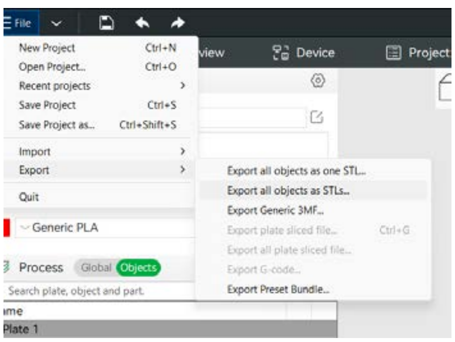
Different 3D printing settings may **increase/decrease print cost & time**. It is not always wise to go with the most affordable or quickest option as this may cause issues such as a lower quality print, difficulty in post-processing (for e.g. support removal), or even a failed print.

Exporting an STL File

Bambu Studio can also be a useful tool for obtaining an STL file to submit to the Service Bureau.

This can be especially helpful for:

- **Converting improper file formats**
  The D+TL only accepts STLs for all 3D Printing submissions, but Bambu Studio allows you to import .3mf, .stl, .oltp, .stp/.step, .svg, .amf, & .obj.
- **Scaling your model to the appropriate size**
  Using the "Scale" tool allows you to scale your model according to percentage ratios or to specific dimensions, down to the 100th of a mm.
- **Quick prototyping**
  e.g. using the "Cut" tool to quickly isolate a piece for a test-fit. *(For more info on the "Cut" tool, see the "Bed Dimensions Exceeded" section under the heading "Scale Issues" in our Common Issues webpage)*



Drop-down menu to export as an STL

Once you have your model prepared, navigate to "File" in the top left corner, go to "Export", and then "**Export all objects as STLs...**" **not** "as one STL."

This will ensure that, whether you have one or multiple objects, each one can be **submitted as its own file** (a requirement for the Service Bureau).

NOTE: this will only save the model's data (orientation, scale, etc.) and will not save any "Process" settings you may have selected (infill percentage, support type, etc.). These will need to be communicated to LAB staff in the Google Form under "*Special Requests*." The D+TL cannot guarantee the outcome of your request.

You can save your projects as .3mf files to preserve any settings and modifications you may have applied. You can also create your own Process Presets to streamline the slicing process for future models. **You cannot submit a .3mf file.**

Beyond Previewing

Navigation in Bambu Studio

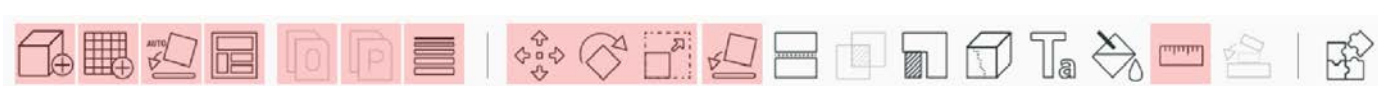
Left Mouse Button	Rotate view
Right Mouse Button **OR** **Mouse Wheel Button**	Pan view
Mouse Wheel/Scroll	Zoom in/Zoom out
Ctrl+Left Mouse Button **OR** **Cmd+Left Mouse Button**	Select multiple objects. Hold Ctrl/Cmd and click on each object you'd like to select.
Shift+Left Mouse Button	Rectangular selection of multiple objects.

Many familiar keyboard shortcuts (Ctrl/Cmd+C for *Copy*, Ctrl/Cmd+V for *Paste*, etc.) are applicable in Bambu Studio. Many others expand the functionality of the software, including shortcuts for the **toolbar**. See the Bambu Wiki for more in-depth information about Keyboard Shortcuts & 3D Scene Operations.

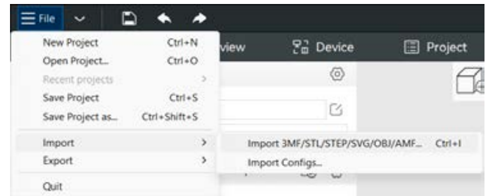
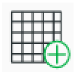
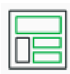
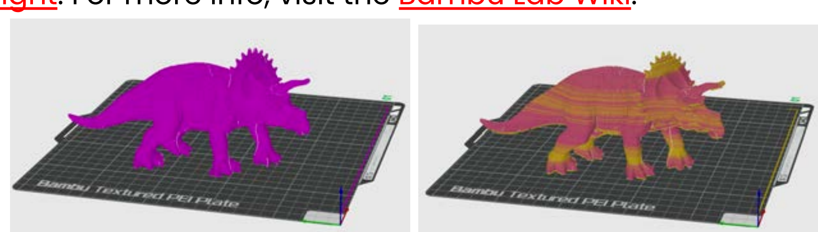
Toolbar Overview

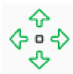
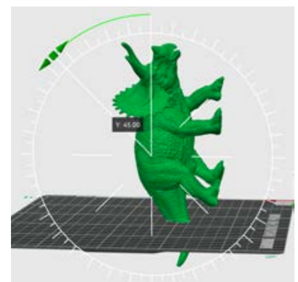
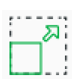
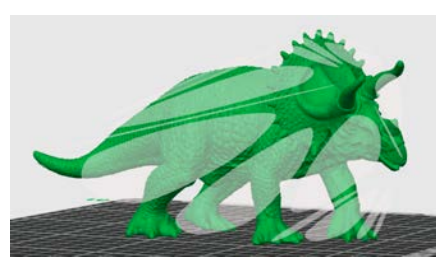
NOTE: tools along the toolbar will be greyed out when you don't have an object selected or when the tool cannot be used on the current object(s).

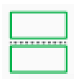
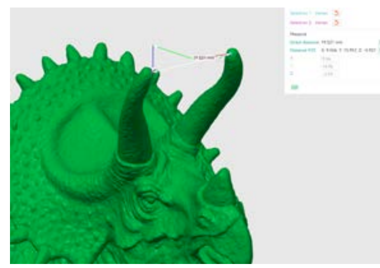
This guide will provide a *quick* overview of the following tools. **For a more in-depth explanation** of the toolbar and other utilities in Bambu Studio, **please visit their Wiki page**. We encourage you to do your own explorations into these tools and what works best for you!



TIP: hovering over buttons, fields, tools, etc. in Bambu Studio will give you its name as well as, oftentimes, a helpful description!

Icon	Tool	Shortcut	Description
	***Add***	***[Ctrl+I] or [Cmd+I]***	Adds models to your project. It can also be accessed through the top menu bar by navigating to "File," "Import," and then importing your file. 
	***Add Plate***		Adds another plate to your project - this is comparable to artboards in vector illustration software. This can be helpful when planning out multiple file submissions.
	***Auto Orient***		Automatically rotates your model(s) to place on the face which is most likely to lead to a successful print.
	***Arrange All Objects***	***[A]***	Automatically spaces out all your objects in the most efficient way possible while avoiding collisions/overlapping. This tool is only applicable when you have more than one object.
	***Split to Objects***		If there are multiple pieces in your project or file, this tool will automatically split them up into separate manipulable objects. Each object will need to be exported as a separate STL file to be submitted.
	***Split to Parts***		This tool only works when an .stl file with two or more meshes is imported. Objects can be made up of parts and can be manipulated using tools such as "Auto Orient" or "Arrange," but parts themselves cannot. For more information about splitting your models, visit the [Bambu Lab Wiki]().
	***Variable Layer Height***		Variable Layer Height is a tool which automatically detects detail levels in your model & applies the appropriate layer height. For more info, visit the Bambu Lab Wiki.  Consistent vs. Variable Layer Height

Icon	Tool	Shortcut	Description
	Navigation Tools		Each navigation tool, when selected, allows you to transform your model along three axes: X, Y, Z. With each tool, you can manually transform your model using the gizmo or enter specific values into the fields provided.
	***Move***	***[M]***	At nearly all times, Bambu Studio allows you to click & drag on your model(s) and free-move them in any direction. Using the "Move" tool however allows you to easily move your model to specific coordinates and/or along specific axes.
	***Rotate***	***[R]***	The "Rotate" tool allows you to rotate your model along certain axes & according to certain values. Clicking and dragging on any of the rotator gizmos is a **Free Rotation**. Align your cursor to any of the internal notches to **rotate in increments**. 
	***Scale***	***[S]***	The "Scale" tool allows you to scale your model according to percentage increases & decreases or according to specific measurements. You can also manually scale your model. **TIP: you can use the "Scale" tool to double check the dimensions of your model.** When using the tool, the measurements in the pop-up window beside "Size" correspond to your object's dimensions. Keep in mind that, by default, Bambu Studio & the D+TL operate **in mm.** **We recommend always enabling "uniform scale"** - this keeps the original size ratio of the model on all axes. Non-uniform scaling can result in warped/disfigured models. For more information, please see the "Unable to Scale Uniformly" section of our Common Issues guide.
	***Lay on Face***	***[F]***	"Lay on Face" is a manual version of "Auto Orient" where your model is split up into its printable surfaces (represented by the translucent, white, ellipses). Clicking on any of these 

Icon	Tool	Shortcut	Description
			shapes will automatically rotate your model to lay flat on the selected face. It's almost always best to orient your model on its most flat surface to ensure proper bed adhesion.
	*Cut*	*[c]*	The "Cut" tool can be used to ensure a model has a **perfectly flat printing surface**, to **split up larger models**, to **print a specific section** as a prototype, and more. For more information, visit the Bambu Lab Wiki or our Common Issues guide.
	*Measure*	*[u]*	The "Measure" tool allows you to measure different points of your model including edge length, diameter of circles, distance between two surfaces, angle between two lines, etc.  NOTE: we recommend only using the measure tool to double check your model or as a consulting tool as accuracy is not guaranteed.

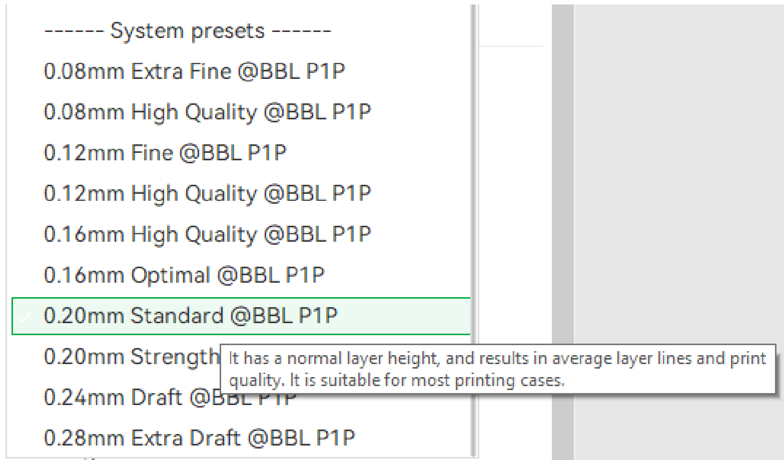
FDM Print Settings

Bambu Studio comes with built-in ***System Presets*** which affect the **layer height**, the **wall thickness**, the **infill density**, the **print speed** and more.

0.20mm Standard Setting

0.16mm Optimal Setting

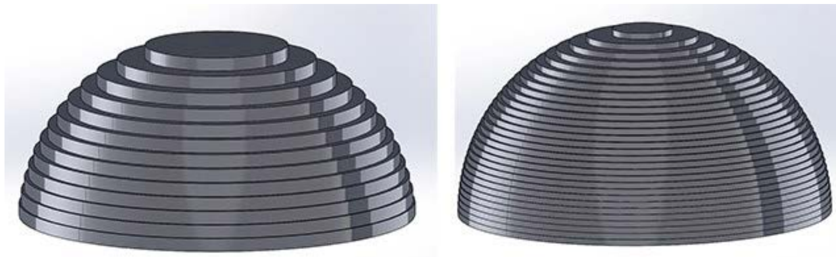
Hover over any of the presets in the dropdown menu to reveal a short description about which settings it affects as well as the intended results.



At the D+TL, the default preset is the **0.20mm Standard** with supports enabled. The majority of models print successfully & accurately at this layer height. For more detailed models, we typically recommend using the 0.16mm Optimal or High Quality setting, as any finer does not have discernible differences & increases the chances of failed prints.

Layer Height

The quality of a 3D print is impacted by the thickness or **height of each layer.** You can think about print quality like the "resolution" of an image. Low resolution is more "pixelated" - each layer will be more visible. High resolution is more detailed - the layer lines will be less apparent.



Generally, the more detailed a print is, the **longer its print time will be**. Certain kinds of models benefit more from a higher print quality: organic forms, rounded shapes, smaller prints, and models with intricate details for example. However, not all prints need the higher resolution.

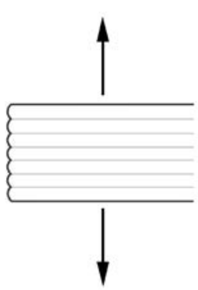
To modify the layer height in Bambu Studio, users can select the Process Setting preset with the quality they desire. Bambu Studio has calibrated a combination of settings which lend to a higher rate of success for your desired quality level, streamlining the slicing process.

The layer height and other settings related to print quality can also be manually adjusted under the "*Quality*" tab in the Process window.

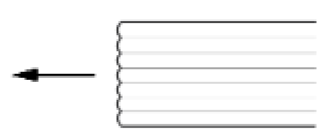
Layer Orientation

As FDM printers print layer by layer, prints tend to experience **weakness along the layer lines**. If the object will have force applied to it, it is important to consider the orientation of the "grain", as it affects the object's strength.

Force perpendicular to layers:
Part is weaker



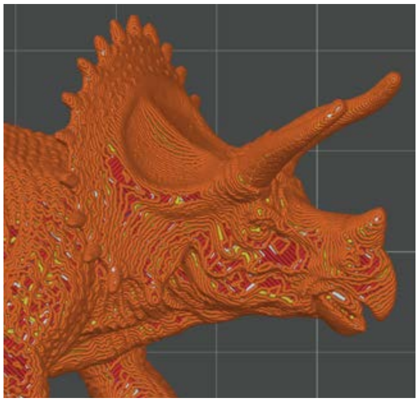
Force parallel to layers:
Part is stronger



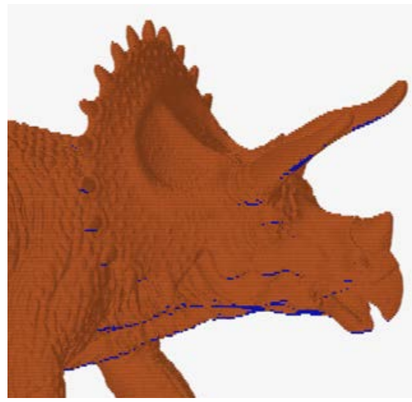
Similarly, any critical details should be oriented **parallel** to the build platform, as the printer can achieve higher levels of detail by stacking thin layers rather than within each layer.

For e.g., a model of a person should be placed standing upright rather than laying down in order to capture all the details of their face.

Critical details perpendicular to layers:
Details are lost



Critical details parallel to layers:
Details are preserved



When the triceratops' face is placed perpendicular to the layers (i.e. it is positioned laying down), many details of its face and scale texture are lost and the surface appears messy.

Part orientation also affects the surface finish, print time, support requirements, and more. For more information about part orientation, we encourage you to explore online resources.

Standard vs. Optimal Print

As mentioned, the D+TL generally uses the 0.20mm Standard setting. The 0.16mm Optimal setting is our default for prints that could benefit from a higher print quality. Let's look at some of the differences.

0.20mm Standard Setting

Weight with supports: 17g
Weight without supports: 12g

0.16mm Optimal Setting

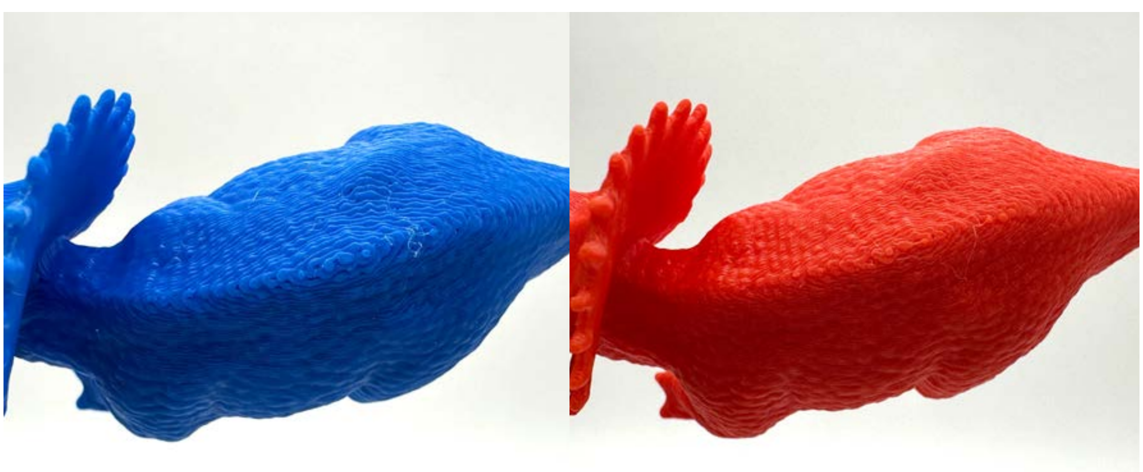
Weight with supports: 17g
Weight without supports: 13g

Both models generated very similar supports, but the *Optimal*'s finer layer height allowed smaller details to print without supports (for e.g. the end of the tail). Generally, higher quality prints will be heavier due to more densely packed layer lines.

0.20mm Standard Setting **0.16mm Optimal Setting**

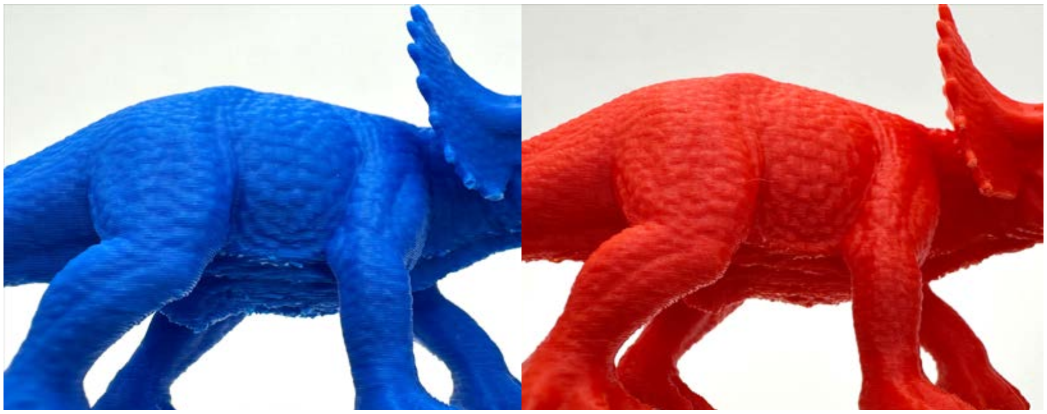
On the *Optimal* print, layer lines are less apparent, the texture is smoother (e.g. on its shoulder), and the facial features are sharper.

The *Standard* print's horns have clear layer segmentation, while the *Optimal* is smoother.
As well, the seam of the 3D print (where each layer starts & finishes) is much smoother (visible on the top right of each model).



The difference in layer heights becomes more apparent from a bird's eye view, as the *Standard* print has a more topographical appearance, while the *Optimal* print has a smoother, more cohesive finish.

0.20mm Standard Setting **0.16mm Optimal Setting**



Overall, both prints were successful and the level of detail your project entails is up to you. Keep in mind that there is only a difference of 0.04mm between these two examples and finer detail levels exist - with their own advantages & disadvantages.

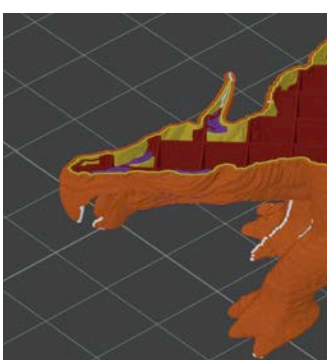
Wall Thickness

Wall thickness refers to the thickness of the surface of the printed object. While a thicker wall adds strength and rigidity to your model, it **does not impact the final size of the print.** All additional wall structures are added internally.

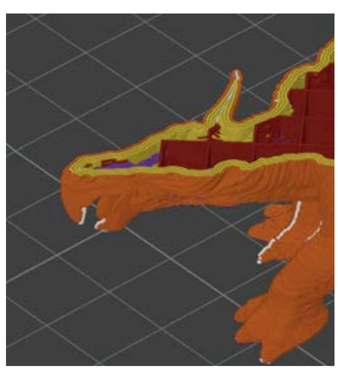
In Bambu Studio, this setting can be modified under *Strength* -> ***Wall Loops***.

The default setting for both *0.20 Standard* and *0.16 Optimal* is **2 wall loops**. This is the general required *minimum* for a successful print. Each wall loop is approximately the width of the hot end nozzle (in this case, 0.4mm). This means that the **walls of your model need to be at least 0.8mm thick** in order to be accurately sliced, but we recommend a **min. of 1mm**.

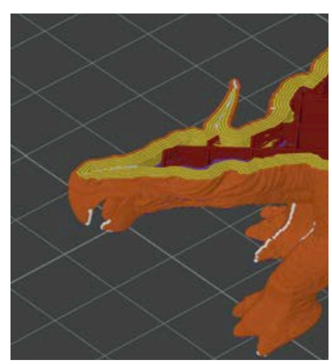
2 wall loops
D+TL default + recommended min.



4 wall loops

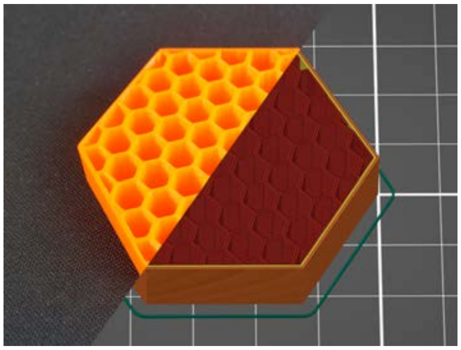


6 wall loops
recommended max.



2 wall loops is our recommended min., while 6 wall loops is the recommended max. For stronger prints, beyond wall loops, it would be best to modify the infill and/or orientation of your model.

Infill



Example of printed & sliced model with the "Honeycomb" infill pattern.

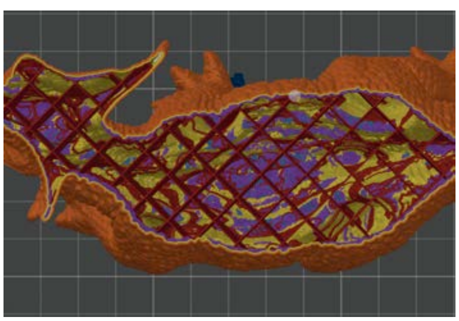
Infill refers to the automatic generation of 3D printing material to fill the empty interior void in your model, adding support between its walls. It may be an unseen interior structure but it plays a vital role in the overall strength of your print.

Any infill settings in Bambu Studio are under the ***Strength*** tab, within the ***Sparse Infill*** section.

Density

Generally, a higher infill density indicates a stronger print. However, this does not mean that every print should be printed at 100% infill. Surprisingly, this often tends to result in a weaker, more brittle print; it is the internal infill structures that grant the required flexibility to withstand pressure. As well, a print with a **higher infill density will take longer to print and weigh much more**.

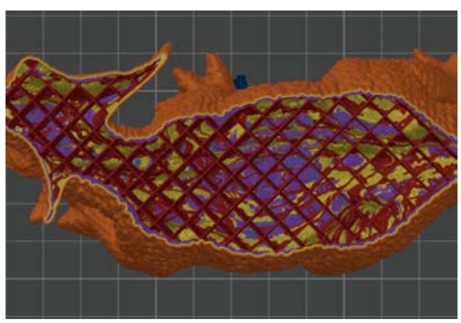
10%



110.12g
8h51m

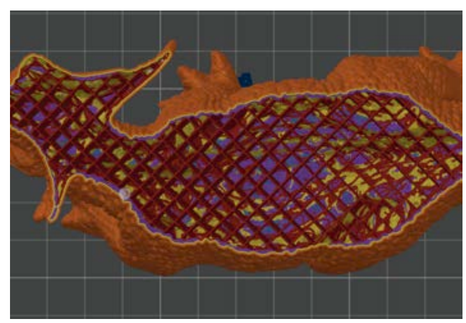
15%

D+TL Default Setting



121.01g
9h13m

20%



131.62g
9h34m

Density, weight, and print time comparison of the "Grid" infill pattern.

The recommended range of infill density is between **10-30%**, with an absolute maximum of 70% - anything over 70%, the cons begin to outweigh the pros & many infill patterns are unable to successfully print. By default, **the D+TL uses an infill density of 15%.**

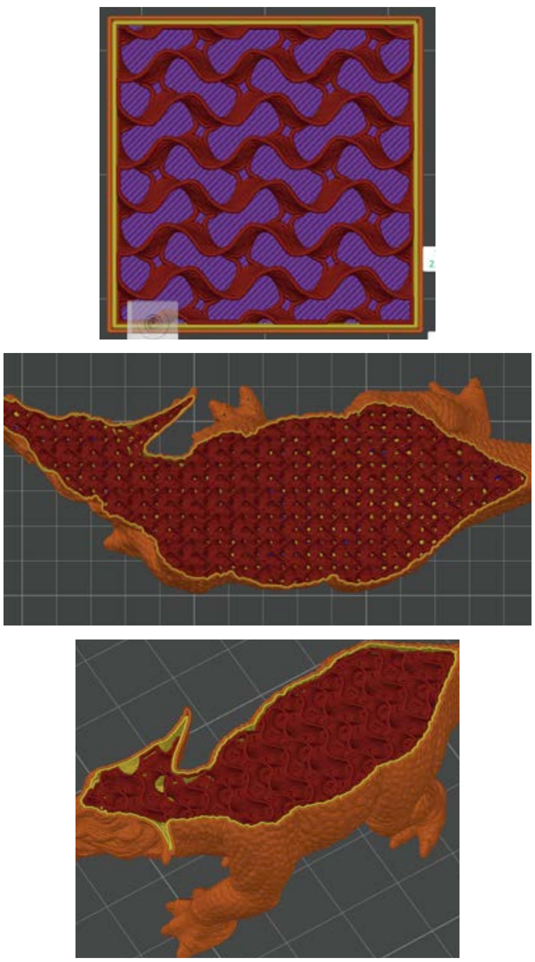
Type

The *type* of infill you use also has an **impact on the resulting strength** of your print, based on their geometry and printing patterns. Due to complexity & amount of material used, certain infills tend to **increase print time & final print weight**. Different infill patterns also have varying rates of success.

Certain infill patterns also encourage certain properties such as heat insulation, increased flexibility, and flotation. For more information regarding infill kinds and their advantages & disadvantages, consult the Bambu Wiki or other online resources.

Gyroid

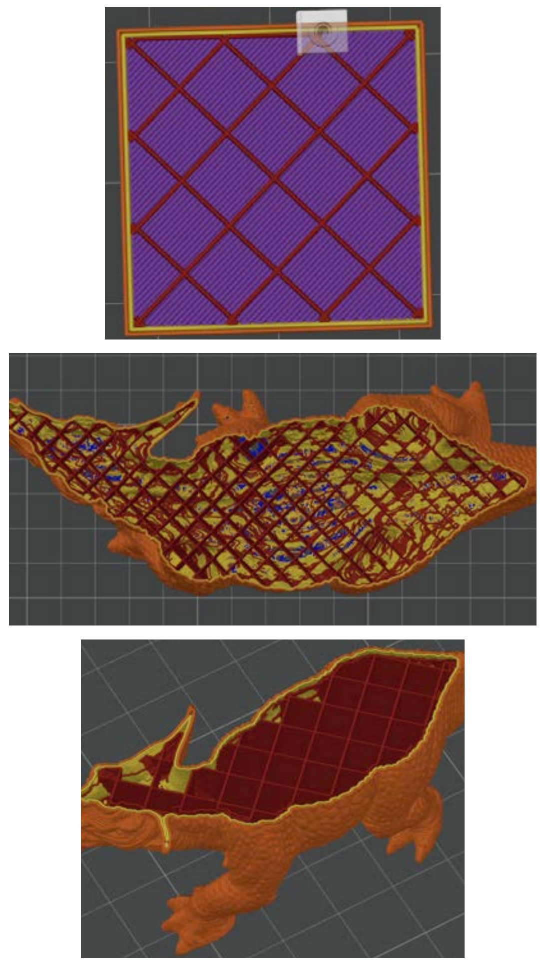
3D printing community favourite



Estimated weight: 119.10g
Estimated print time: 5h58m

Grid

D+TL Default Setting

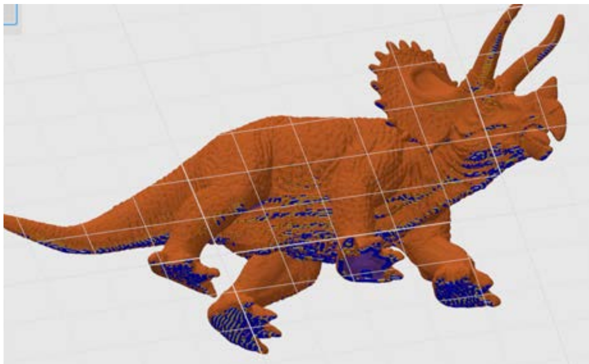


Estimated weight: 120.30g
Estimated print time: 5h47m

Infill pattern comparison at 15% density

Modifications to infill must be communicated to staff under the *Special Requests* section of the 3D printing submission form.

Support Structures

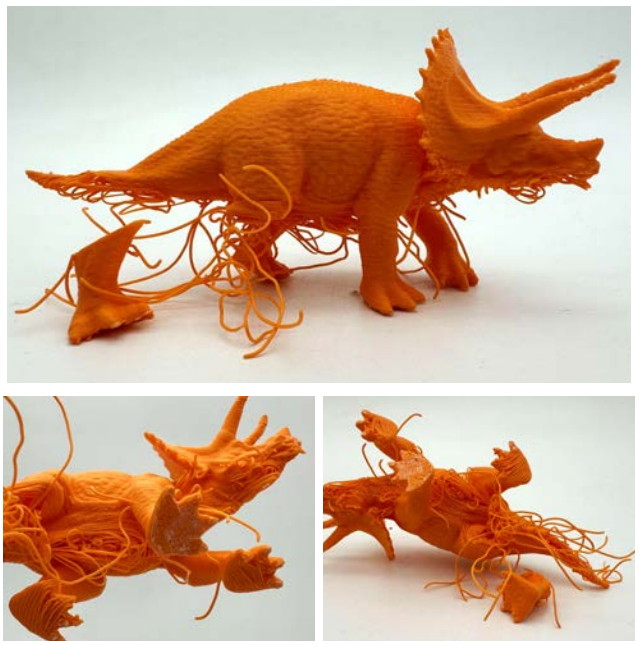


Sliced triceratops model, overhang sections shown in blue.

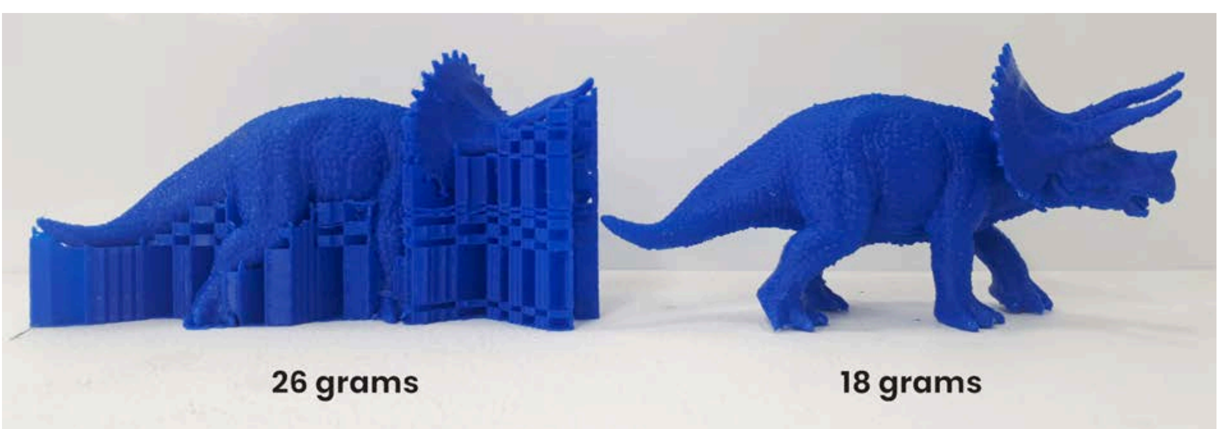
The FDM printing process involves heating plastic filament to a semi-liquid state in which it is unable to hold its shape well. This means that the underside and any overhangs are likely to droop or may fail to print altogether. To avoid this, slicing software **generates support structures** for us. In Bambu Studio, this is done in the *Process* window, in the *Support* tab, and by checking the *Enable Support* box (choose either of the *auto* settings).

Generally, supports increase the material weight and duration of a print, but they are essential for FDM printing success.

If you have a model which has very **many overhangs and/or internal cavities, consider printing in SLS** instead as this printing technique does not require the use of any support structures.



Example of print without support structures.



Weight comparison between an FDM print before and after support removal

NOTE: the final cost of your 3D printing job is calculated based on the weight **including** your support material.

Support Removal

Flush Angle Cutters

Supports can cause little imperfections in the object's surface and can be a hassle to remove. Supports can often be removed by hand - to help remove any stubborn or hard to reach support material the LAB recommends using "**flush angle cutters**." Any further blemishes can be filed, sanded, or otherwise masked in post-processing.

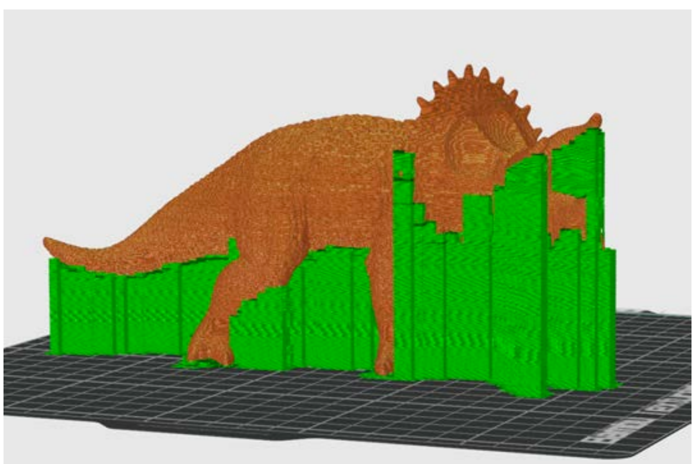
Come visit us during our Open LAB hours for support in removing your 3D printing supports!

Support Types

Within the *Support* section, there are two main types of support structures:

Normal Type

Recommended for any simpler, geometric/angular prints with larger & planar overhangs.



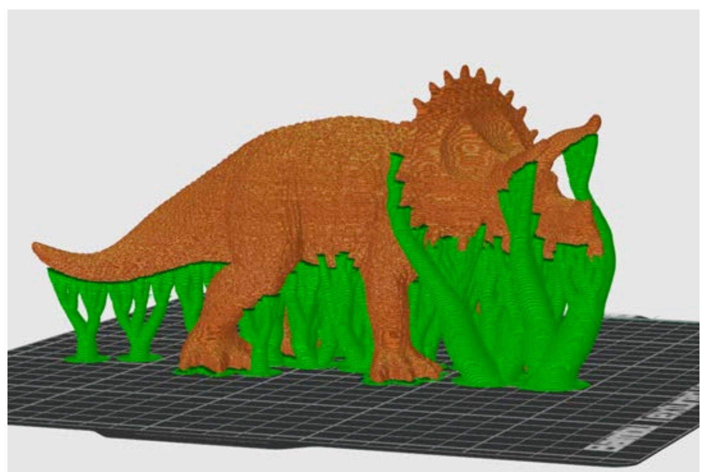
Estimated total weight: 118.65g
Estimated model weight (no supports): 71.19g
Estimated print time: 5h44m

Normal supports lend to:

Higher success rates
More even surface results
(model dependent)

Tree Type

Recommended for complex, more organic-looking objects where most of the overhangs are small and non-planar.



Estimated total weight: 93.01g
Estimated model weight (no supports): 71.19g
Estimated print time: 5h10m

Tree supports lend to:

Faster prints
Lighter total weight
Easier removal of supports

Depending on the model, its complexity, and the resulting generated support material, the **print time and weight can vary** greatly between support types. Each support type also has further customization options that we encourage you to explore. For more information regarding Supports, visit the Bambu Wiki.

Each support choice has its pros and cons, varying from model to model. Staff at the D+TL have developed best practices for choosing the appropriate support settings to ensure the success of each print. However, if you have a specific support requirement, submit your modifications under the *Special Requests* section of the 3D printing submission form.

Print Speed

Though the printing speed can be modified by the user under "*Speed*," staff members only perform speed modifications when attempting to troubleshoot failing prints - typically by slowing down the printing speed. Adhering to Bambu Studio's pre-tested printing speeds helps to achieve more accurate print time estimates as well as a higher success rate.

Otherwise, the print speed is chiefly **impacted by the rest of the slicing parameters.** For e.g., smaller layer heights, denser infills, certain infill types, etc. could increase the printing time.

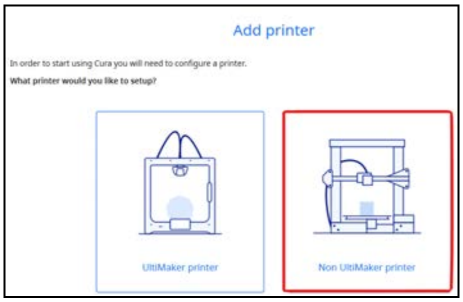
FDM Post-processing Tips

There exist so many possibilities for the post-processing of an FDM print. We encourage you to do your own research online to find the techniques best suited to you and your intended results. Some possible options include: sandpaper, spray paint, jewelry polishing tools, epoxy filler, UV resin coating, and more (as well as any combination of multiple techniques). However, it is also very common to clean up the print - remove supports & any filament remnants - and display as is!

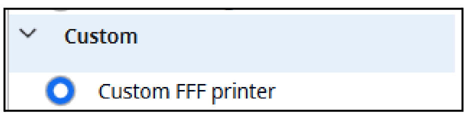
Feel free to visit us during our **Open LAB hours** for more support with any aspect of the 3D printing process, be it **3D scanning, model making, slicing, printing, post-processing** or any other questions!

Getting Started with Cura (Archived 2024)

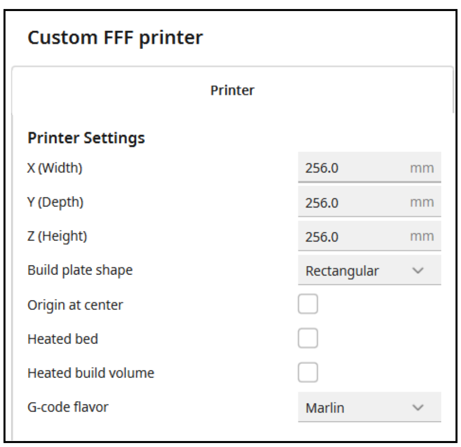
Ultimaker Cura is a free slicer **for FDM prints**. Upon downloading the software, it will ask you to Create an Account, this is optional and can be skipped using the icon at the bottom right of the page.



"Add Printer" menu. "Non-Networked Printer" option is highlighted (step 1).



"Custom FFF Printer" is selected (step 2).



Adding *virtual* printer settings on Cura (step 3).

Please note, the following section of this document is a guide to set-up the Cura software with a virtual printer for ***education and cost estimation purposes only***, not for specific printer parameters. Click here for a full Cura tutorial.

1. Under the Add a Printer Menu, select Add a Non-Networked Printer.
2. A menu will appear with printer models. Find the "Custom" category, select "Custom FFF printer".
3. Proceeding to the next step takes you to a printer setup page. Edit the X, Y and Z settings to match 256mm, so the virtual printer's scale matches the D+TL printers.
4. Click "next", and you're ready to get started in Cura. Congratulations!

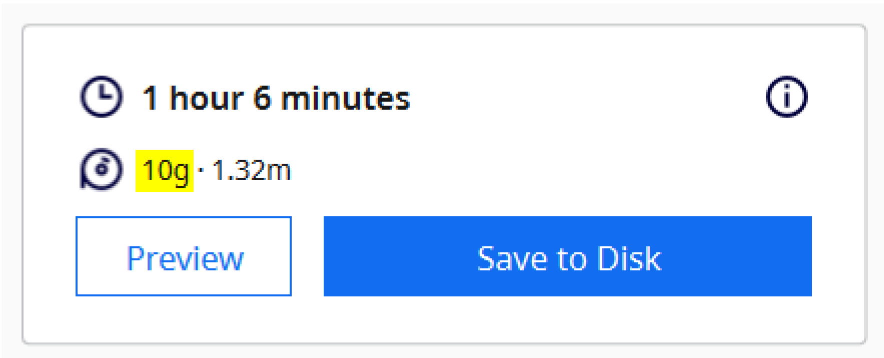
Once you've set-up your virtual printer, import your 3D model with *File > Open File(s)*. You should be able to view your model within the bounds of the virtual printing platform. In the top right side of the main Cura interface, you'll find a drop-down menu for various print settings. This is where you'll define the material quality of your print. Feel free to explore these menus and mess around with the values.

To explain some of the setting options, let's take a look at the pertinent parts of the printer and printing process ***(updated Summer 2024, see: FDM Print Settings).***

Estimating FDM costs in Cura

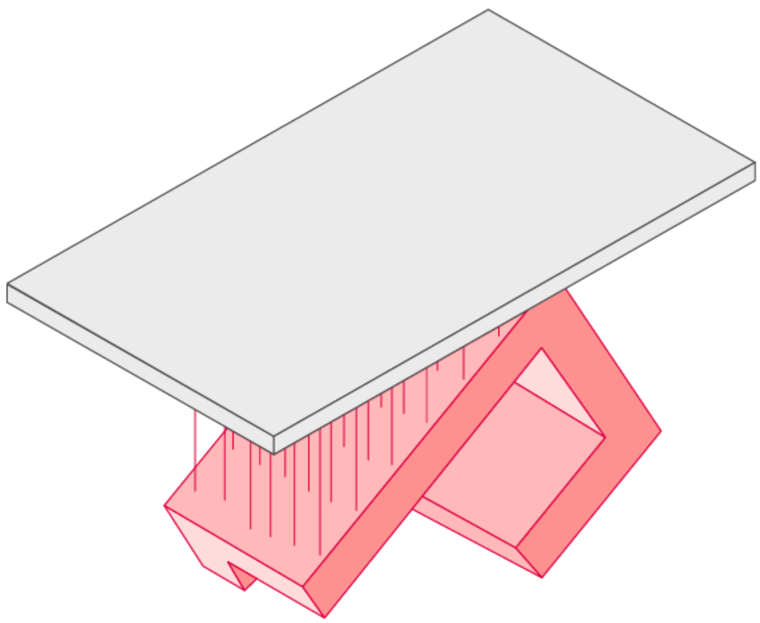
At the LAB, FDM prints are priced based on their weight. For FDM prints, it costs $0.15 / gram of PLA plastic.

Once you have oriented your model, added supports, assigned print quality and infill, Cura will calculate the approximate duration and weight of your 3D print.



NOTE (August 2024): this estimate is meant to be used solely as a consulting resource as Bambu Studio & Cura use different algorithm & model generation methods. Your final print weight & time may vary greatly depending on a variety of factors.

SLA Printing

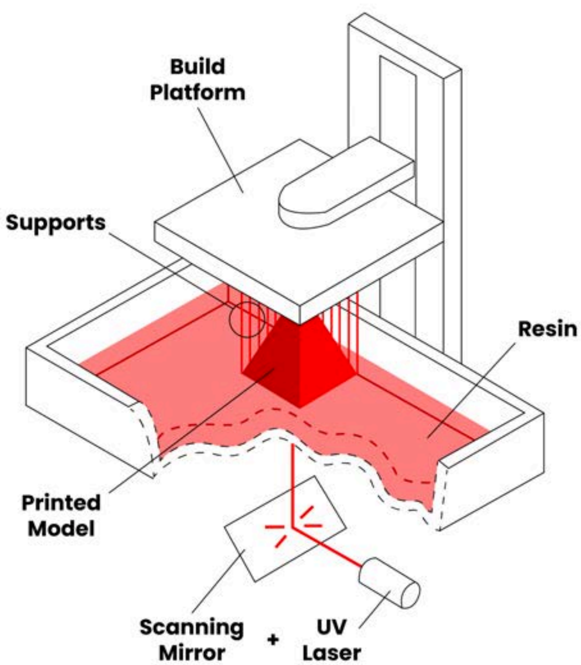


How do SLA Printers work?

Stereolithography (SLA) printing works by building up UV cured resin, layer by layer. At the LAB, we specifically use *inverted* stereolithography which means that, contrary to traditional 3D printing techniques (such as FDM Printing), the model(s) prints upside down. For each layer, the printer lowers the print bed into a pool of resin and uses a UV laser to selectively harden the resin in the shape of your model.

The LAB offers two kinds of resin printing mediums: standard clear and standard white.

Simplified SLA 3D Printer Diagram



Simplified diagram of an SLA 3D printer

Build Platform: This is where your model is adhered throughout the printing process. The platform moves up and down according to the slicing specifications, dipping into the resin pool for each layer. The size of the build platform and its maximum height determine the available printing area - at the D+TL, our max. dimensions are **145mm x 145mm x 193mm**.

Supports: Supports are structures which support your **printed model** throughout the printing process, ensuring its structural integrity (for more info, see: SLA Supports).

Resin: The pool of resin that the build platform dips into. When the print is complete, the build platform raises the object entirely out of the pool so that the excess resin can drain & drip off. All the unused resin returns to the reservoir and can be reused. To print in different kinds of resin, the entire reservoir needs to be emptied, cleaned out, and swapped.

Scanning Mirror & UV Laser: The scanning mirror helps to direct the laser beam and cure the intended section for the current layer. The resin is UV sensitive and solidifies wherever the laser touches it.

SLA Slicing Software

PreForm logo in D+TL Red.

SLA printing technology is typically favored for models that benefit from higher print resolution & quality.

In order to print your models, 3D printers need to follow specific directional instructions called *toolpaths*. **3D slicing software** is the tool which provides these directions to the printer by interpreting the array of polygons from your .STL file into printable, layered toolpaths.
They can also provide estimates for the **duration of 3D printing time** and **the amount of material** that will be used.

The Design + Technology LAB's slicer of choice for our SLA printers is PreForm. This slicer is *free*! The LAB encourages you to preview your SLA print in PreForm prior to submitting to the Service Bureau to get helpful information, such as:

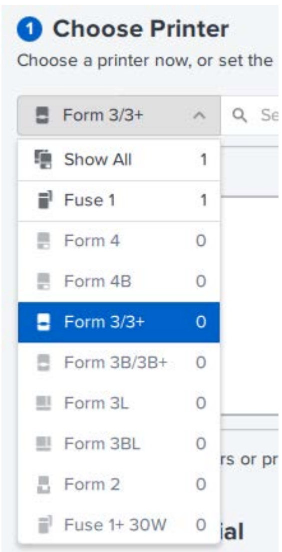
- **Quantity of detail:** the vertical thickness of each layer of the model being printed. This can help you visualize any areas that may be too thin to print successfully (for e.g. details smaller than 1mm may not print cleanly).
- **Support material:** Where support structures will be applied to ensure the adherence of model(s) to the platform and overall success of prints.
- **Estimated time required** to complete the print
- **Scale:** The maximum SLA printing volume is 145 x 145 x 193 mm
- **Print Orientation:** The orientation of your object in relation to the print "grain," any drainage holes, or support structures (see: SLA Printability and Hollowing Models)
- "**Printability**" errors with your model

Getting Started in PreForm

When printing using SLA technology, each consecutive layer is hardened as a solid layer. This means that the final overall 3D printed object is completely solid on the interior: there are **no wall thickness or infill settings**. This makes the slicing process more straightforward, compared to FDM.

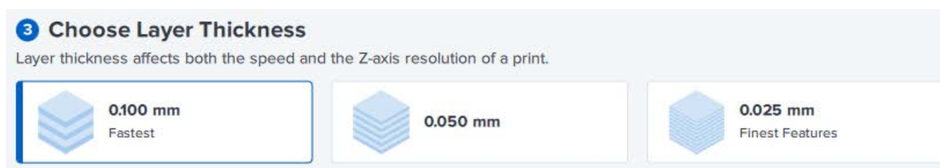
Set-up in PreForm is as simple as:

1. Choosing the printer type (the LAB uses **Form 3** for SLA)
2. Choosing the resin type. The LAB uses **"Clear"** or **"White"**.
3. Selecting your preferred layer thickness. The LAB default is **0.100mm**, material & model dependent. Consult the SLA Print Quality section for more info.
4. Hit "Apply."



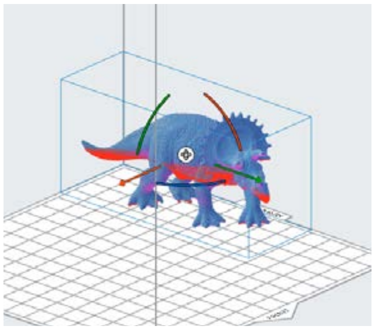
PreForm Printer Selection Menu.

Resin choices in PreForm.



Layer thickness options. The default D+TL settings can be found in SLA Print Quality.
Different materials will have different layer thicknesses available.

SLA File Setup


Example model on initial import.

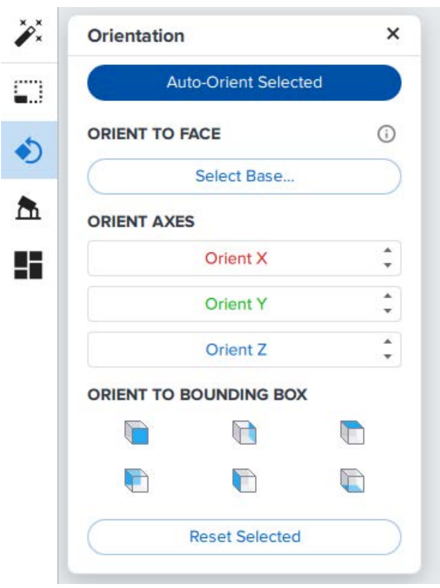
Once you've selected your printer, material, and layer thickness, you can import your model into PreForm for visualization and estimation purposes.

To import your model into PreForm, you can **drag & drop** your model, hit **[Ctrl/Cmd+O]**, or **navigate to the top bar** (*File -> Open*).

To navigate in PreForm, use...

- *Left mouse button*: to select and/or transform your model(s)
- *Right mouse button*: to rotate
- *Scroll*: to zoom
- *Scroll/middle mouse button*: to pan

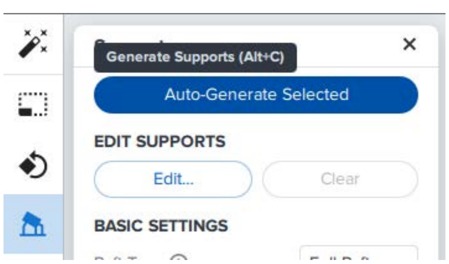
For more info on how to use PreForm, visit their website & consult their guide.



Orientation menu with "Auto-Orient Selected" highlighted.

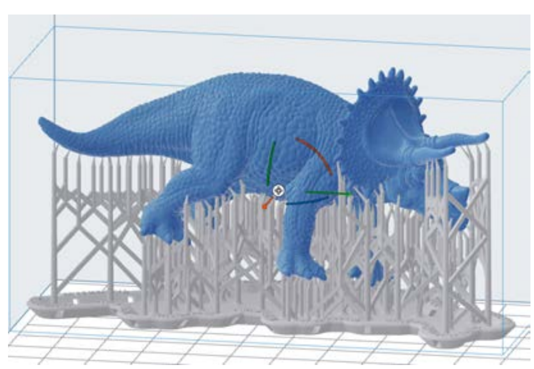
Using the left hand menu, prepare your model:

1. **Determine the orientation.**
   We recommend selecting your model and using the "**Auto-Orient Selected**" function. Through this tool, the slicer automatically determines which orientation will lend to a more successful print.
2. **Generate the support structures.**
   Similarly, once oriented, navigate to the Supports menu, and click "**Auto Generate Selected**." You don't need to modify any of the other settings in the Supports menu.

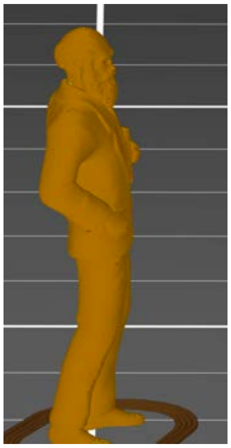


Supports menu with "Auto Generate Selected" highlighted.

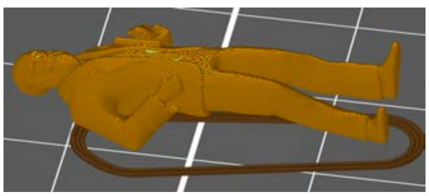
Once finished, you'll be able to visualize your model with its support structures.


Sliced example model.

A)



B)



Example orientation for best conservation of details (A) vs. loss of details (B).

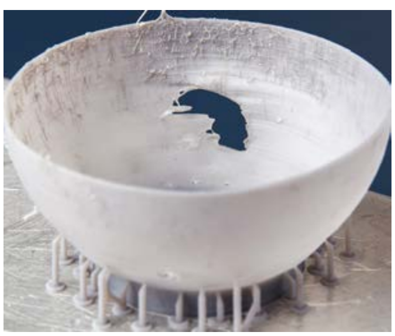
SLA Printability

3D printers build up prints in a series of thin *horizontal* layers. Make sure you keep this in mind when orienting your model: critical details should be oriented parallel to the build platform.

for e.g. a person should be printed standing upright rather than laying down to conserve details in the face.

Due to the physical nature of SLA printing, there is often the chance of your print "suction cupping" against the resin tank. This phenomenon occurs when there is a cavity or hollow portion of your object, creating a vacuum seal which can lead to print failure. Learn more about "suction cupping" here. To avoid this, use a 3D design software to either fill the hollow or add drainage holes to minimize suction during printing.

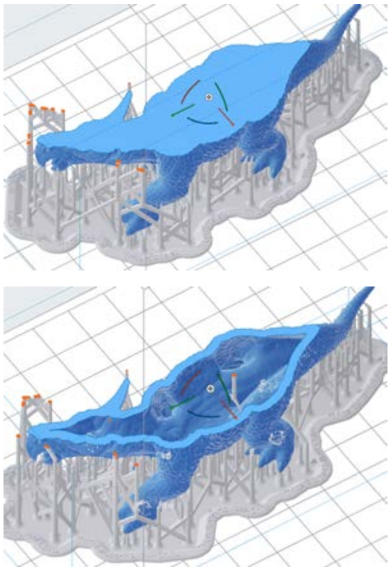
PreForm has a *Print Validation* tool, visible on the right hand side of the window, which double checks your model for various potential errors such as lack of support structure and cupping.



Example of blowout due to "suction cupping.

Print Validation window.

Check out this article for more helpful SLA design tips!



Example triceratops model, solid vs hollowed out. Notice the generation of internal support structures.

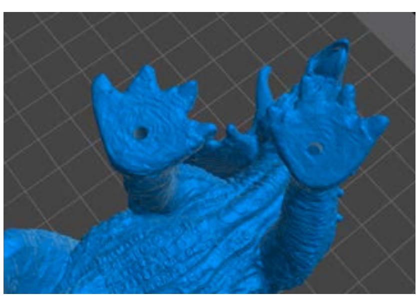
Hollowing Models

To design a hollow model, you can import your models into many free software applications such as Chitubox or PrusaSlicer. Through apps like these, you'll have access to various tools, including hollowing - you'll be able to empty your models, printing only the shell. Otherwise, **all SLA prints are solid objects** (100% infill). Many applications allow you to finetune their settings. We recommend a wall thickness of at least 2mm.

The main advantage to printing a hollow model is **saving on unnecessary material cost**. It also opens up the possibility of filling your model with other liquids, reduces required support structures, and more.

Print time & weight comparison

Solid model	**Hollow model**
Print time: 2h50m	Print time: 2h55m
Estimated Volume: 38.53mL	Estimated Volume: 26.65mL
Approx. Cost: **$17.34**	Approx. Cost: **$12.00**



Triceratops model with drainage holes in feet.

With any hollow print, you'll need to add **drainage holes** to ensure a successful print. Drainage holes prevent suction cupping, allow for excess resin to drain during printing, and allow for flushing of the inside of the model during post-processing. We recommend a min. of two drainage holes - one on the lowest side and the highest side of the model. Drainage holes must be placed **after** considering print orientation. Also, consider the placement of your drain holes and how they may affect the final result of your print. Ensure to communicate any orientation modifications or other notes in the *Special Requests* section of the 3D printing submission form.

PreForm (and other software) will often alert you to potential "cupping" instances and prompt you to reorient your model/add drainage holes. For more information about hollowing your model(s) and drain placement, we recommend looking into online resources such as this video, explaining cupping issues and drainage holes, or PrusaSlicer's blog post about hollowing.

Estimating SLA costs in PreForm

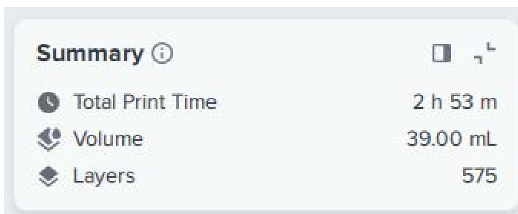
When a model is imported and support structures added, you'll be able to see the approximate **time** and **material** details of your print. These details can be used to estimate the cost of your print.

At the LAB, SLA prints are priced **based on their weight**. A LAB Staff member will weigh the print(s) after an alcohol wash, a UV cure, and with their supports attached. For SLA prints, the LAB charges:

$0.45 / gram of **standard resin.**

The PreForm software calculates the volume (mL) of resin that will be printed. FormLabs' resin weighs approximately 1 g per mL. You can use that approximation to estimate the print's total weight.

Total Resin in mL x 0.45 = **Print Cost Estimate**



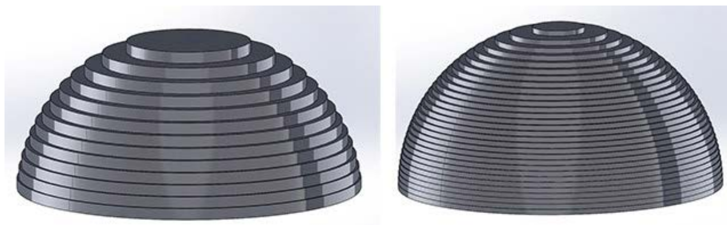
Example Print Summary.

Using the example, if printed in *clear resin*:
39.00mL x 0.45 = **$17.55 approx. cost**

The final cost of your print(s) will likely vary from this estimate. It is intended to be used only as a guideline.

SLA Print Quality

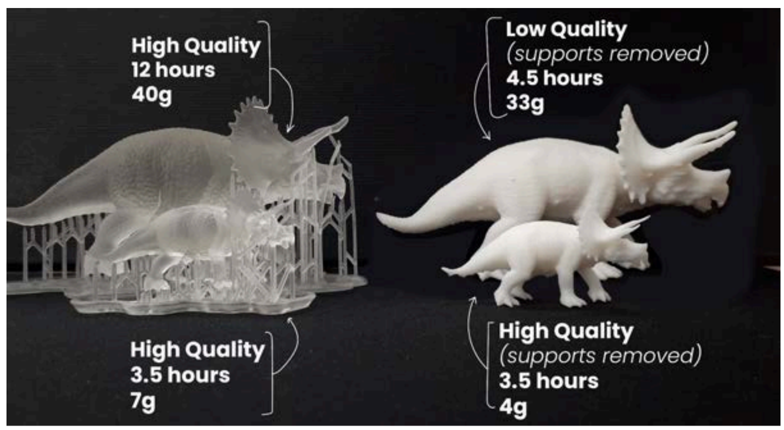
The quality of a 3D print is impacted by the **height of each layer.** You can think about print quality like the "resolution" of an image. Thicker layer heights are "lower resolution" or more "pixelated" as each layer is more visible. Thinner layer heights are "higher resolution" - more detailed with a smoother finish. SLA printers already produce a higher quality print (compared to FDM, for example) and as such, even the lower resolution settings will be considerably detailed.



Visualization of layer thickness comparison.

Changing the layer thickness does not affect the overall size of the print.

Generally, higher resolution prints take **longer to print** and may **weigh more**.



PreForm offers different layer height options according to the material you are using.

Material	Selection in PreForm	Layer Thickness Options			
		Adaptive	0.025mm	0.050mm	0.100mm
Clear Resin	"Clear"	✔	✔	✔	✚
White Resin	"White"	✔		✔	✚

✚ is the default layer thickness setting D+TL Staff use for each material. If you'd like to print with a different layer thickness, please indicate it to LAB Staff in the *Special Requests* section of the 3D printing submission form.

SLA Supports

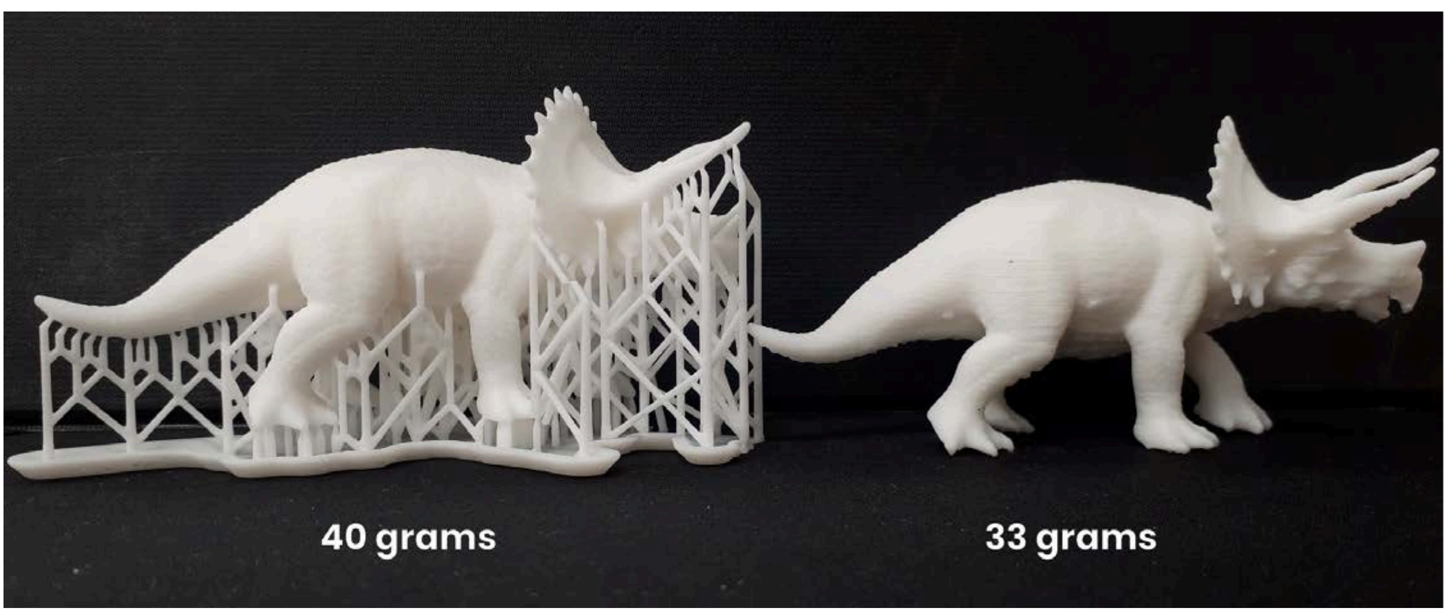
Flush Angle Cutters

SLA printers work by curing resin layer-by-layer and each new layer must be supported by the layer beneath it. If your model has overhangs which are not supported by a previous layer, there's a good chance it will print poorly. To avoid this, it is recommended to use additional 3D printed material called "supports".

Supports can cause little imperfections in the object's surface and can be a hassle to remove. Supports can often be removed by hand - to help remove any stubborn or hard to reach support material the LAB recommends using "**flush angle cutters**." Any further blemishes can be filed, sanded, or otherwise masked in post-processing.

Come visit us during our Open LAB hours for support in removing your 3D printing supports!

Generally, supports increase the material weight and duration of a print, but they are essential for SLA printing success.



Weight comparison between an SLA print before and after support removal.

If you have a model which has very **many overhangs and/or internal cavities, consider printing in SLS** instead as this printing technique does not require the use of any support structures.

SLA Post-processing Tips

There exist so many possibilities for the post-processing of an SLA print. We encourage you to do your own research to find the techniques best suited to you and your intended results.

At the D+TL, we do some preliminary post-processing for all SLA prints before they are ready to be picked up.

All SLA prints are given an initial alcohol wash in 99% isopropyl alcohol. The prints are then put into a curing station which emits heat and UV light. This helps to strengthen the prints, eliminate any residual tackiness, ensure dimensional accuracy, and more.

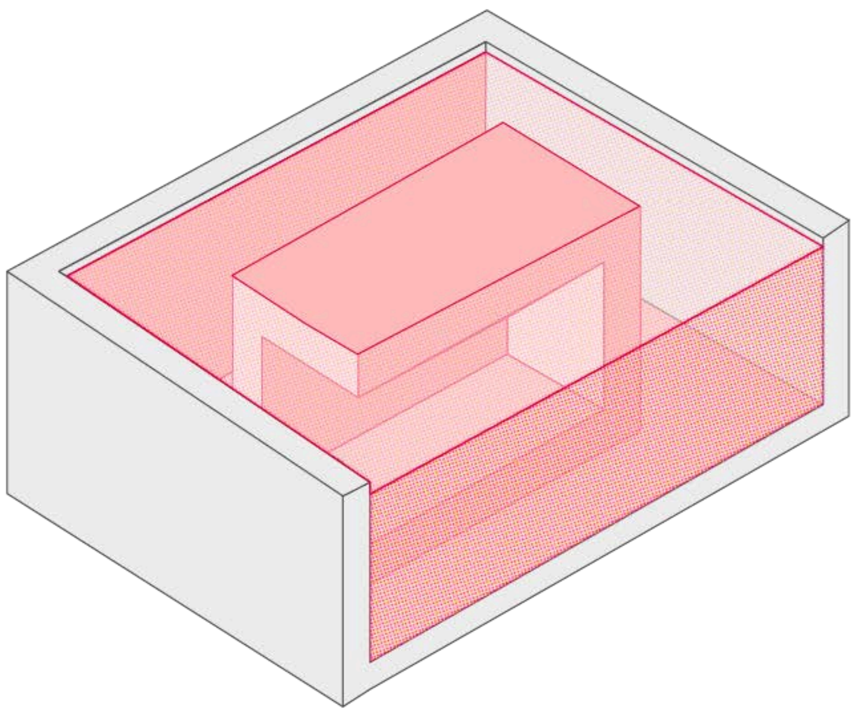
Comparison of various clear resins' resistance to UV exposure.

One of the benefits of the **traditional resin** materials - standard white and standard clear - is that they **can be dyed** using any alcohol based ink or synthetic fabric dye, such as Rit DyeMore dyes. As well, many techniques exist to emphasize the clarity of transparent resin prints, such as sanding or applying a clear gloss. NOTE: without a coating or other protection, **clear resin prints yellow** when exposed to UV light (e.g. sunlight).

Some other possible post-processing options include: sandpaper, spray paint, jewelry polishing tools, epoxy filler, UV resin coating, and more (as well as any combination of multiple techniques). However, it is also very common to remove supports & display as is!

Feel free to visit us during our **Open LAB hours** for more support with any aspect of the 3D printing process, be it **3D scanning, model making, slicing, printing, post-processing** or any other questions!

SLS Printing

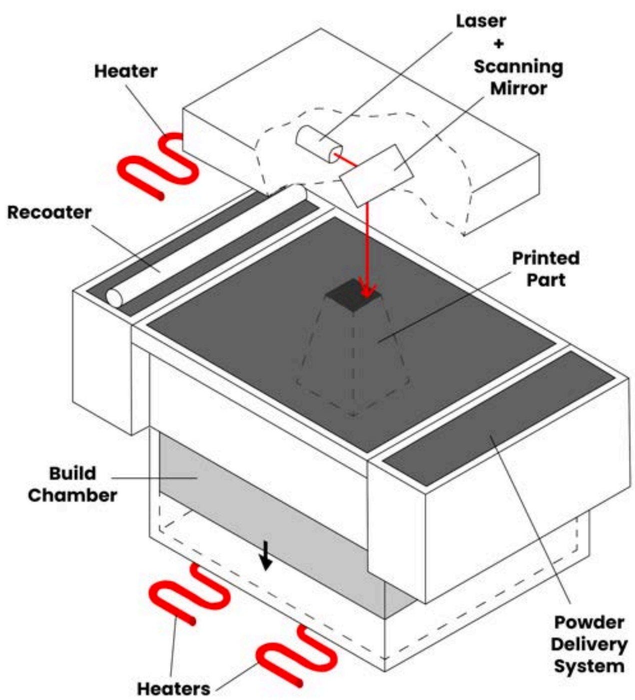


How do SLS Printers work?

Selective Laser Sintering (SLS) printing works by *sintering* - heating without fully melting - polymer powder together into a solid structure. The powder particles are selectively softened with a UV laser, layer-by-layer, fusing them together. Any unfused powder then supports the part during the printing process which means that, contrary to other 3D printing techniques (such as FDM or SLA), the model(s) print **without any support material**. All of the excess unfused powder can also be collected during clean-up & used again for future prints, making SLS one of the more sustainable 3D printing techniques.

The LAB only offers one kind of SLS printing medium: Nylon 12 Powder which comes in dark gray.

Simplified SLS 3D Printer Diagram



Simplified diagram of an SLS 3D printer

Recoater: the recoater spreads a *thin* layer of powder across the build platform, preparing each layer for sintering.

Laser & Scanning Mirror: these tools accurately target & heat specific areas, fusing the powder in those areas to form the desired 3D part.

Heaters: before printing, the heaters elevate the temperature of all the powder in the bin. This helps the UV laser be more effective by raising the temperature just below sintering. You must wait for all of the powder to cool entirely once the print finishes.

Build Chamber: the build chamber houses the build platform and raises & lowers it according to the printing instructions.

Powder Delivery System: also often called the "overflow bin," this compartment is responsible for housing excess powder that the recoater pushes across the printing area.

SLS Slicing Software

PreForm logo in D+TL Red.

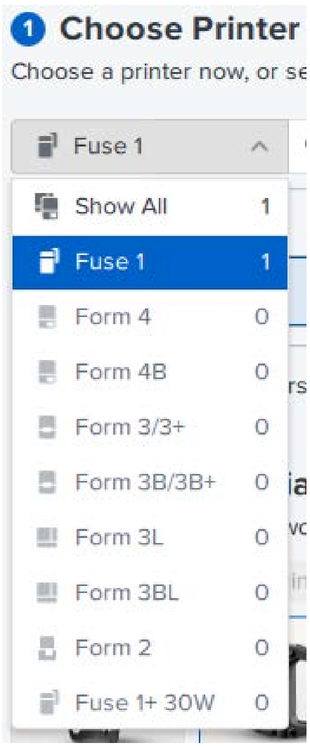
SLS Printing is typically used for functional parts & prototypes and favored for its durability.

In order to print your models, 3D printers need to follow specific directional instructions called *toolpaths*. **3D slicing software** is the tool which provides these directions to the printer by interpreting the array of polygons from your .STL file into printable, layered toolpaths.
They can also provide estimates for the **duration of 3D printing time** and **the amount of material** that will be used.

The Design + Technology LAB's slicer of choice for our SLA printers is PreForm. This slicer is *free*! The LAB encourages you to preview your SLS print in PreForm prior to submitting to the Service Bureau to get helpful information, such as:

- **Quantity of detail:** the vertical thickness of each layer of the model being printed. This can help you visualize any areas that may be too thin to print successfully (for e.g. details smaller than 1mm may not print cleanly).
- **Support material:** Where support structures will be applied to ensure the adherence of model(s) to the platform and overall success of prints.
- **Estimated time required** to complete the print
- **Scale:** The maximum SLS printing volume is 145 x 145 x 193 mm
- **Print Orientation:** The orientation of your object in relation to the print "grain," any drainage holes, or support structures (see: SLA Printability and Hollowing Models)
- "**Printability**" errors with your model

Getting Started in PreForm

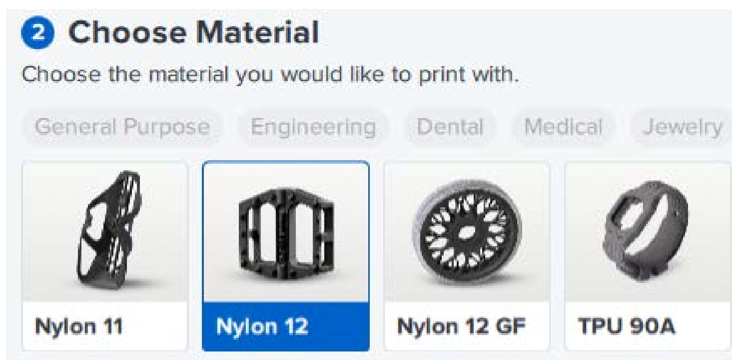


PreForm Printer Selection Menu.

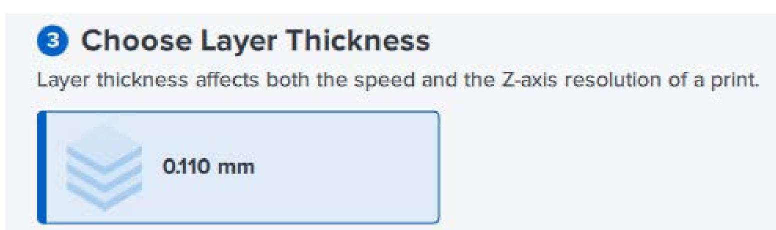
When printing using SLS technology, each consecutive layer is sintered with unfused powder used to support the part(s). This means that users need to indicate the final overall 3D printed object is typically completely solid on the interior, unless specified otherwise. : there are **no wall thickness or infill settings**. This makes the slicing process more straightforward, compared to FDM.

Set-up in PreForm is as simple as:

1. Choosing the printer type (the LAB uses **Fuse 1** for SLS)
2. Choosing the material. The LAB uses **"Nylon 12"**.
3. Selecting your preferred layer thickness. There is only one option for SLS prints: **0.110mm**.
4. Hit "Apply."



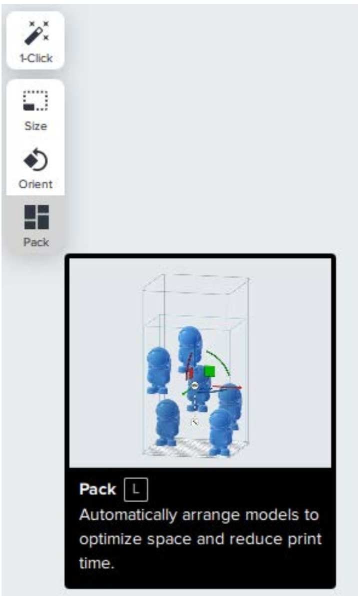
Material Selection in PreForm.



There is only one layer thickness option for SLS prints.

SLS Print Settings

Although other printing types are cheaper and the preferred process for most models, the SLS printing technology offers a **stronger, more durable** product than its counterparts.



"Pack" menu option in PreForm.

When printing using SLS technology, each consecutive layer is hardened as a solid layer, supported by loose nylon powder. This means that the final overall 3D printed object is completely supported and solid on the interior: there are **no wall thickness, infill or support settings**. This makes the slicing process more straightforward, compared to FDM and SLA.

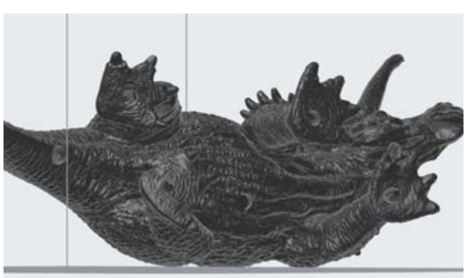
SLS Printability

Due to the lack of support structures & print settings, most of the printability of a model relies on the design of the part. Check out this article for helpful SLS design tips!

3D printers build-up prints in a series of thin *horizontal* layers. For higher success rates & lower costs: models should be kept in as horizontal of an orientation as possible. However, compared to FDM & SLA, this is less critical to consider for SLS prints.
PreForm offers a tool which automatically places your part in the most optimal printing position with the **"Pack" tool**.

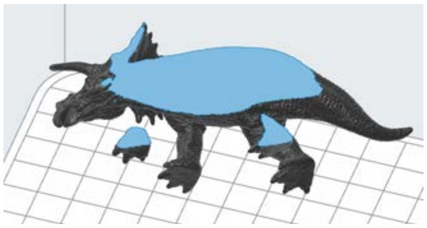
Hollowing Models

Hollowing an SLS model can help with lowering print costs while also reducing heat buildup and/or geometric deformations in certain models. There are two options for designing & printing hollowed prints: entirely hollow on the inside or with loose powder filling the internal cavity.

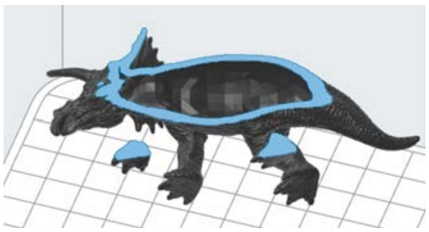


3D model with drainage holes.

If the model is entirely hollow, you'll need to place a minimum of **2 drainage holes** for the loose powder to escape during the printing process. The drainage holes must have a diameter of **at least 3.5mm**, or else they risk closing up during the printing process.



Example triceratops model, solid vs hollowed out.



Without drainage holes, all the internal loose powder will be sealed inside the model.

To design a hollow model, you can import your models into many free software applications such as Chitubox or PrusaSlicer. Through apps like these, you'll have access to various tools, including hollowing & drainage hole placing. Otherwise, **all SLS prints are solid objects** (100% infill). Many applications allow you to finetune their settings - we recommend a wall thickness of **at least 4mm**.

Feel free to visit us during our **Open LAB hours** for more support with designing hollowed SLS models and the placement of drainage holes!

Estimating SLS costs in PreForm

Once the model is imported & packed, you'll be able to see the time and material details of your print. These details can be used to estimate the cost of your print.

At the LAB, SLS prints are priced **based on their weight**. A LAB Staff member will weigh the print(s) *after* some post-processing. For SLS prints, the LAB charges:

$0.50 / gram of **sintered** powder and a **$7.00 processing fee**

The PreForm software calculates the weight (kg) of nylon powder that will be used. You will only be charged for the **sintered powder** (your model) as the rest of the loose powder can be recycled. You can use that approximation to estimate the print's total cost.

NOTE PreForm supplies the weight in **kilograms** - multiply the weight by 1000 to convert to **grams** before calculating the estimate!

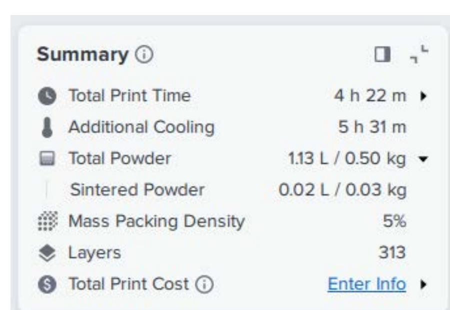
To calculate your estimate:

(Total weight in **grams** x **0.50**) + 7.00 = **Print Cost Estimate**

Using the following example:

(30**g** x **0.50**) + 7.00 = **$22 approx. cost**

The final cost of your print(s) will likely vary from this estimate. It is intended to be used only as a guideline.



Example summary window detailing print time & powder usage.

SLS Post-processing Tips

There exist so many possibilities for the post-processing of an SLS print. We encourage you to do your own research to find the techniques best suited to you and your intended results.

At the D+TL, we do some preliminary post-processing for each SLS print before it is picked up.

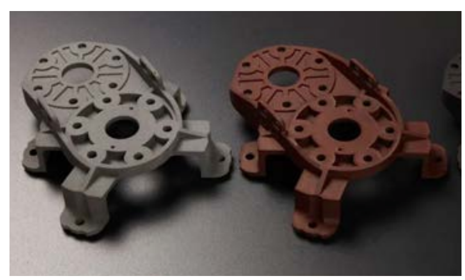
Example print before & after tumbling in the media blaster.

We start with an initial dusting + vacuum to remove any excess, unfused powder. This cleans the print and allows us to collect & reuse as much excess powder as possible. The print(s) are then put into a media blaster which shoots fine glass particles at the print(s) while it tumbles, further cleaning and polishing the print(s).

Your post-processing technique often depends on the part's intended use. To smooth & seal SLS parts, many creatives use vapor smoothing. This technique is especially useful as SLS prints are known to be porous, which can be an issue in some use cases.

SLS print vs. vapor-smoothed SLS print.

However, their porosity means that SLS prints are relatively easy to dye. Though industrial methods are recommended, it is possible to dye them using household synthetic dyes such as Rit DyeMore, as seen here (keeping in mind that the powder is already a dark gray).



SLS print vs. SLS print dyed with Rit Dye.

SLS prints can also be spray painted, ceramic coated, electroplated, and more. However, it is also very common to simply use them as is! You can start by visiting this Formlabs link and do more research online to see what best suits your desired outcome.

Feel free to visit us during our **Open LAB hours** for more support with any aspect of the 3D printing process, be it **3D scanning, model making, slicing, printing, post-processing** or any other questions!