

3D Printing Reference Guide

Spring 2022

The
Creative
School

**Design +
Technology LAB**



Table of Contents

3D Printing General Overview	3
FDM Printing	4
FDM Slicing Software	5
Getting Started with Cura	6
FDM Print Settings	7
Estimating FDM costs in Cura	8
FDM Quality vs Speed	9
Low Resolution/Fast Print	10
High Resolution/Slow Print	11
FDM Supports	12
SLA Printing	13
SLA Slicing Software	14
SLA Print Settings	15
SLA Printability	15
Estimating SLA costs in PreForm	16
SLA Quality vs Speed	17
Low Resolution/Fast Print	18
High Resolution/Slow Print	19
SLA Supports	20

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**.

The process of printing in 3D involves several steps:

1. You'll need a **3D model!** You can make your own in various 3D modeling softwares (Fusion 360, Rhinoceros, TinkerCAD, etc), or you can source models from online libraries like [thingiverse](#).
2. **Export** your 3D model as an .STL file. [See our tutorial documents](#) for exporting the STL in your preferred software.
3. Assign **print settings** to your STL with a “slicing” software, such as [Cura](#) for FDM prints, or [Preform](#) for SLA prints.
4. **Print!**

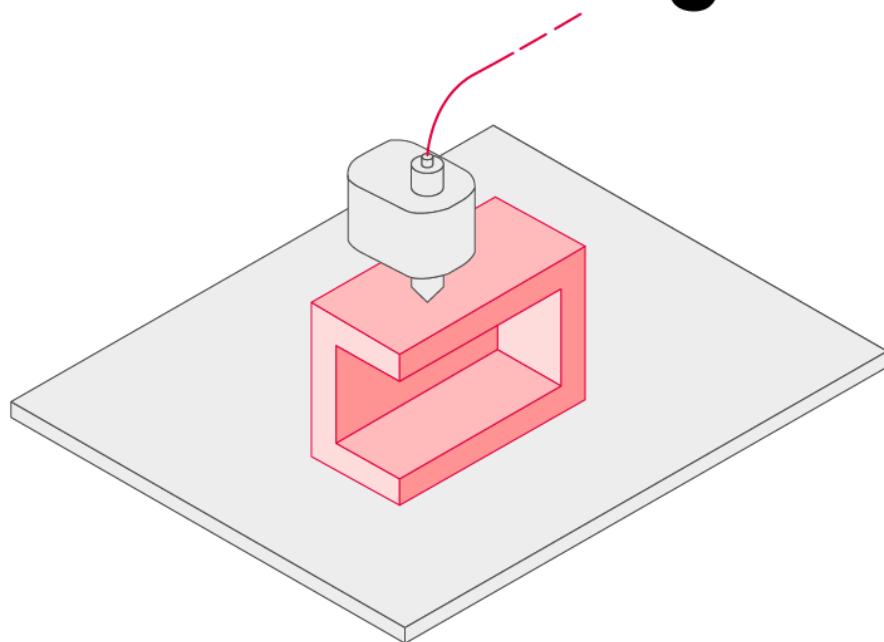
Select the best 3D Printing technology for your model size, geometry and budget. FDM printing is often less expensive and of a lower quality. Click here to learn more about [FDM printing](#) and [SLA printing](#).

[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



FDM Slicing Software

A 3D slicing software is the in-between step after modeling and before printing. The slicer interprets the STL's array of polygons into printable, layered toolpaths (toolpaths = specific locations where the machine travels and does its thing). The slicing software can estimate 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 FDM printers is [Simplify 3D](#). We use [PreForm](#) with our SLA printers.

If you want to preview your FDM print in a slicer, [try Ultimaker Cura](#) (it's free).

Previewing your prints with a slicer allows you to get helpful information, such as:

- Quantity of detail / resolution being printed
- Where support material will be applied
- Time required to complete the print
- Areas that will be too thin to print successfully (details smaller than 1mm may not print cleanly)
- Object scale (The maximum printing volume of the Lab's FDM printers is 300 x 300 x 400mm)
- The orientation of your object in relation to the print "grain" (see page 7)
- "Printability" errors with your model

Getting Started with Cura

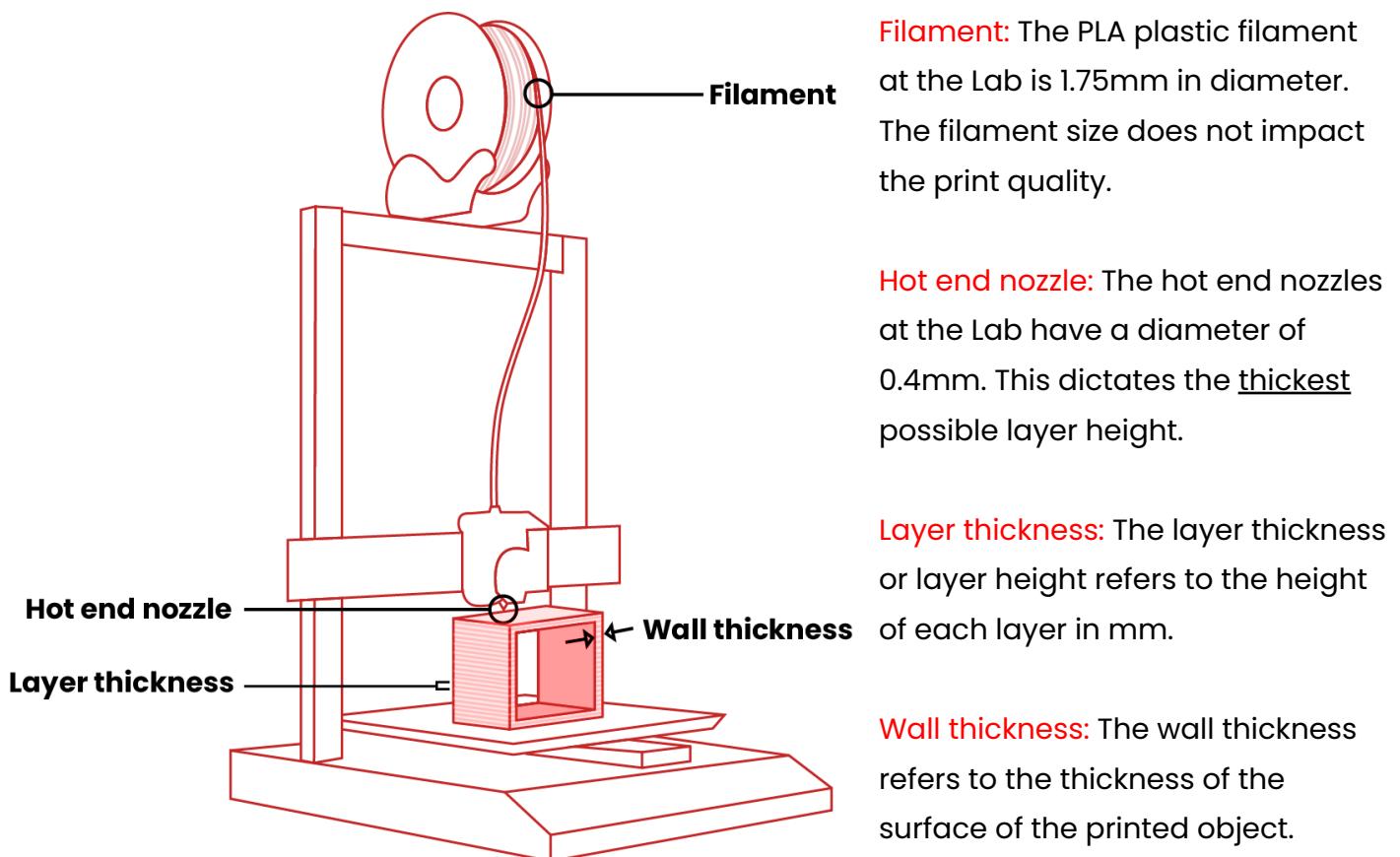
Ultimaker Cura is a free slicer **for FDM prints**. Upon downloading the software, it will ask you to Create an Account, this is optional and can be skipped using the icon at the bottom right of the page.

Under the Add a Printer Menu, select Add a Non-Networked Printer. A menu will appear with printer models. The Lab uses a printer called "**Artillery Sidewinder X1**".

[Click here for full Cura tutorial.](#) (Skip to section 3: Cura 3D Quick Start Guide)

In the top right side of the main Cura interface, you'll find a drop-down menu for different print settings. This is where you'll get to define the material quality of your print.

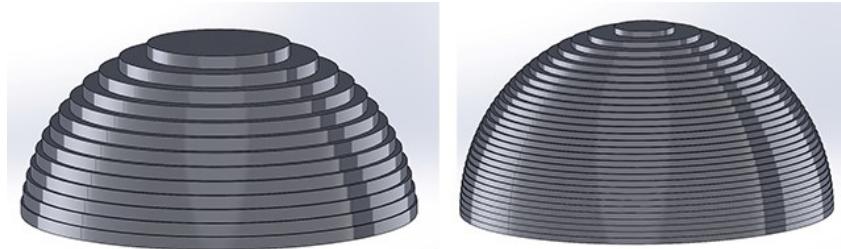
To explain some of the setting options, let's take a look at the pertinent parts of the printer and printing process.



FDM Print Settings

7

Layer thickness or height refers to the height of each layer in mm. Higher values produce faster prints in lower resolution, lower values produce slower prints in higher resolution. The layer thickness/height can span anywhere from 0.05mm to 0.4mm, though it is recommended to be between 0.12mm - 0.22mm.

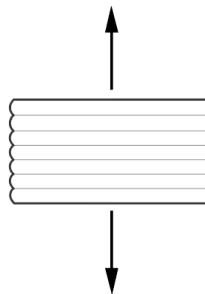


3D printers build up prints in a series of thin *horizontal* layers. Make sure you keep this in mind when orienting your model: **critical details should be oriented parallel to the build platform.**

If the object will be under pressure, it is important to consider the orientation of the “grain”, as it affects the object’s strength.

Tension load normal to layers:

Part is weak

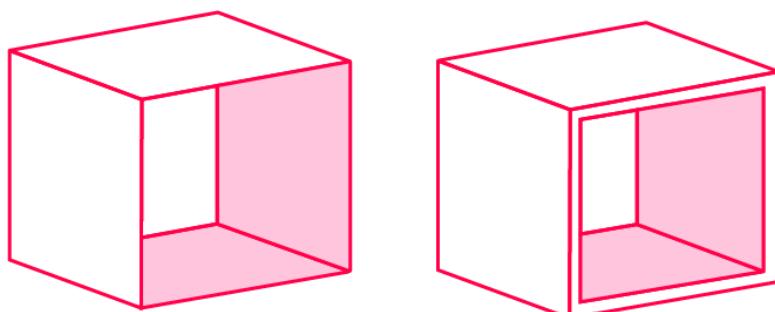


Tension load parallel to layers:

Part is strong



Wall thickness refers to the thickness of the surface of the printed object. The wall thickness has to be greater than or equal to the layer thickness, and is recommended to be a multiple of the nozzle diameter.



Infill is an unseen interior structure of 3D prints that plays a very important role in the overall **strength** of your model. Infill is responsible for connecting the outer shells of your 3D print. The Lab can print your object with **6–36% infill**. You can specify your infill preferences when submitting a print job with the Lab.

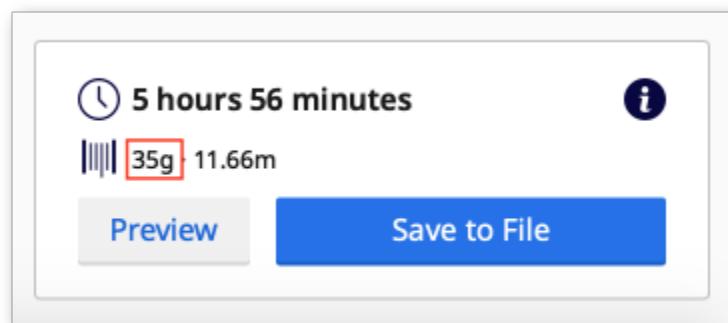
For models that require more strength or need to be assembled with other parts, it is best to select a higher print density to ensure they can sustain enough applied forces. For projects that are for prototyping or visualization purposes, a low print density will suffice.



Estimating FDM costs in Cura

At the Lab, SLA prints are priced based on their weight. For FDM prints, the Lab charges **\$0.10 / 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.



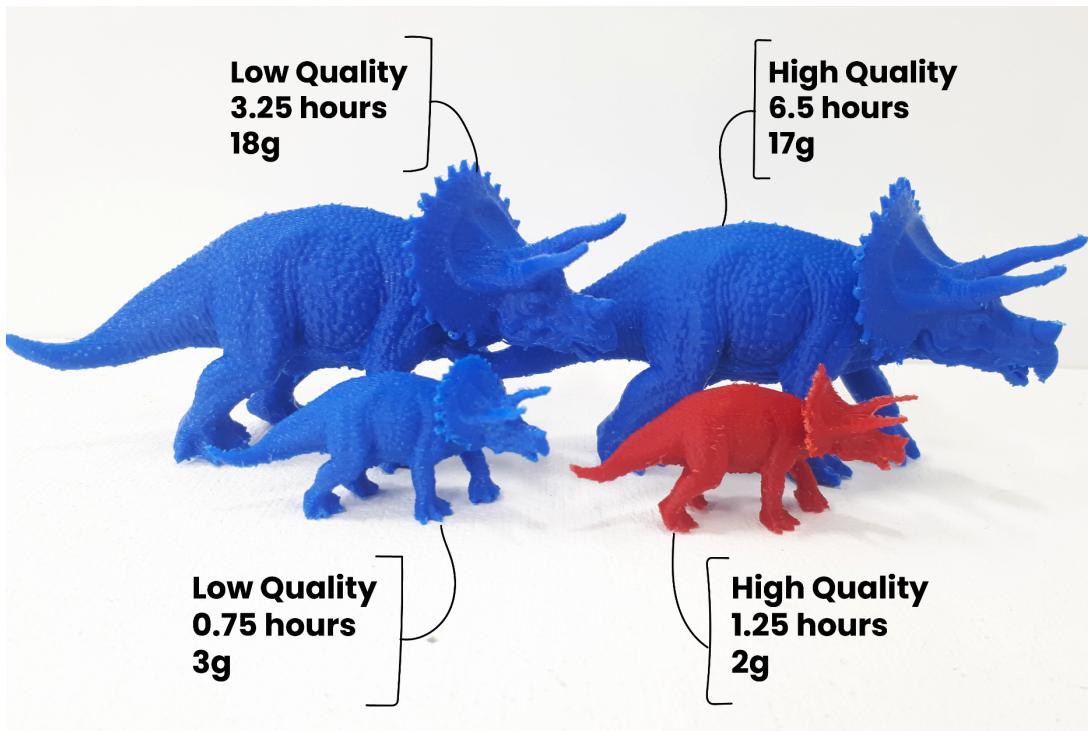
FDM Quality vs Speed

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. Low resolution is more “pixelated.” High resolution is more detailed. You can read more information about the specifics below. The Lab offers three quality options for 3D printing: Low, Medium and High quality.

Low Resolution FDM

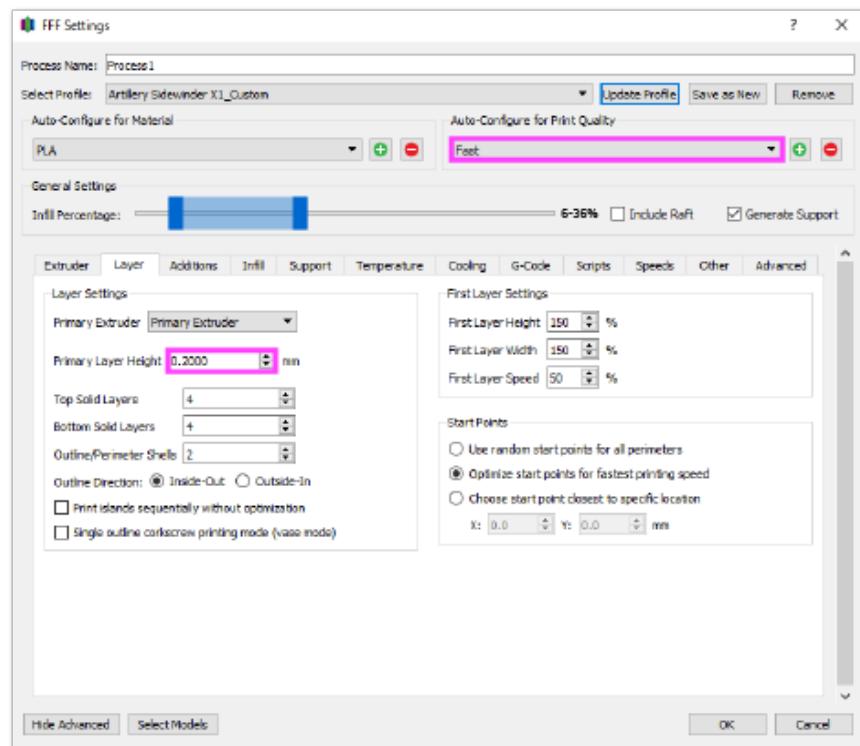


High Resolution FDM



Low Resolution/Fast Print

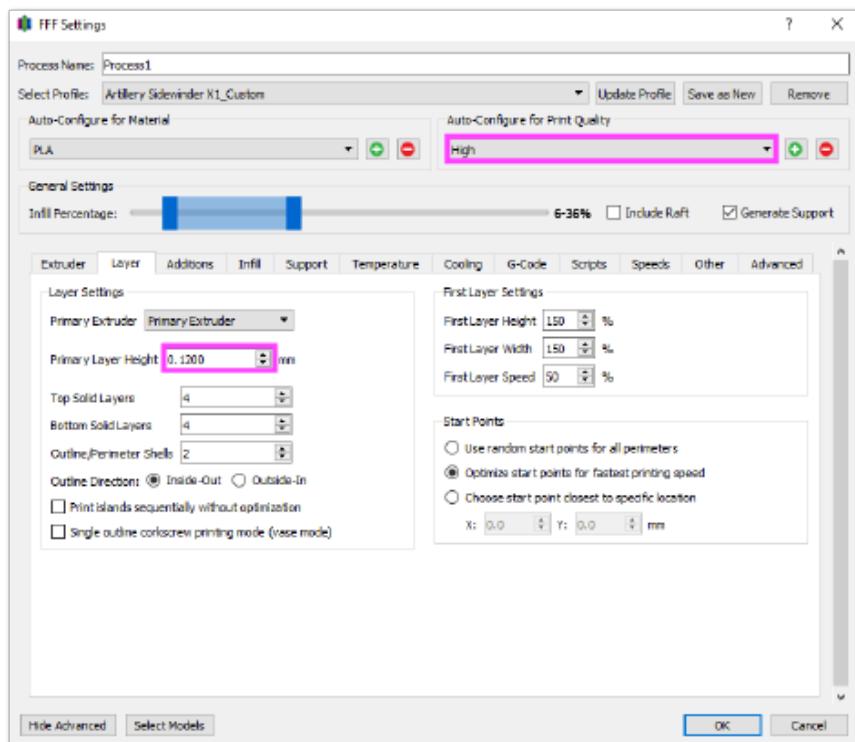
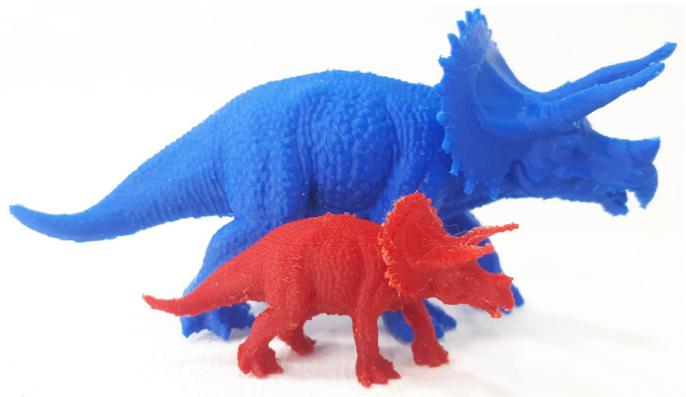
Lower quality will print faster. Printing at a low quality increases the height of each layer of the object. This does not affect the final size of your object. Thicker layers do not allow for fine surface detail and **the layered texture is more visible**. The walls of a low quality print will be slightly thicker and use slightly more plastic.



The above image shows the Lab's **FDM** print settings (in Simplify 3D) for low quality prints. The layer height is set to 0.2 mm.

High Resolution/Slow Print

Higher quality will print slower. Printing at a high quality decreases the height of each layer. This does not affect the final size of your object. Thinner layers **look smoother and can be more detailed**. The walls of a high quality print will be slightly thinner and use slightly less plastic.



The above image shows the Lab's **FDM** print settings (in Simplify 3D) for high quality prints. The layer height is set to 0.12mm.

FDM Supports

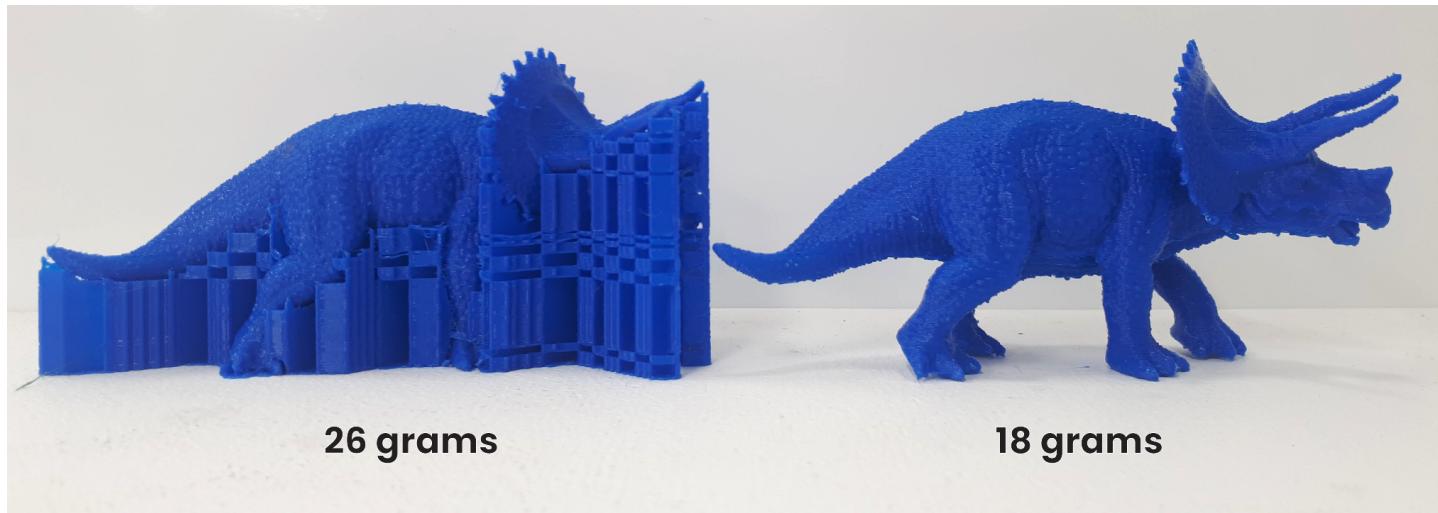


Flush Angle Cutters

When the plastic comes out of the printing nozzle, it is liquified and does not hold its shape well. Each new layer must be supported by the layer beneath it. If your model has an overhang which is not supported by anything below, there's a good chance it will droop. To avoid this, it is recommended to use additional 3D printed support structures called "supports".

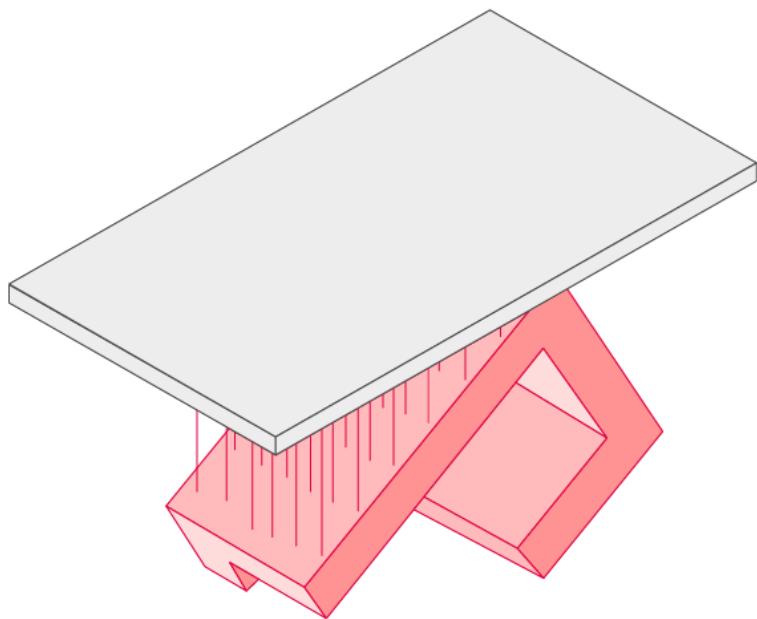
Supports increase the material weight and duration of the print. They can cause little imperfections in the object's surface, and can be a hassle to remove.

The Lab recommends using "flush angle cutters" to help remove stubborn support material.



The above image compares the weight of an FDM 3D print before and after the supports are removed.

SLA Printing



SLA Slicing Software

A 3D slicing software is the in-between step after modeling and before printing. The slicer interprets the STL's array of polygons into printable, layered toolpaths (toolpaths = specific locations where the machine travels and does its thing). The slicing software can estimate 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 printer is [PreForm](#). This slicer is free! The Lab encourages you to preview your SLA print in PreForm prior to printing to get helpful information, such as:

- Quantity of detail / resolution being printed
- Where support material will be applied
- Time required to complete the print
- Areas that will be too thin to print successfully (details smaller than 1mm may not print cleanly)
- Object scale (The maximum SLA printing volume is 145 x 145 x 175 mm)
- The orientation of your object in relation to the print "grain" (see page 15)
- "Printability" errors with your model

SLA Print Settings

Although FDM printing is cheaper and the preferred process for most models, the SLA printing technology offers a much higher print resolution quality than its counterpart.

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.

Applying print settings in PreForm is as simple as:

1. Selecting your preferred layer thickness (see pages 17-19)
2. Apply auto-generated support structures (see page 20)

If you want to print a hollow model, you can use a mesh-editing software to [hollow-out your .STL file to print a shell only](#) (instructions linked). This saves on unnecessary material cost.

SLA Printability

Check out [this article](#) for helpful SLA design tips!

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**.

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.

To avoid “suction cupping”, use a 3D design software to either fill the hollow or add drainage holes to minimize suction during printing.

Estimating SLA costs in PreForm

When a model is imported and support structures added, 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 (explained below).

DETAILS	
 Print Time	~ 7 h
 Layers	885
 Volume	98.60 mL

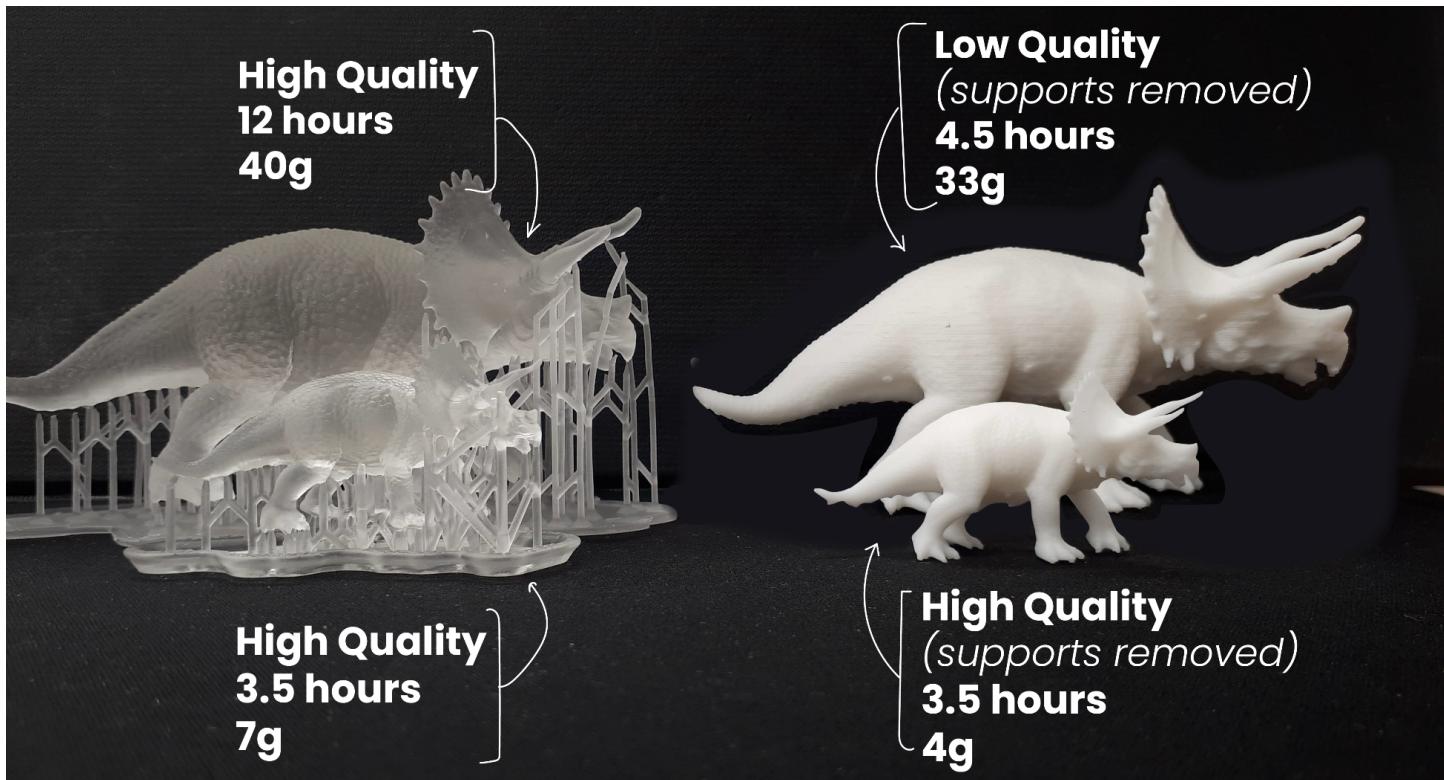
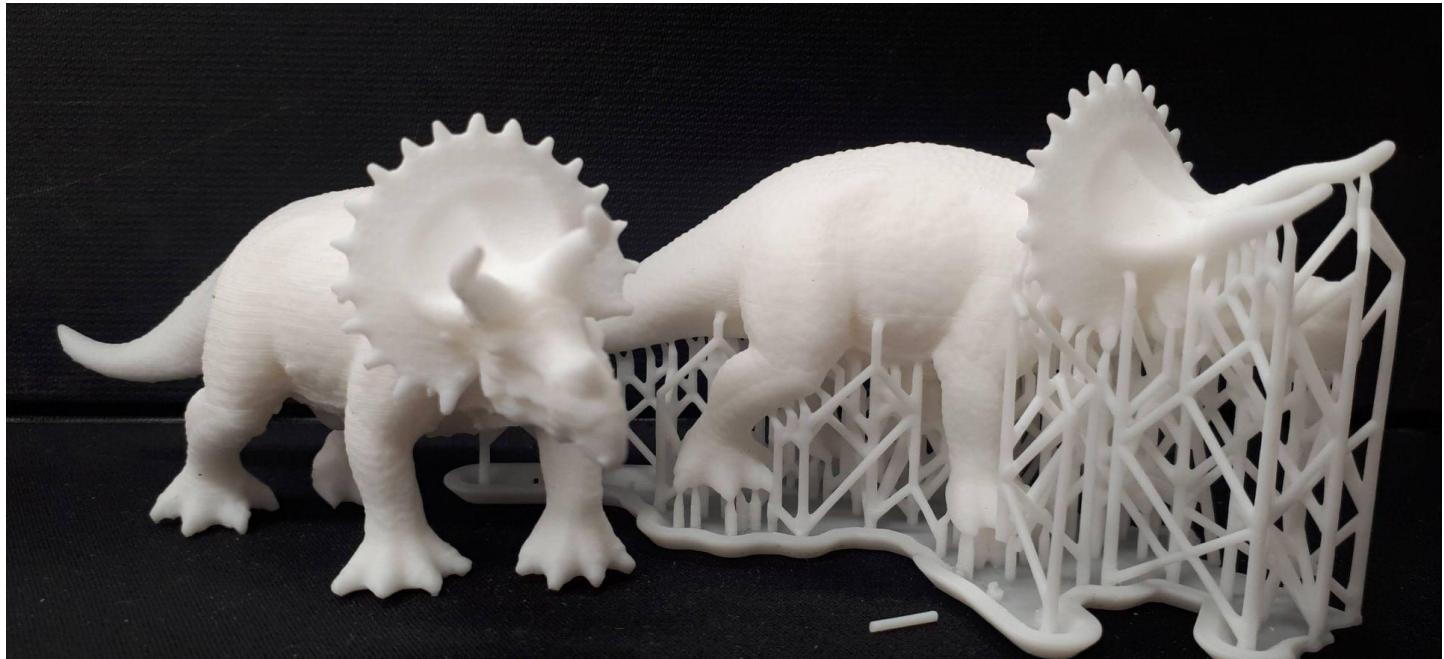
At the Lab, SLA prints are priced based on their weight. For SLA prints, the Lab charges **\$0.33 / gram** of plastic. The PreForm software calculates the volume (mL) of resin that will be printed. Formlabs **resin weighs approximately 1 g/mL**. You can use that approximation to estimate the print's total weight.

SLA Quality vs Speed

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. Low resolution is more “pixelated.” High resolution is more detailed. You can read more information about the specifics below. The Lab offers three quality options for 3D printing: Low, Medium and High quality.

Low Resolution SLA (supports removed)

High Resolution SLA (with supports)



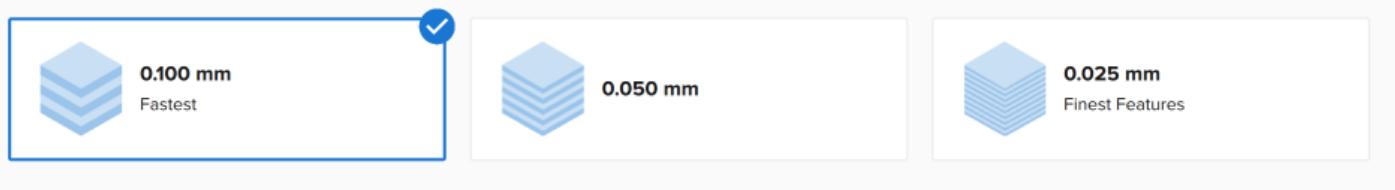
Low Resolution/Fast Print

Lower quality will print faster. Printing at a low quality increases the height of each layer of the object. This does not affect the final size of your object. Thicker layers do not allow for fine surface detail and **the layered texture is more visible**.



Choose Layer Thickness

Layer thickness affects both the speed and quality of a print. Thicker layers print faster, but sacrifice detail. Thinner layers print slower, but are able to capture finer details.



The above image shows the Lab's **SLA** print settings for low quality prints. The layer height is set to **0.1 mm**.

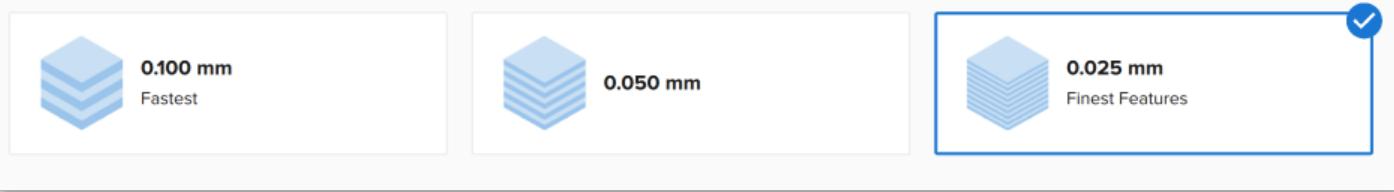
High Resolution/Slow Print

Higher quality will print slower. Printing at a high quality decreases the height of each layer. This does not affect the final size of your object. Thinner layers **look smoother and can be more detailed**.



Choose Layer Thickness

Layer thickness affects both the speed and quality of a print. Thicker layers print faster, but sacrifice detail. Thinner layers print slower, but are able to capture finer details.



The above image shows the Lab's **SLA** print settings for high quality prints. The layer height is set to **0.025 mm**.

SLA Supports

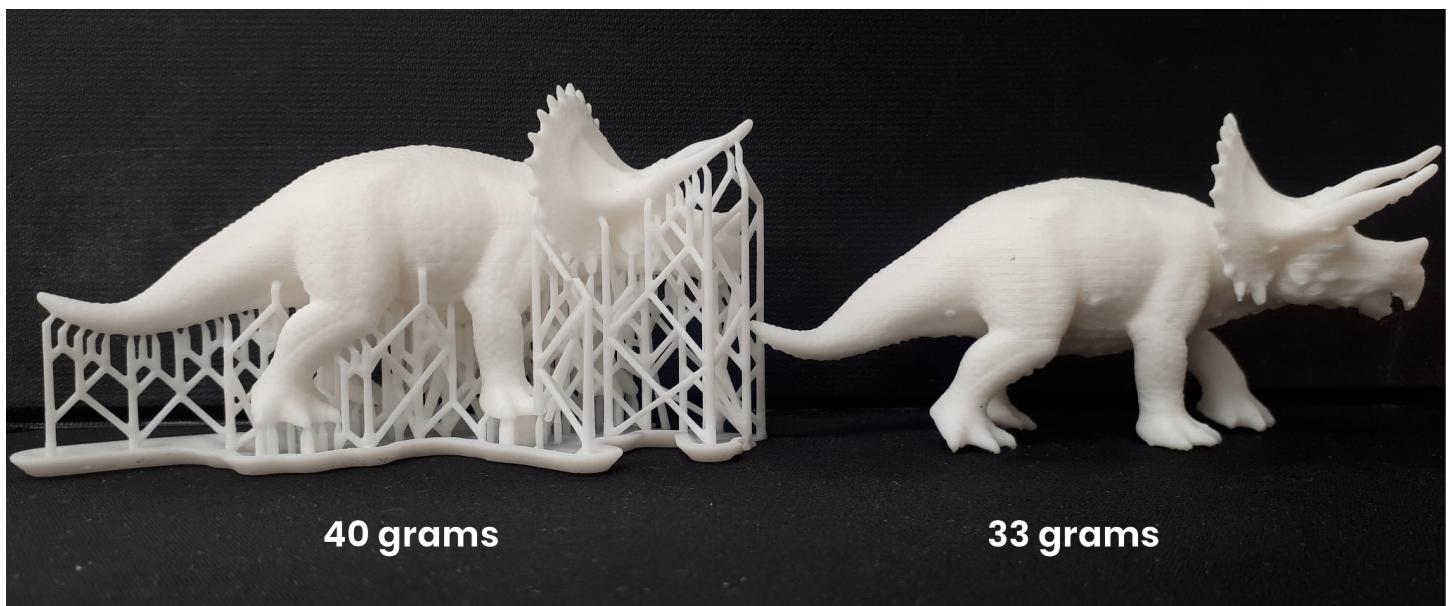


Flush Angle Cutters

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 support structures called "supports".

Supports increase the material weight and duration of the print. They can cause little imperfections in the object's surface, and can be a hassle to remove.

The Lab recommends using "flush angle cutters" to help remove stubborn support material.



The above image compares the weight of an SLA 3D print before and after the supports are removed.