

Tutorial and Demonstration

Gabriel Dorfsman-Hopkins

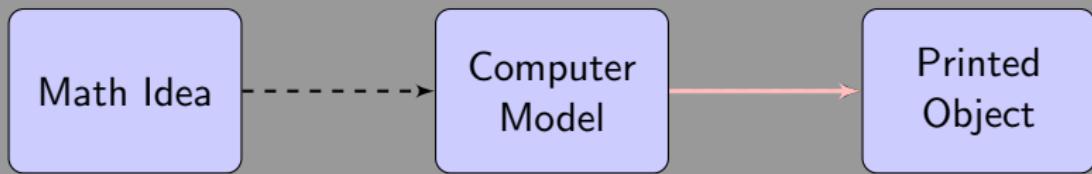
University of California, Berkeley

January 3, 2022



The Mini-Course Goal

There are two main steps to creating a 3d printed mathematical object.



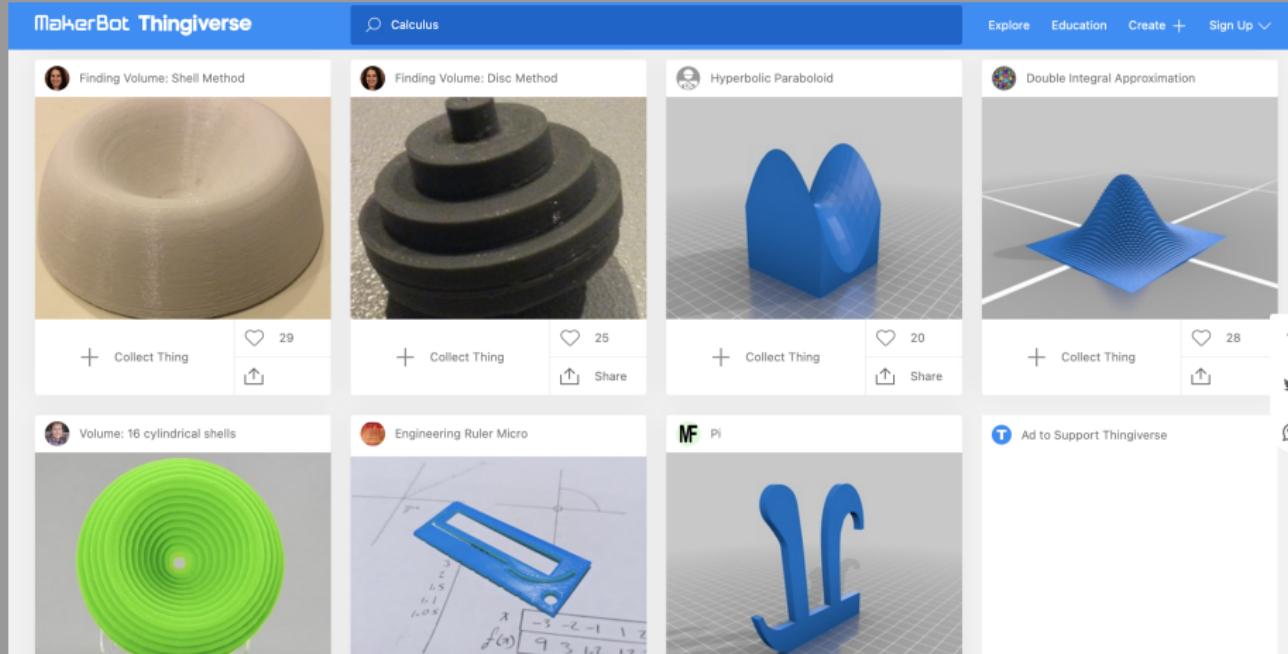
In this tutorial we will be focusing on the second (pink) arrow.

Extrusion Printers

- We saw there are many different kinds of 3d printing and additive manufacturing. I'm going to focus on how to use a desktop extrusion 3d printer, which is one of the more accessible.
- "Programmable Glue-gun"

Aside: A plug for thingiverse

Designing your own 3D models takes some time and effort. If you want to get printing right away, thingiverse is a great place where you can download models to play with your printer and use in the classroom.

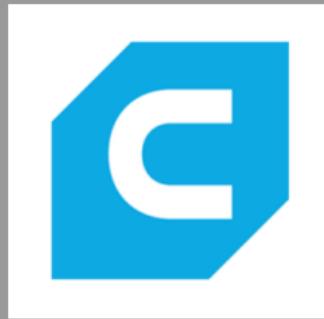


What is a model anyways?

- Let's say we already made (or downloaded) our model...what is it?
- For our purposes, stored as an STL or OBJ file.
- Essentially a text document listing collection of triangles.
- 3d printers don't know how to print a collection of triangles.
- 3d printers know how to follow a tool path...a set of instructions telling the printer head where to go, and how much filament to squirt out.
- To turn triangles into a toolpath we need.....

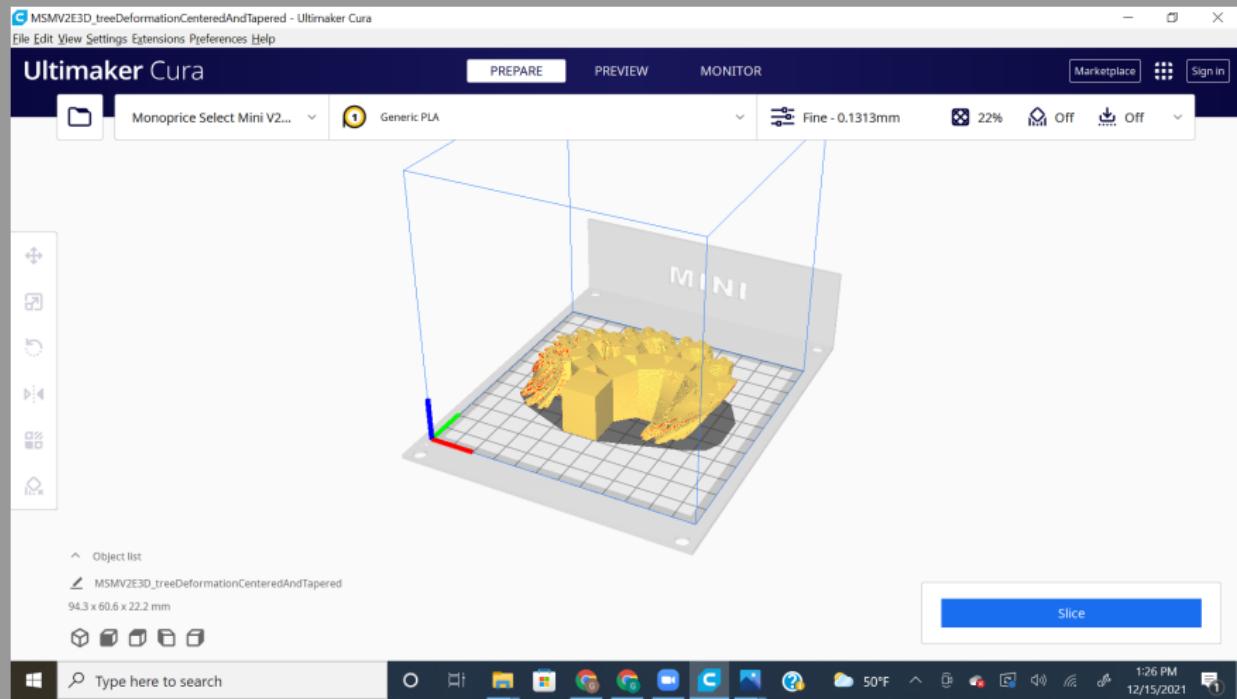
Slicers!

Software to turn geometry described by a collection of triangles into toolpaths that a 3d printer can follow.



Cura is free, open source, and very robust.

What it looks like



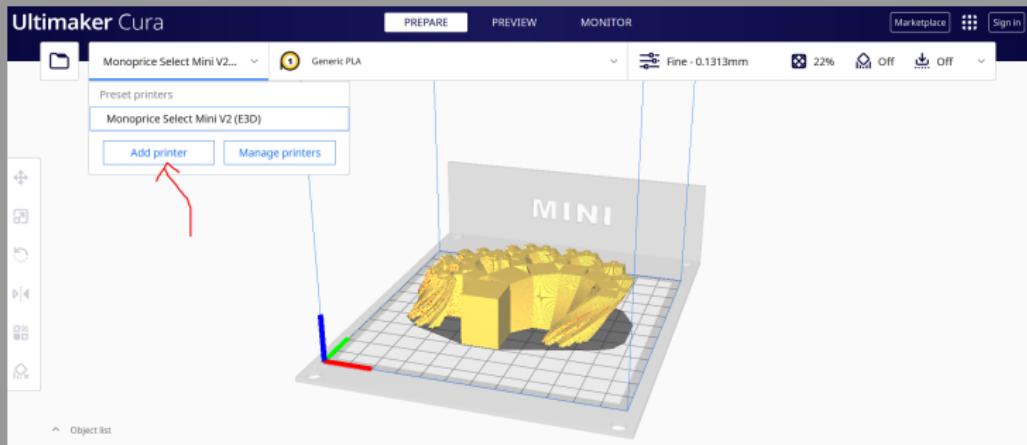
Setting up Cura

The general idea is that Cura will turn an STL or OBJ into a toolpath. In practice there are a lot of settings to fiddle with. Most of them you don't really need to worry about until you are fine tuning (or troubleshooting). The most important are...

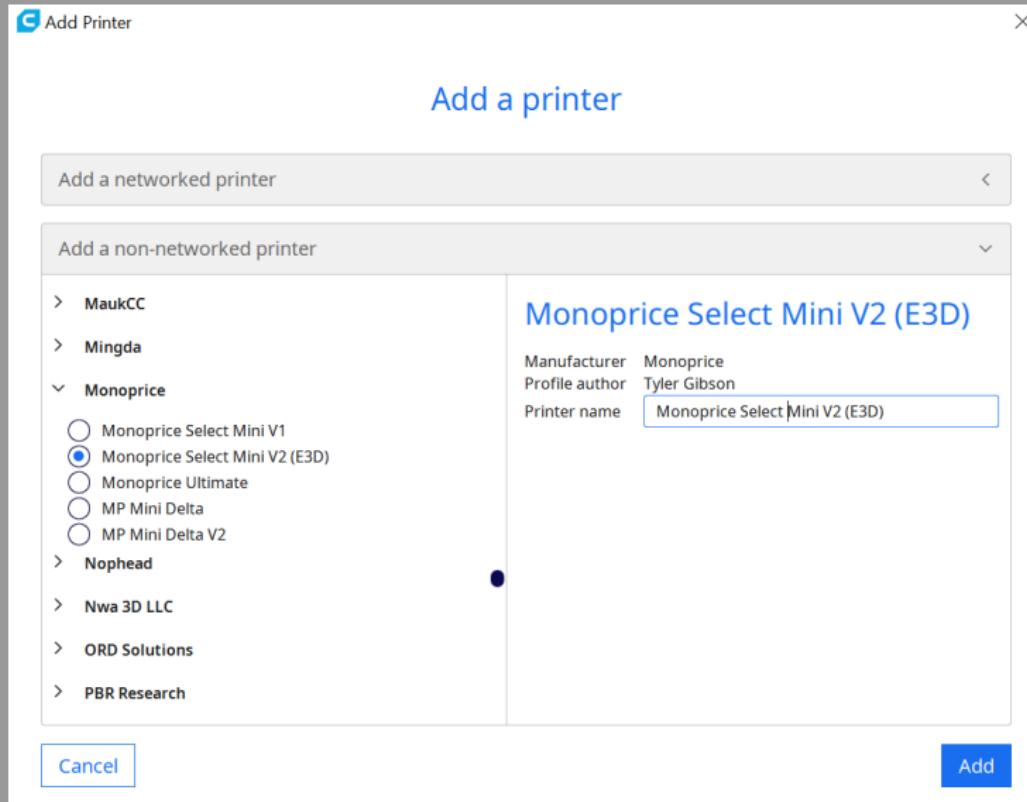
- Machine Settings.
 - Dimensions (depth, width, height) of your machine's build area.
 - Type of build plate (metal, glass).
 - Width of the nozzle (what size filament are you using?)
- Filament Settings.
 - Size (usually 1.75mm or 2.75mm).
 - Material (PLA good for beginners, ABS and PETG common).
- Temperature Settings.
 - Nozzle temperature (listed on your filament).
 - Plate temperature (depends on your buildplate and material, important for PETG, flexible for PLA, I experiment with this if I'm having trouble with adhesion).

Adding a printer

Lucky for us...Cura is open source, and build settings for every 3d printer I've ever worked with are already pre-loaded into the system!

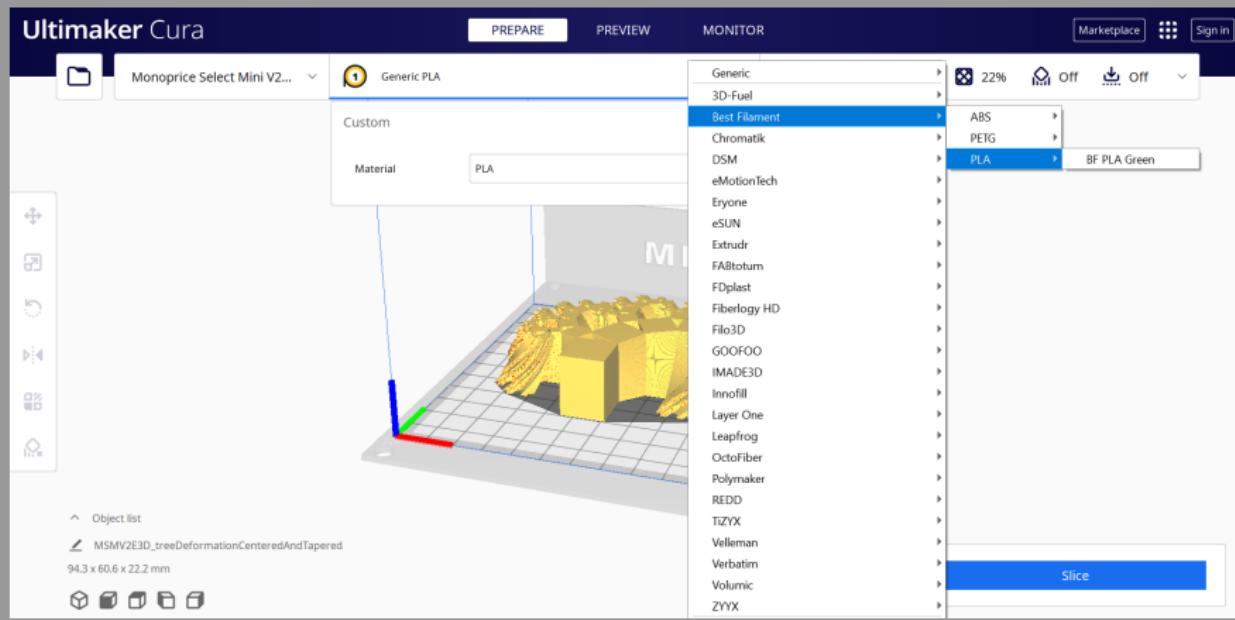


Adding a printer



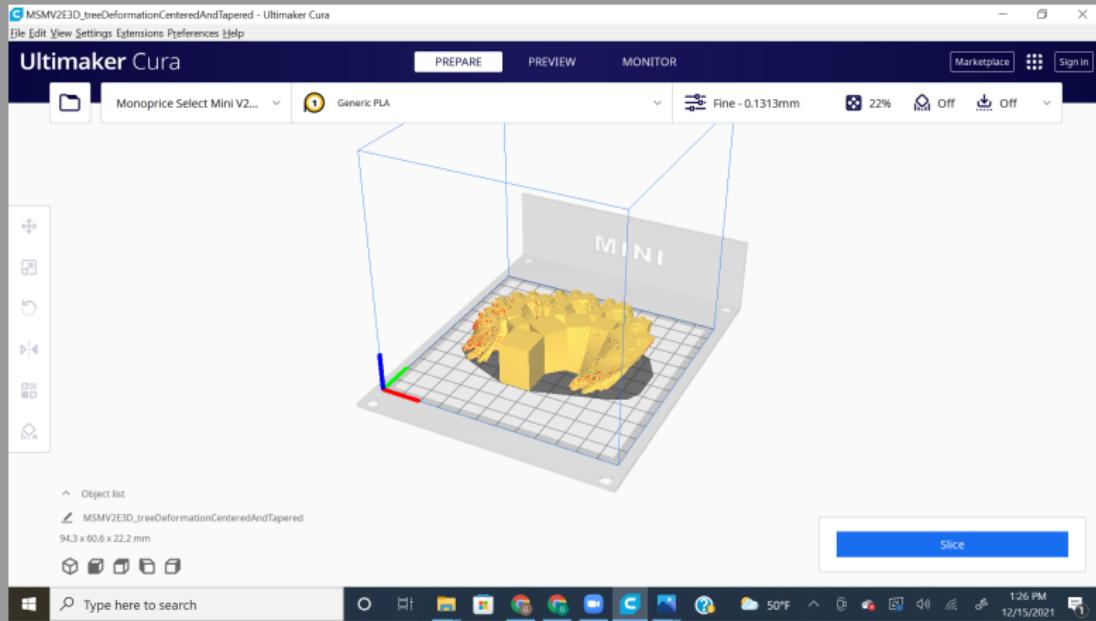
Selecting Filament

It also has settings for many filaments.



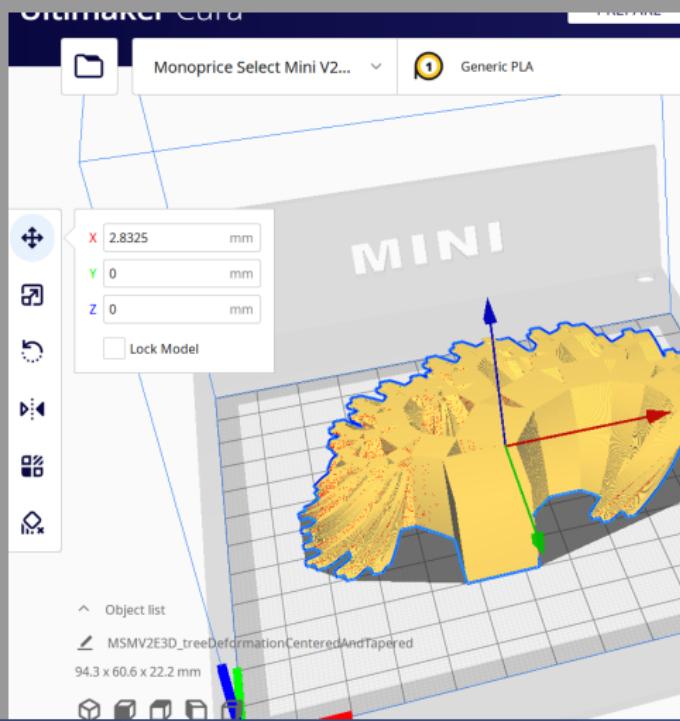
Slicing with Cura

Once you upload your model to Cura, you can scale, move, and rotate it to where you want in the build plate.



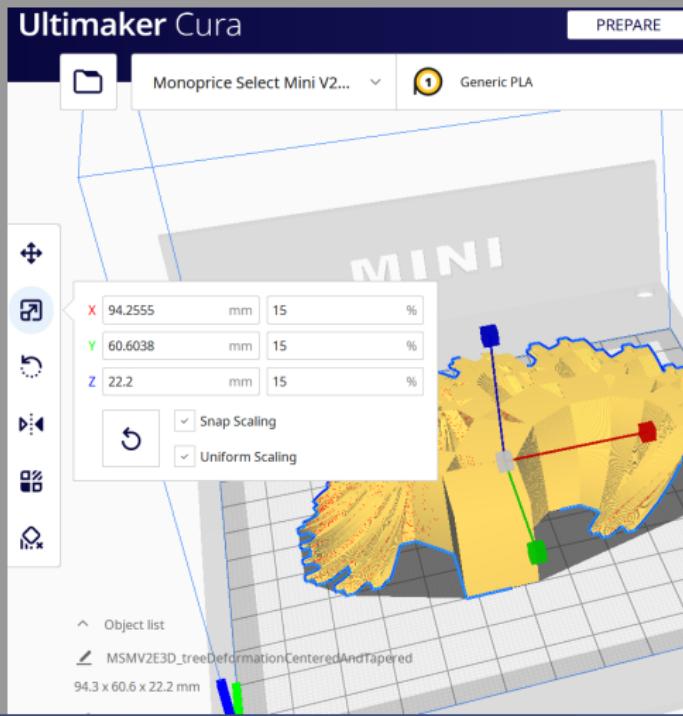
Slicing with Cura

Once you upload your model to Cura, you can scale, move, and rotate it to where you want in the build plate.



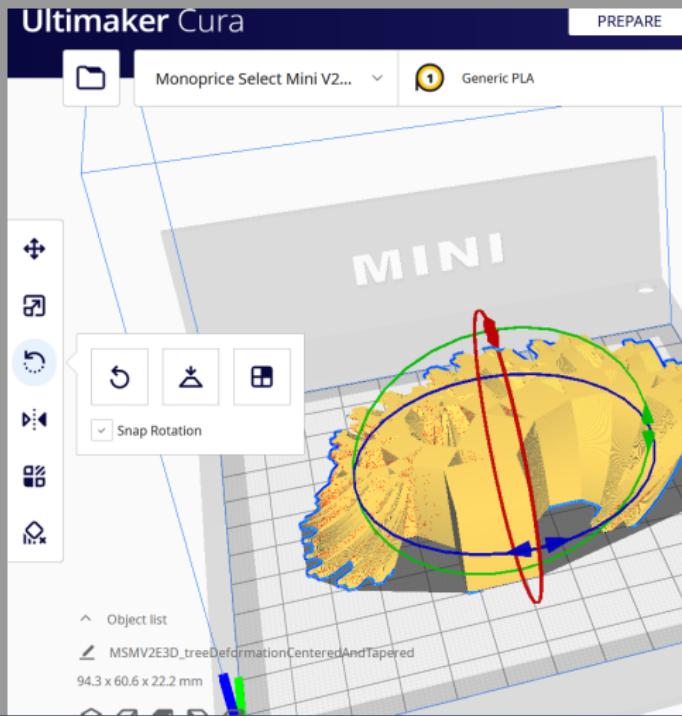
Slicing with Cura

Once you upload your model to Cura, you can scale, move, and rotate it to where you want in the build plate.



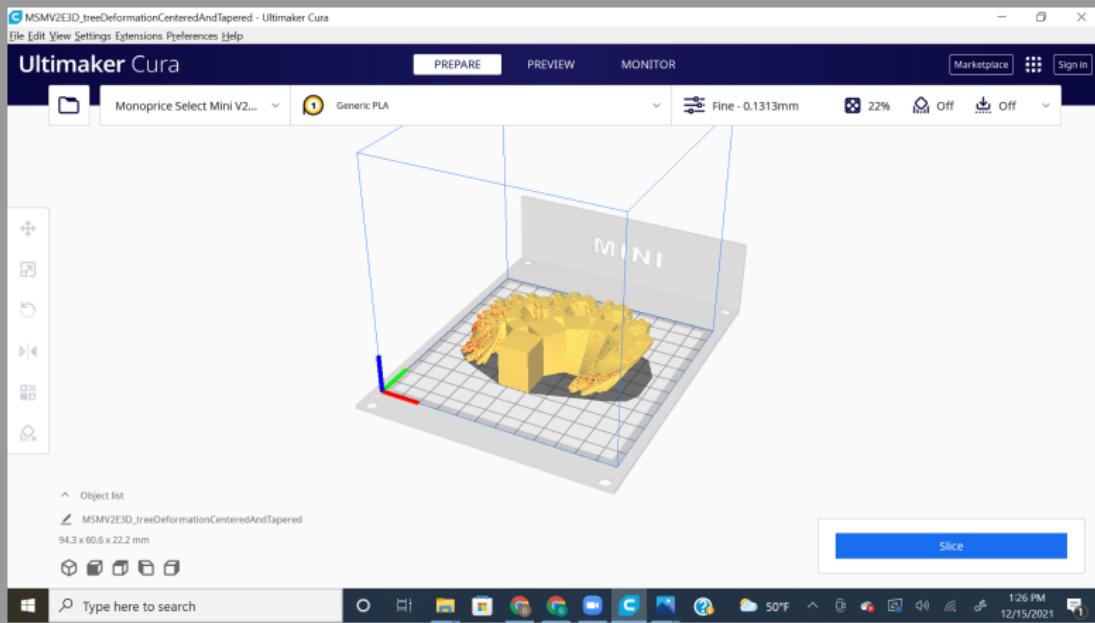
Slicing with Cura

Once you upload your model to Cura, you can scale, move, and rotate it to where you want in the build plate.



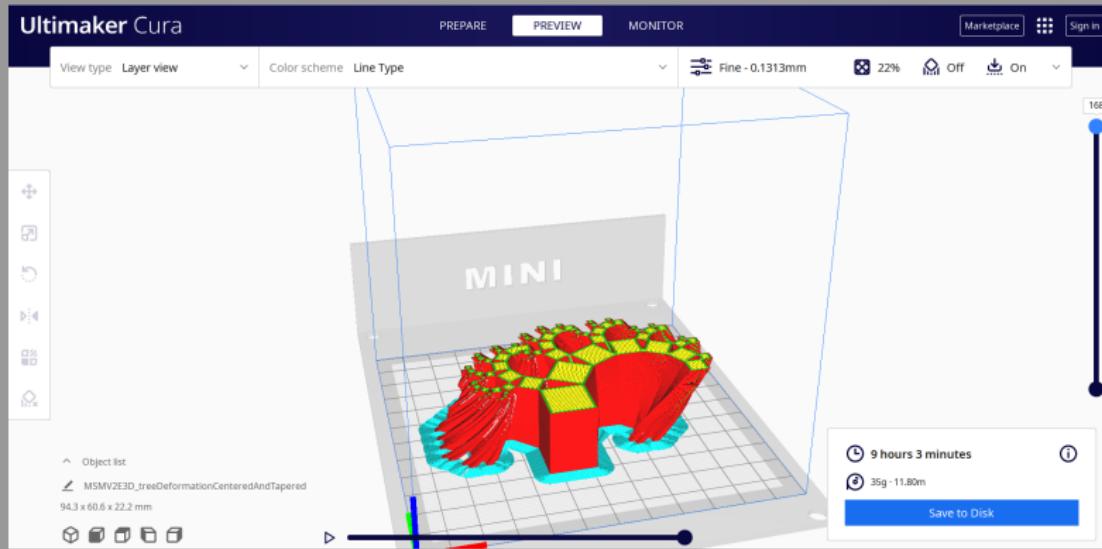
Slicing with Cura

Click "slice" to generate a toolpath (gcode).



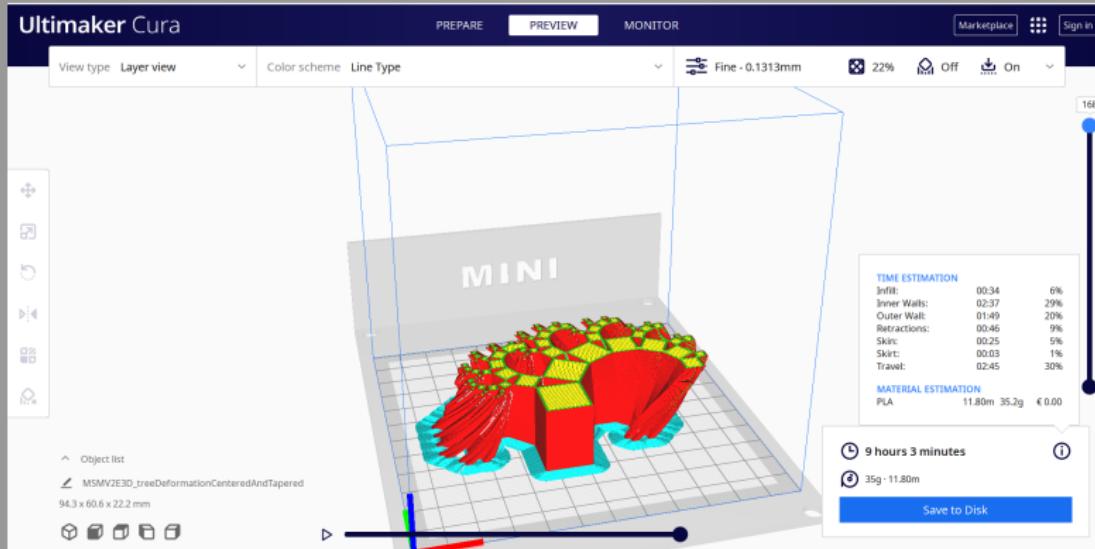
Slicing with Cura

View your preview. It's a nice cartoon of the entire toolpath!



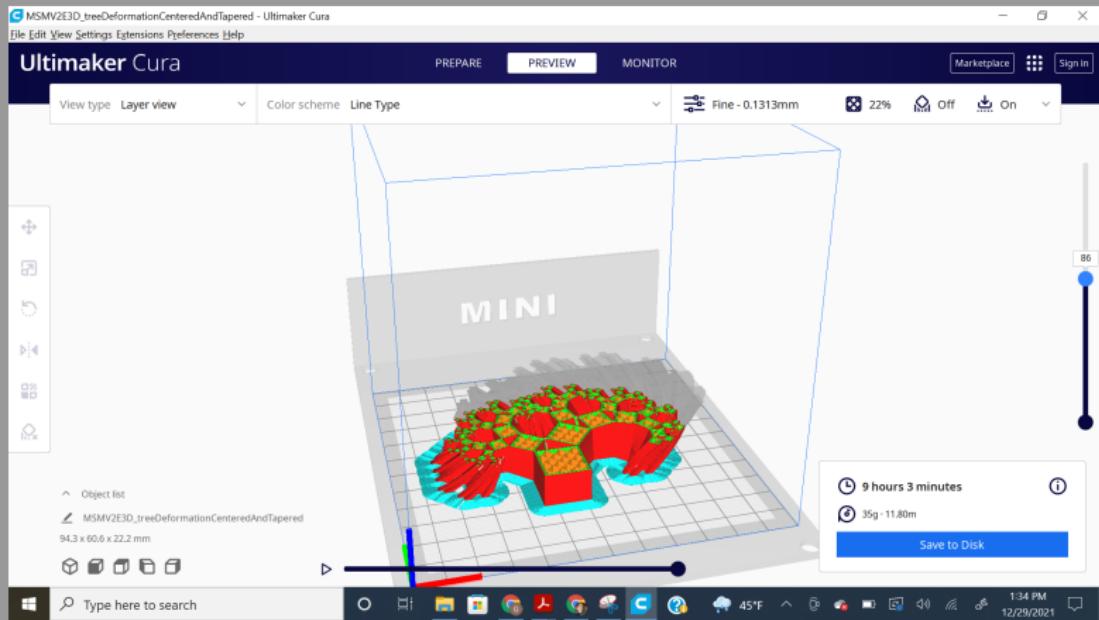
Slicing with Cura

View your preview. It's a nice cartoon of the entire toolpath!



Slicing with Cura

View your preview. It's a nice cartoon of the entire toolpath!



Slicing with Cura

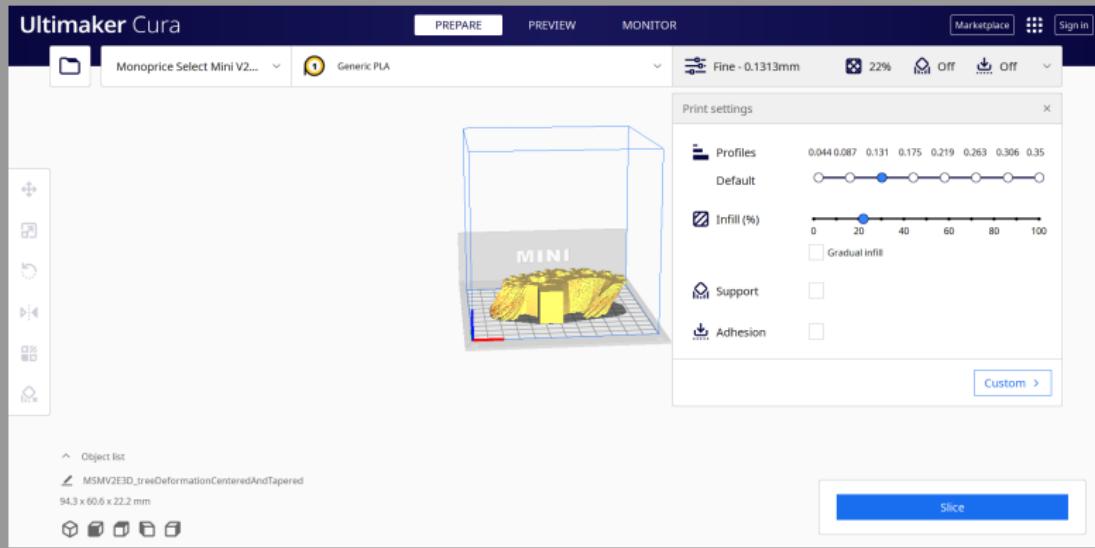
Then you can:

- Print directly.
- Print from wifi
- Save your gcode file (for example to an SD card).

The first 2 options have the advantage of having a monitor mode as the printer talks to your computer. That said, I can't usually leave my computer in one place for the entirety of a long print, so I usually save to an SD card.

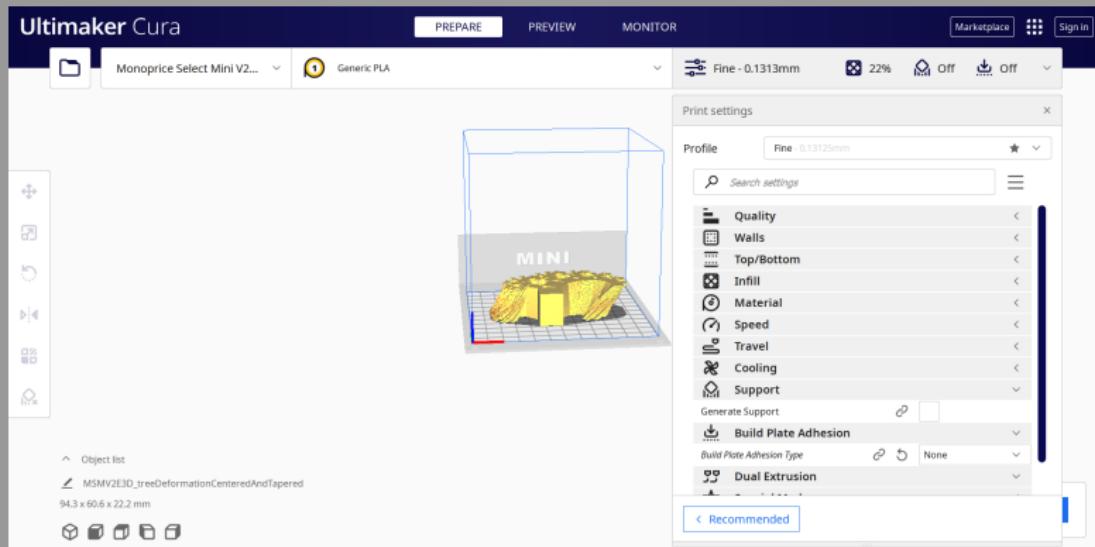
Other Settings

There are a many other settings. Here's how to access them:



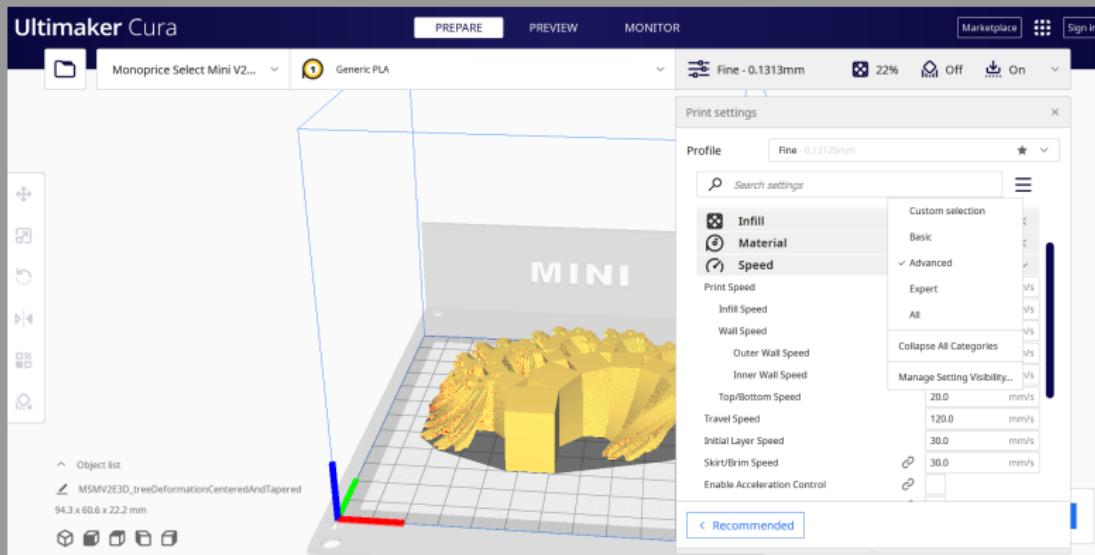
Other Settings

There are a many other settings. Here's how to access them:



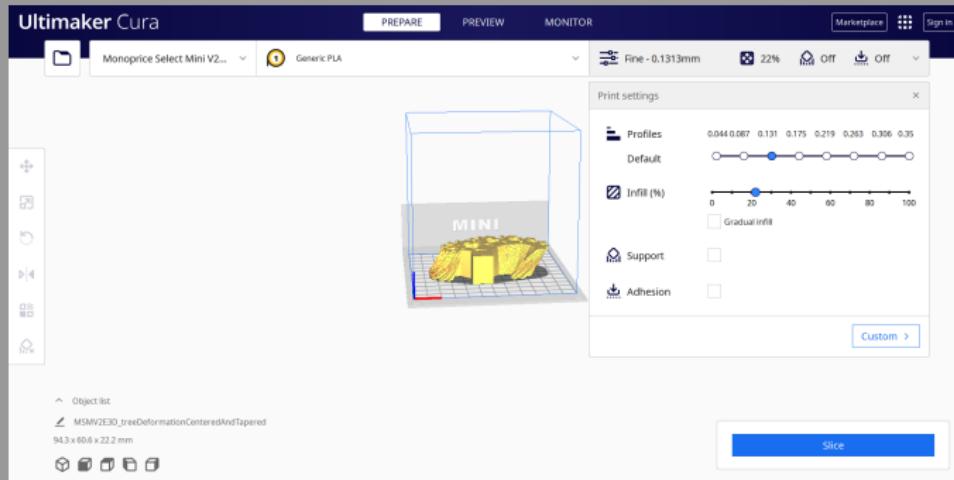
Other Settings

There are a many other settings. Here's how to access them:



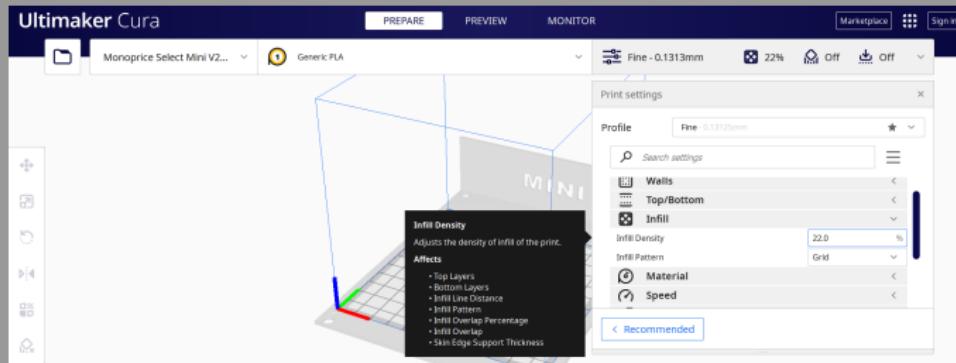
Other Settings: Infill

When you print, the inside isn't 100% filled with material. It is filled with mostly empty space, you choose the pattern and density. This affects the printing speed and strength of the result.



Other Settings: Infill

When you print, the inside isn't 100% filled with material. It is filled with mostly empty space, you choose the pattern and density. This affects the printing speed and strength of the result.



Other Settings: Infill

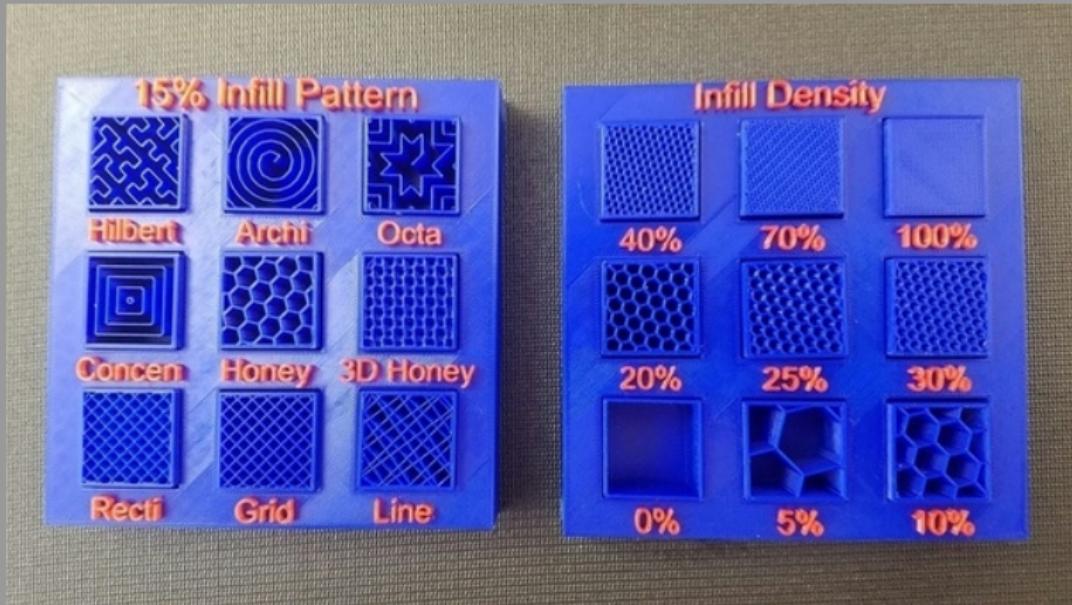
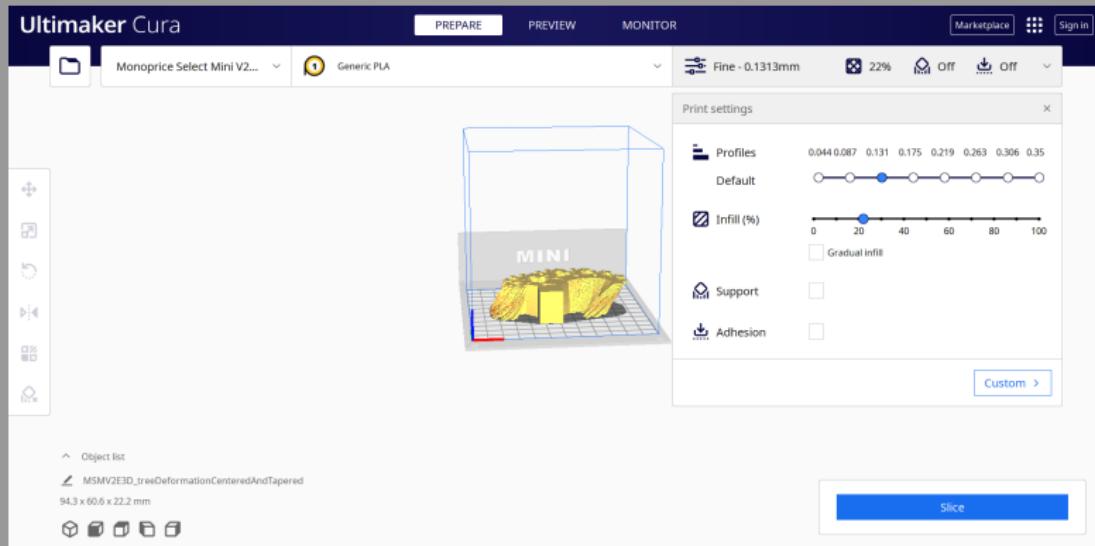


Photo Source: Louis Vaught

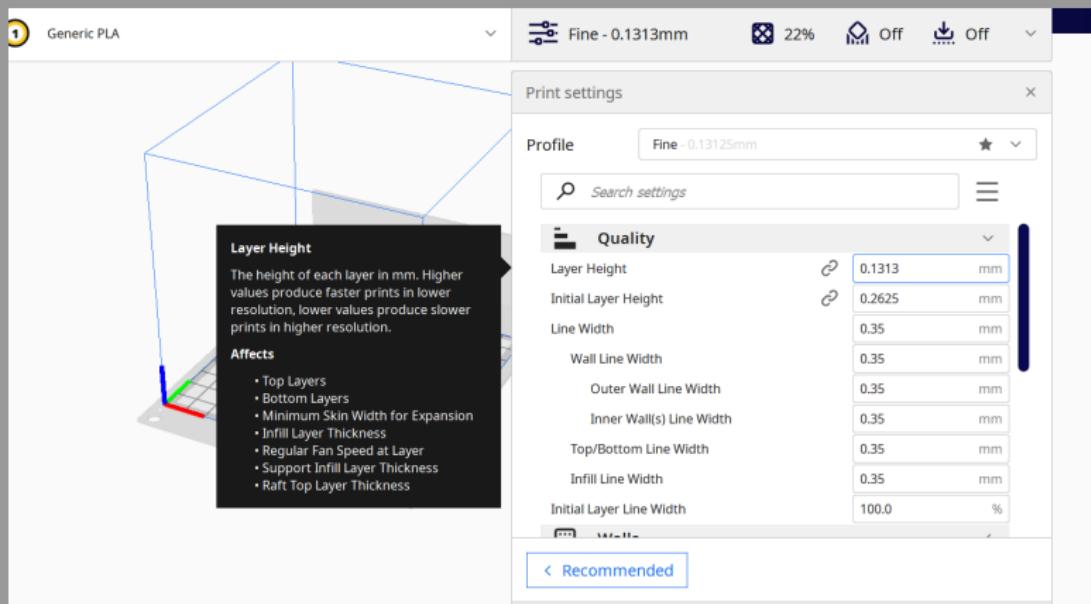
Other Settings: Quality

Essentially choosing resolution.



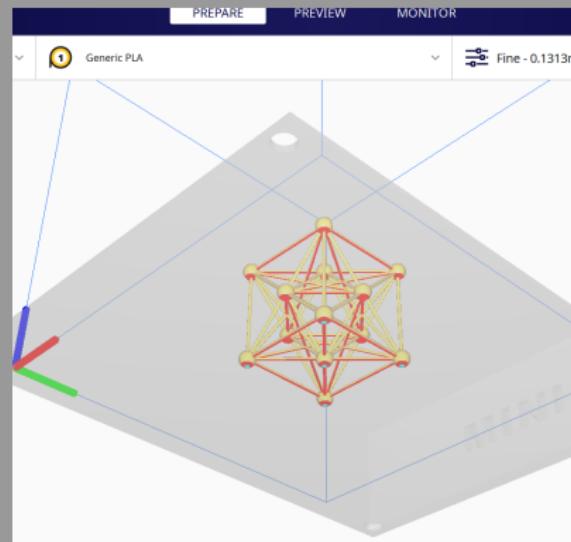
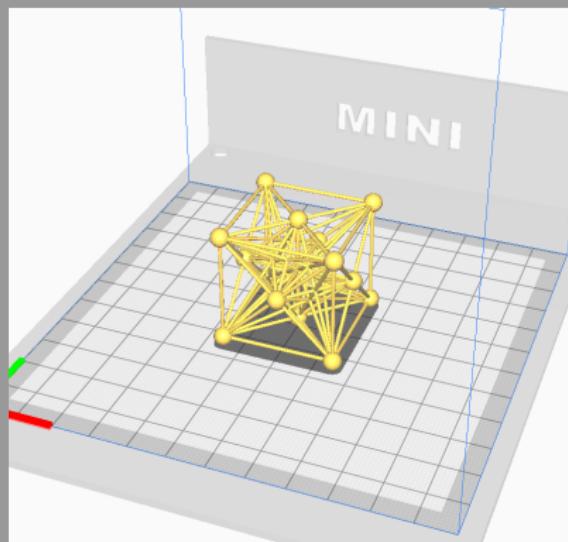
Other Settings: Quality

Essentially choosing resolution.



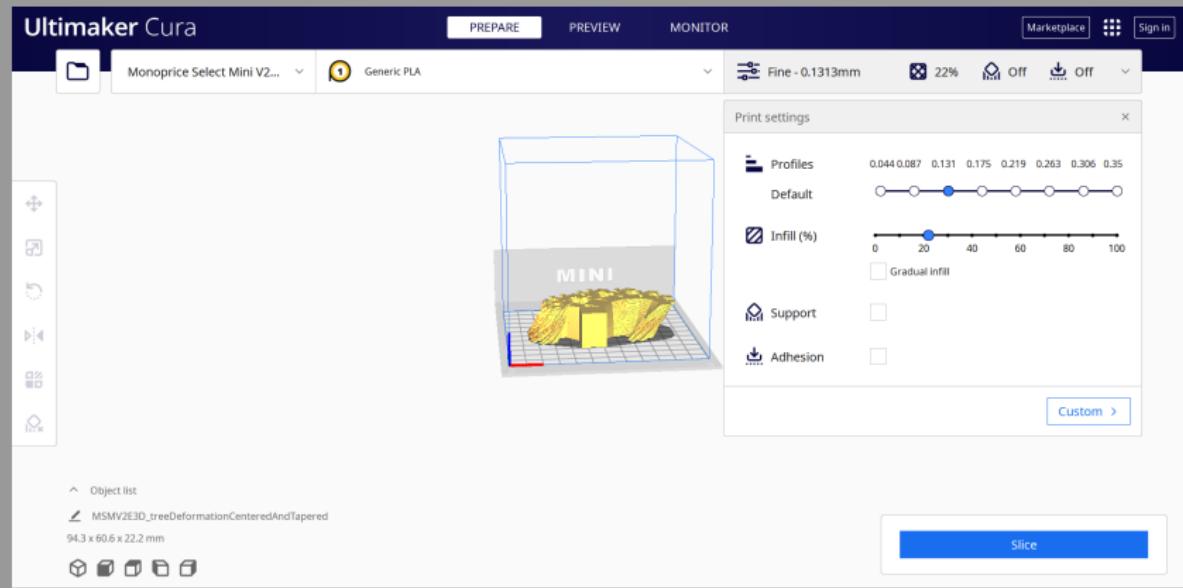
Support Material

A 3d printer builds each layer on the last. So what if you want overhangs or crossbars?



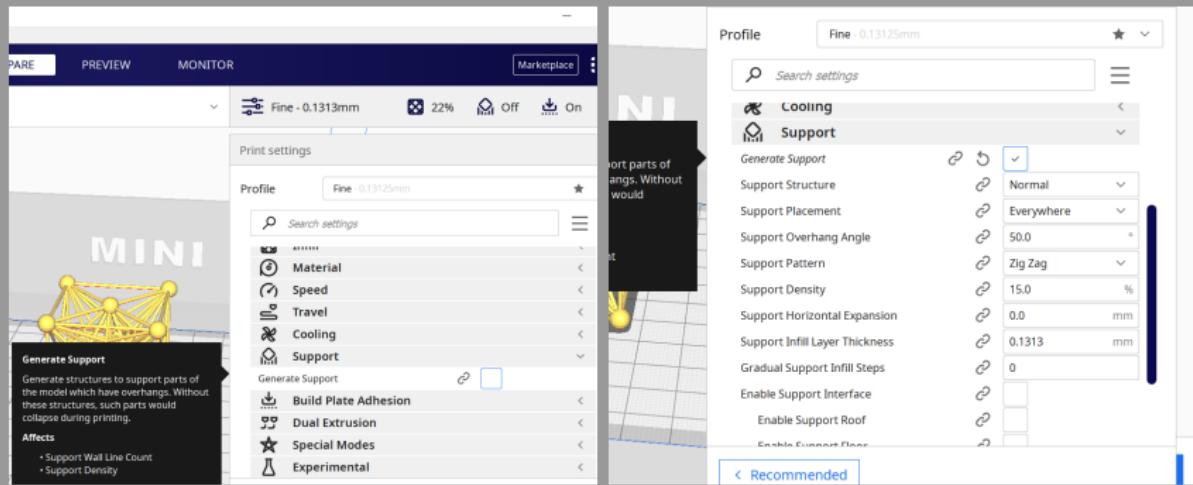
Support Material

You can print **support material**: scaffolding which you then remove after the fact.

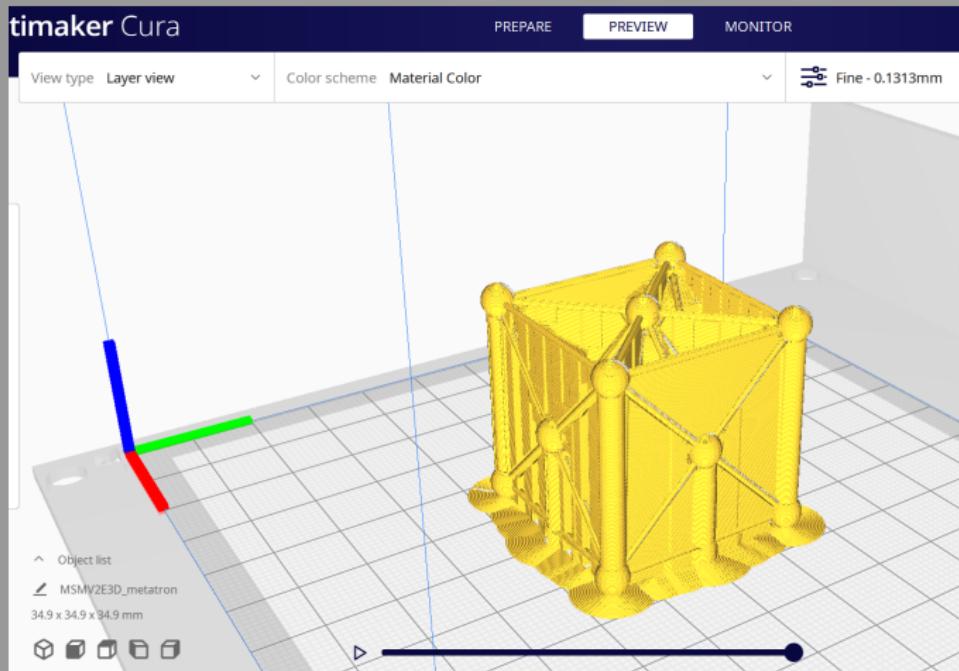


Support Material

You can print **support material**: scaffolding which you then remove after the fact.



Support Material



This can be a bit of a pain to remove, so use sparingly.

Setting Up Your Printer

That takes care of the software side of things. But you need your printer to be functioning too. The 3 most important pieces are.

- Setting up build surface.
- Adjusting buildplate height.
- Loading filament.

Control Axes

Most printers have 4 axes of control you can access directly from the interface.

- The x,y,z axes for the printhead (or nozzle).
- The extrusion axis which feeds the filament.
- There is also the home axis which sends it to the height of printing the first layer.

Control Axes



Build Surfaces

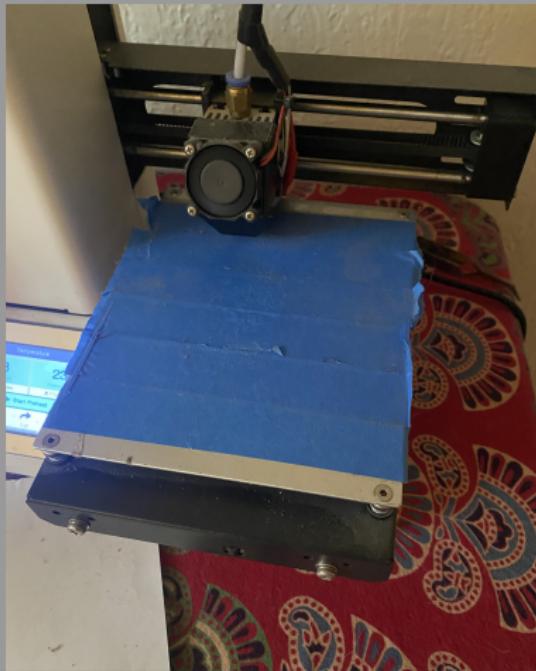
Build plate can have many different materials. Common ones include:

- Glass
- Metal
- Adhesive surfaces you can stick to your plate
- Removable and flexible surfaces

I usually cover my (metal) build plate in blue painters tape, which leads to easy adhesion and removal.

Build Surfaces

I usually cover my (metal) build plate in blue painters tape, which leads to easy adhesion and removal.

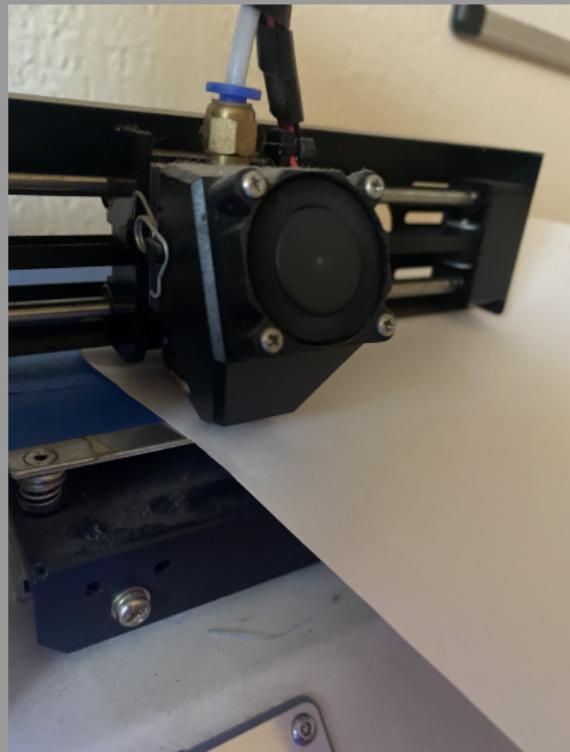


Adjusting buildplate height

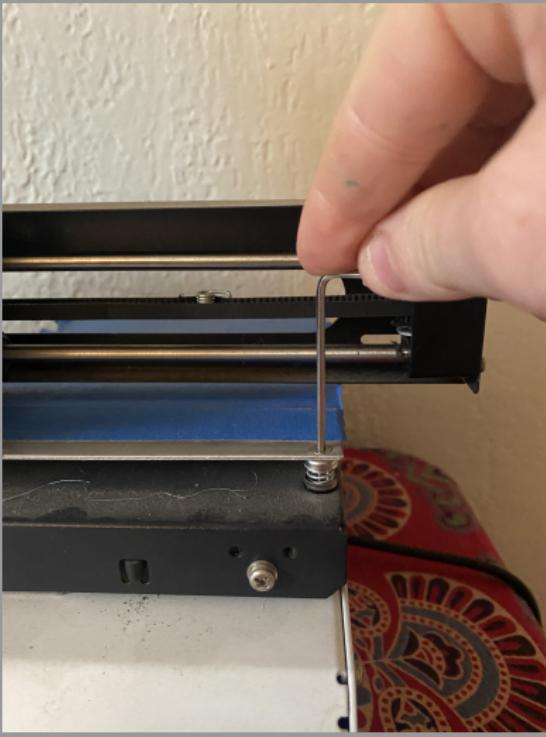
You want the nozzle to be the perfect height so that the first layer adheres to the buildplate. For this I use **The Paper Trick**.

- Bring nozzle to home position.
- Slide a piece of paper under the nozzle. Adjust buildplate height so that it fits, but also barely catches.
- Different printers have different settings for how to raise and lower the build plate. These could be turn wheels, or nuts, or a digital interface. Some even do it automatically!.
- Do the paper trick on the 4 corners of the build plate as well as the center.

The Paper Trick



Adjusting Height



Loading Filament

The process is sometimes automated. The general idea is:

- Heat up nozzle hotter than the filament setting. (So for PLA which wants about 190C, I use 210C).
- Cut the end of the filament so that it has a point.
- Insert filament. Use the extrusion axis to load until it starts coming out of the head.

Inserting Filament



Inserting Filament



OK...Let's print

Once you have your toolpath prepared in Cura, and the 3d printer ready to go, then start your print.



Watch the first layer, it is the most important.

What to look for in the first layer: Height

The first layer can make it clear if your nozzle is too high or too low.

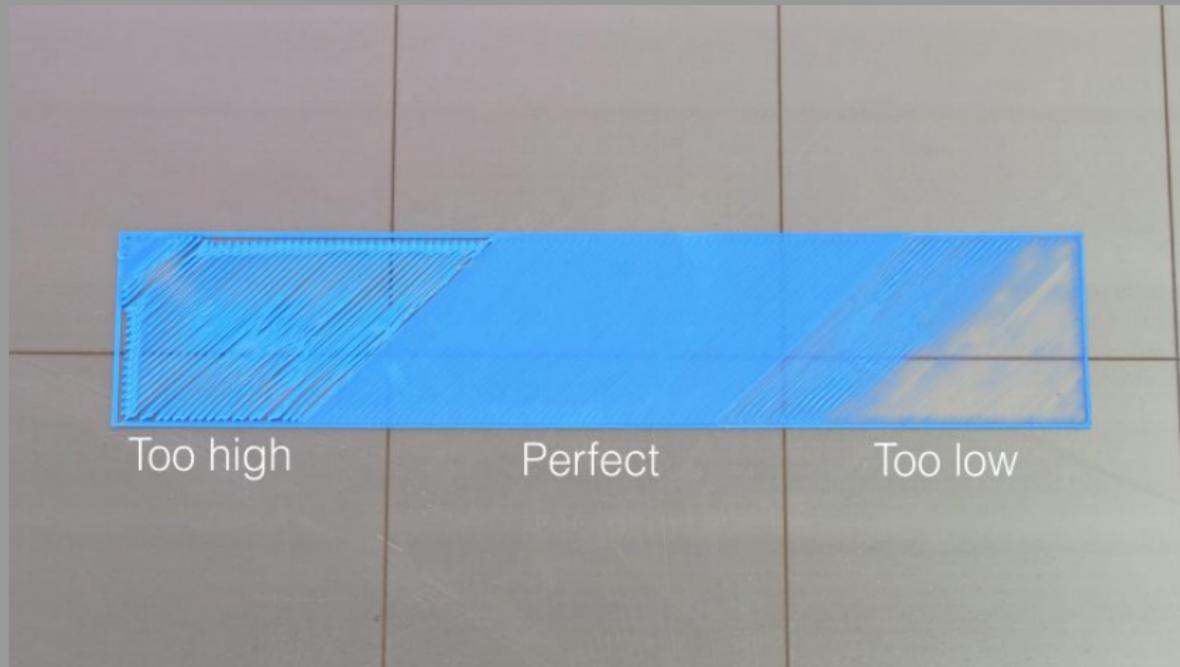
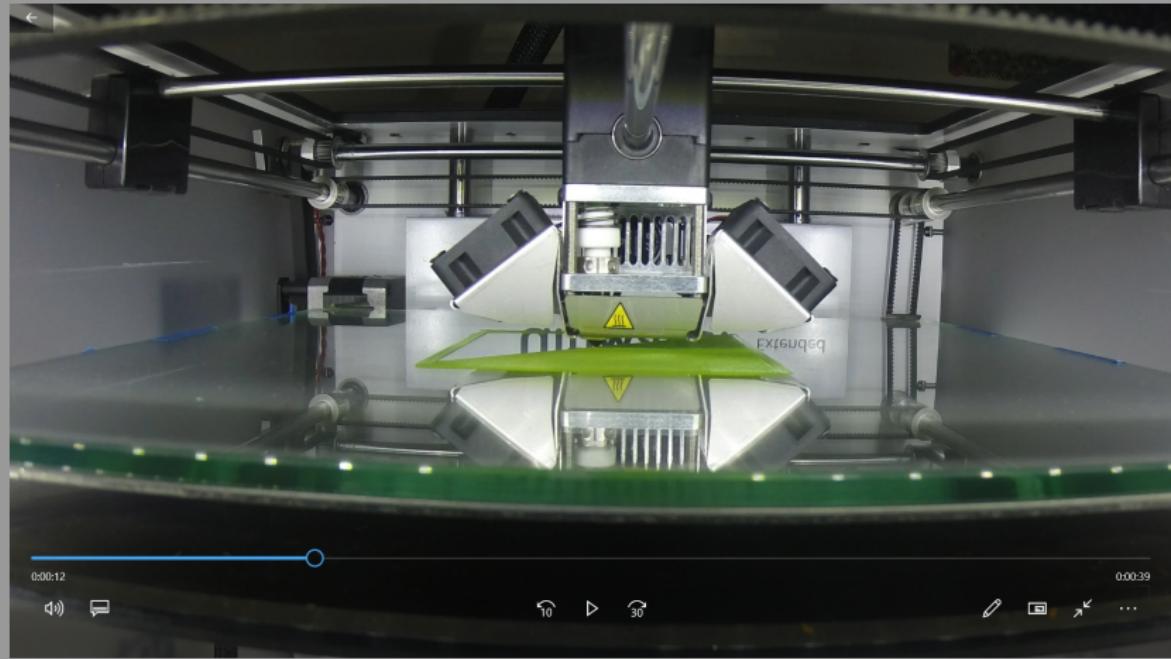


Photo Source: Prusa Knowledge Base

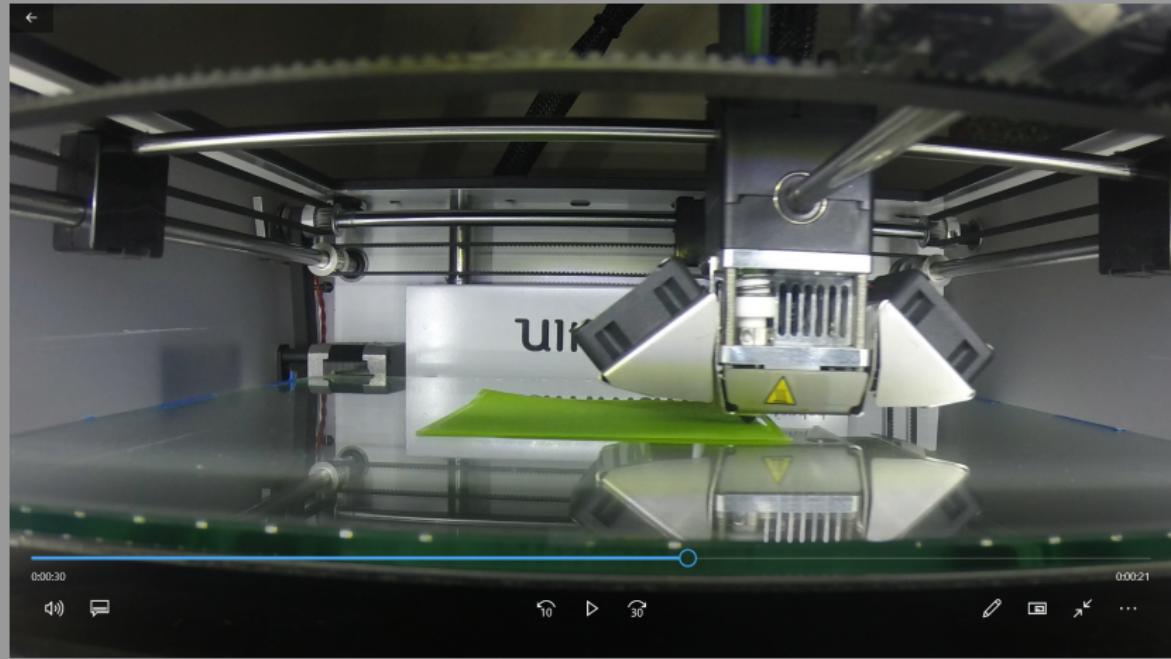
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



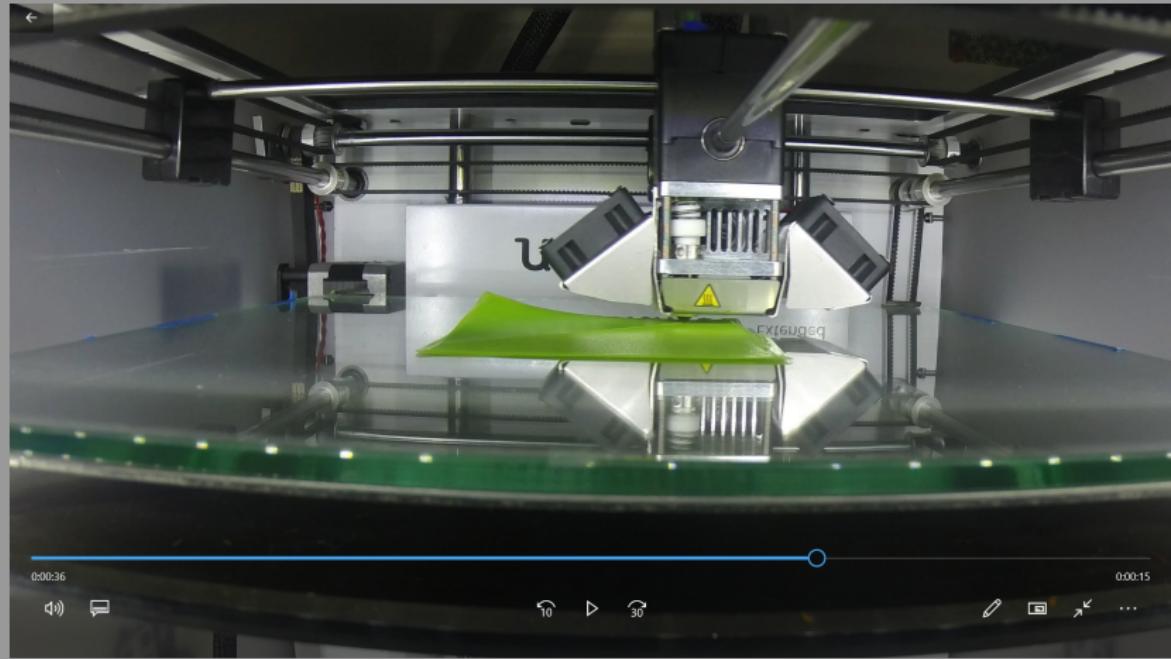
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



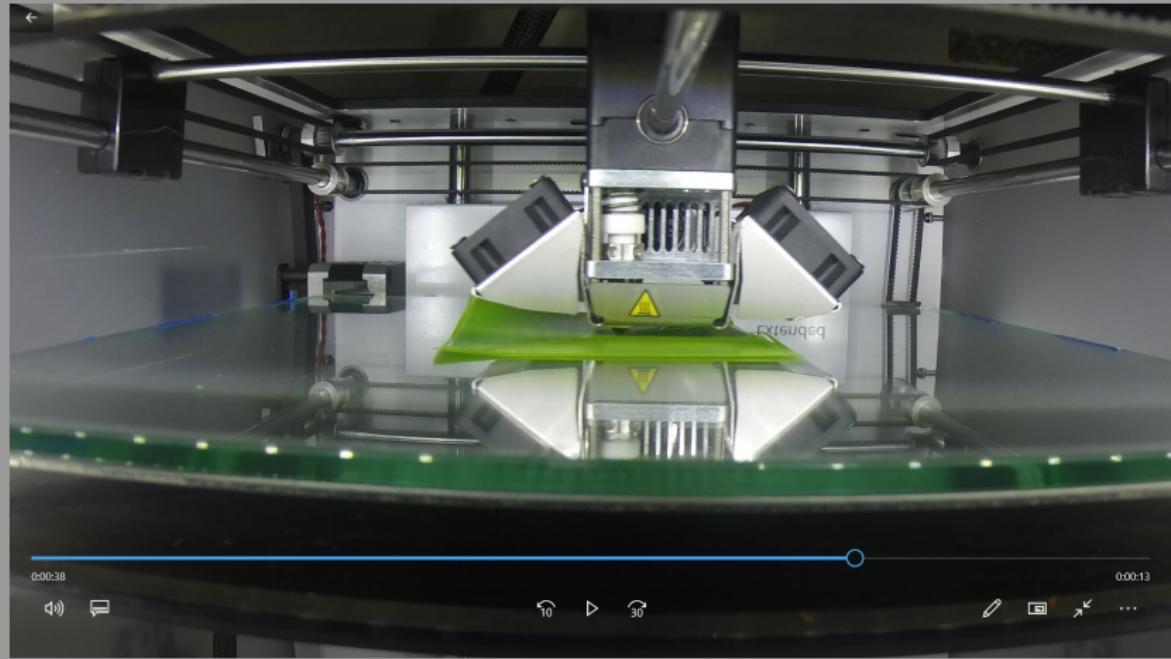
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



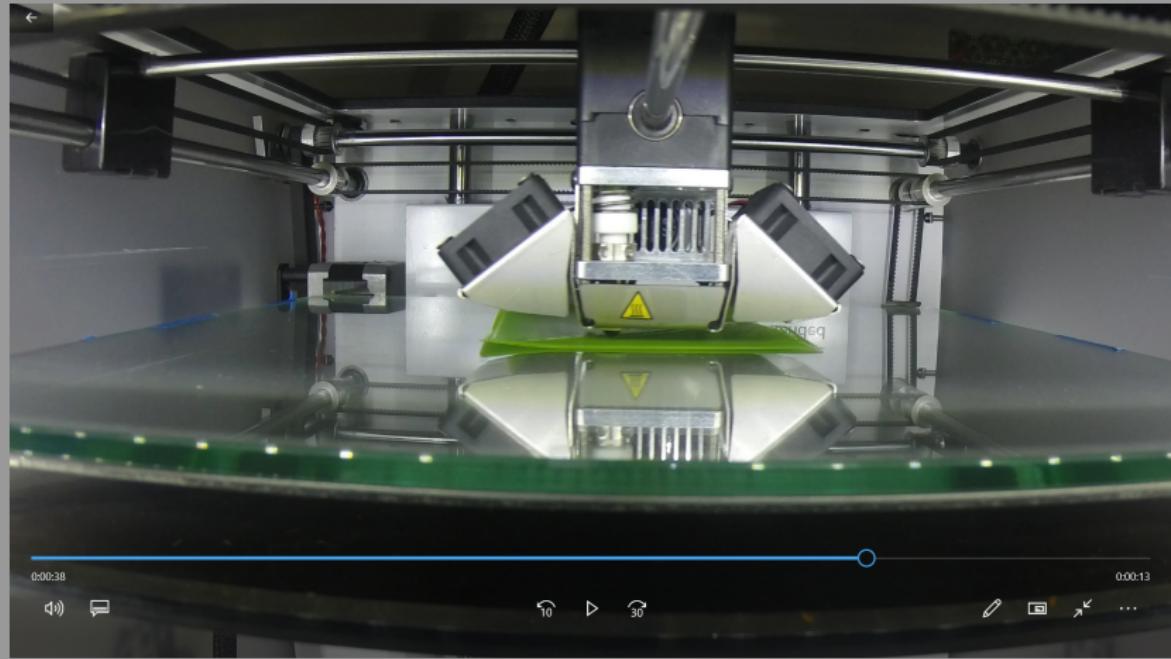
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



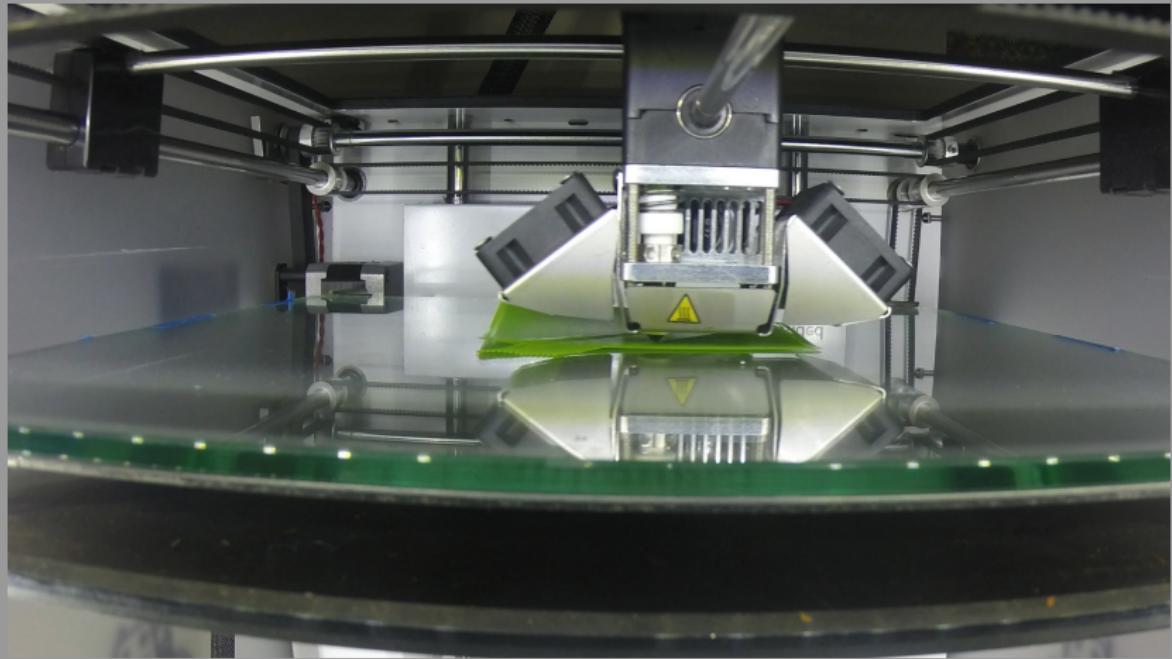
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



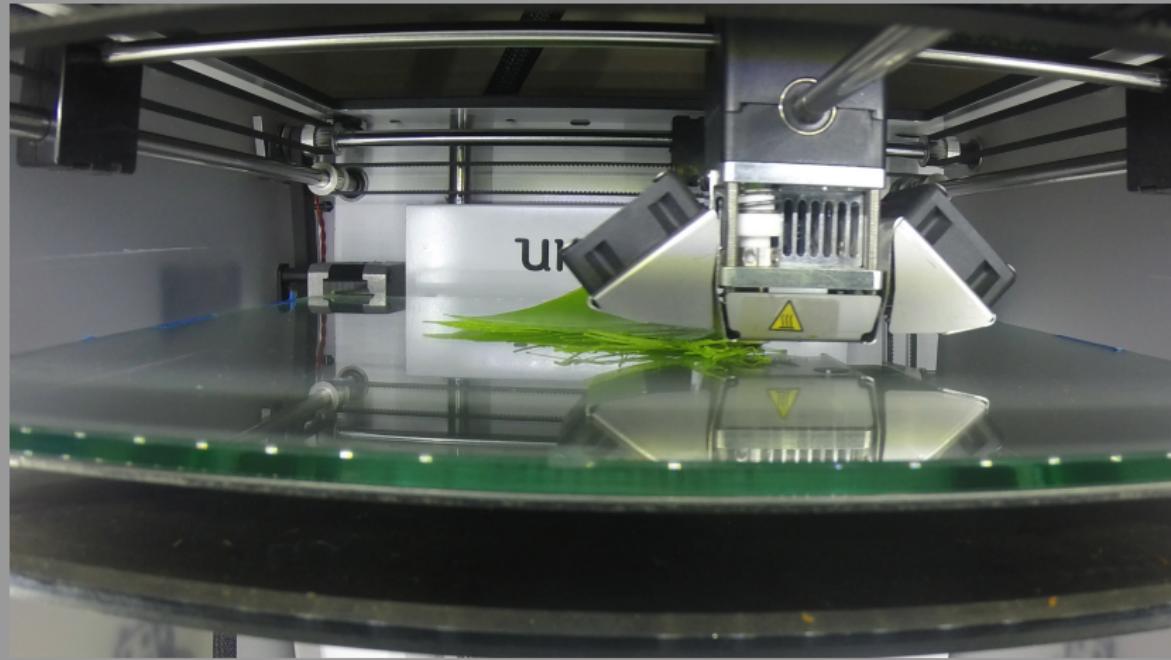
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



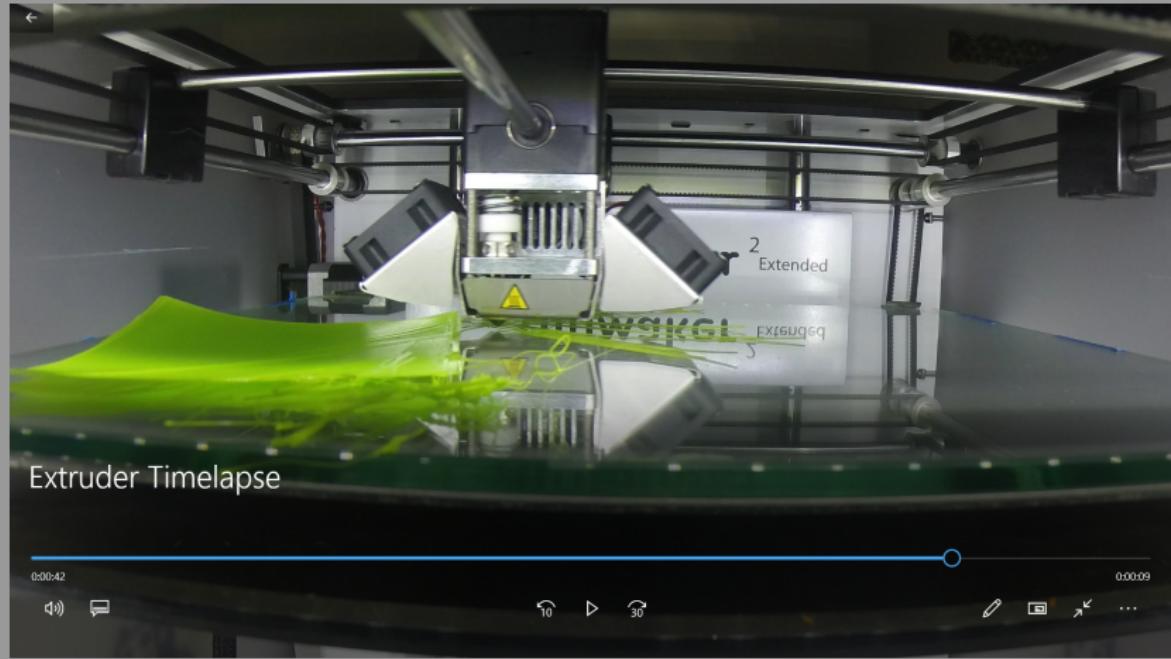
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



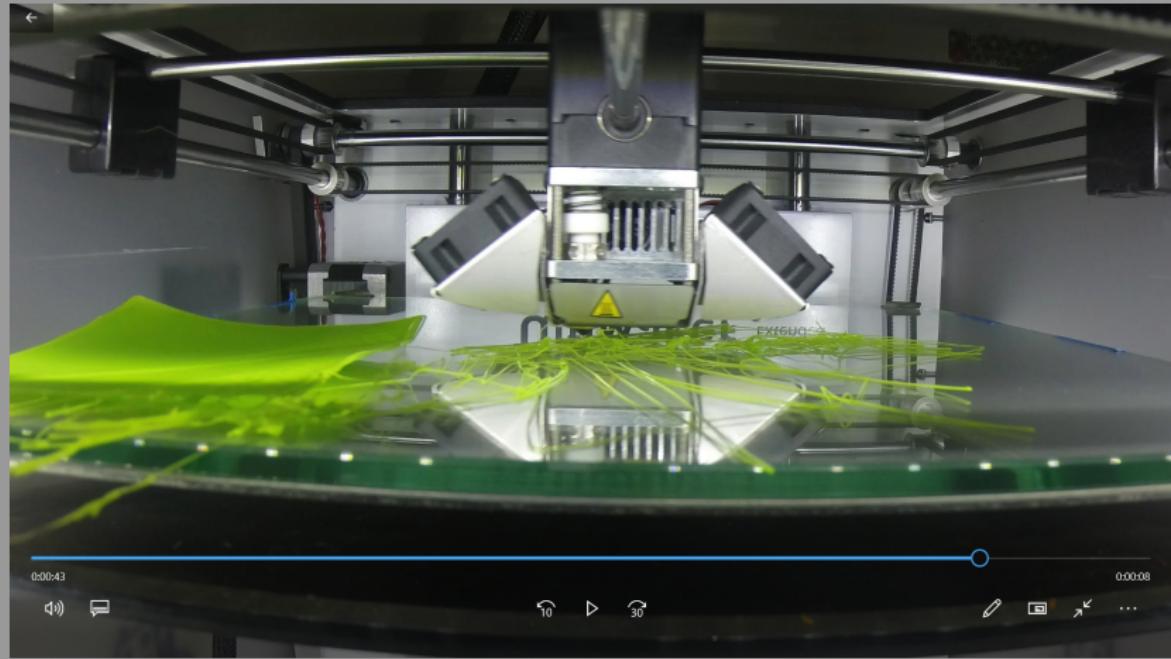
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



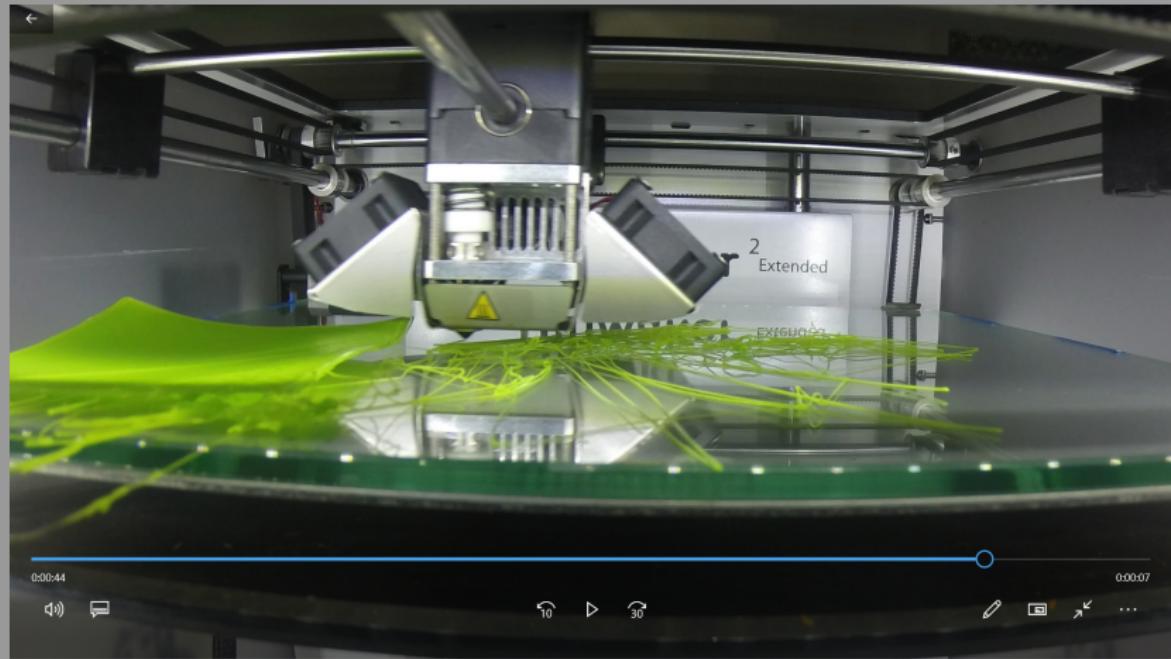
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



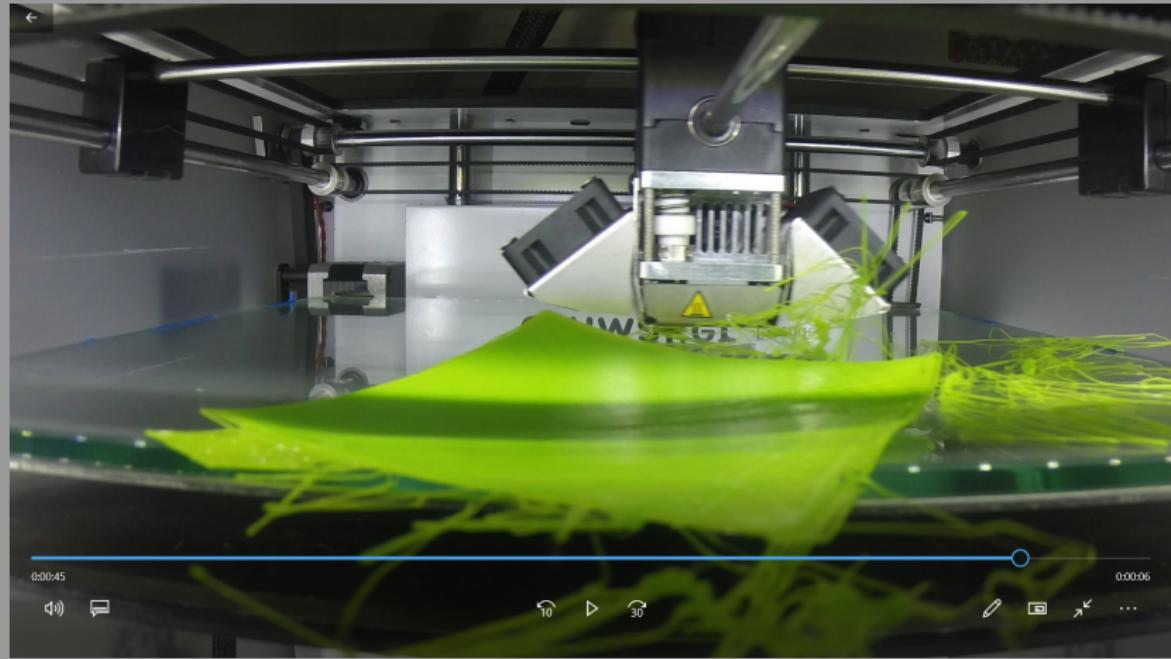
What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



What to look for in the first layer: Adhesion

It is also **very important** that the first layer sticks firmly to the build plate.



Demonstration

Let's run through an example and start a print.

Troubleshooting

That makes it look pretty easy, and it is! But sometimes things go wrong. There are many websites listing common issues and solutions. The one I encounter most often is that the bottom layer doesn't adhere. When this happens there are a few things to do:

- Adjust buildplate height.
- Increase buildplate temperature.
- Slow down the speed of the first layer.
- Use adhesion settings. Build a skirt, brim, or raft.
- Apply water soluble gluestick to printer bed (if the bed is metal or glass).

Initial Layer Speed

The screenshot shows a 3D model of a "MINI" logo being printed. A callout box highlights the "Initial Layer Speed" setting in the print settings panel.

Initial Layer Speed

The speed for the initial layer. A lower value is advised to improve adhesion to the build plate. Does not affect the build plate adhesion structures themselves, like brim and raft.

Affects

- Initial Layer Print Speed
- Initial Layer Travel Speed
- Skirt/Brim Speed

Affected By

- Print Speed

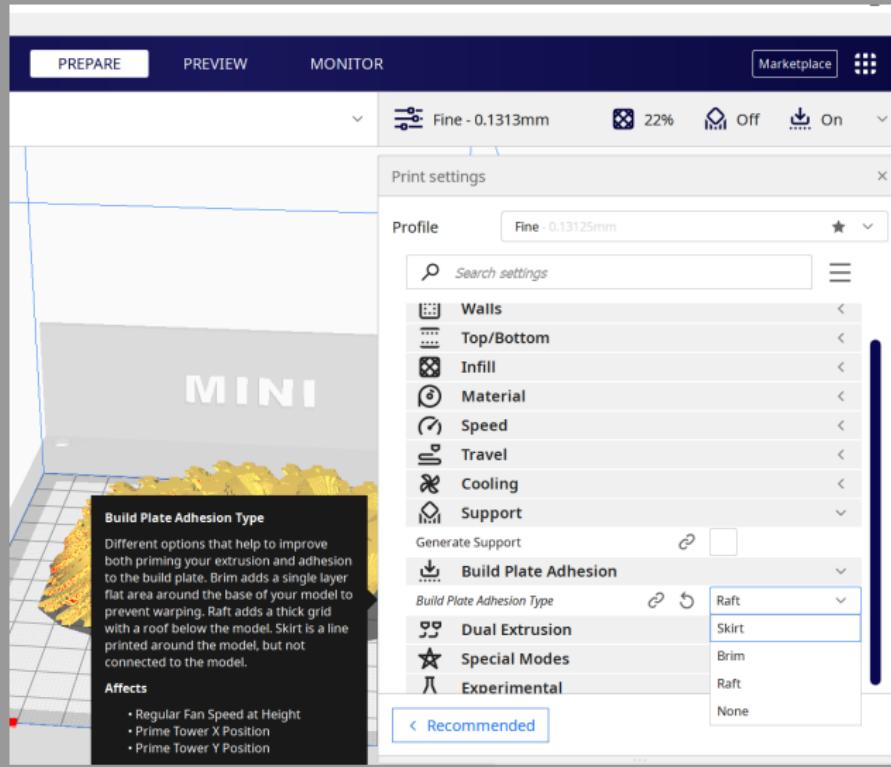
Print settings

Profile: Fine - 0.13125mm

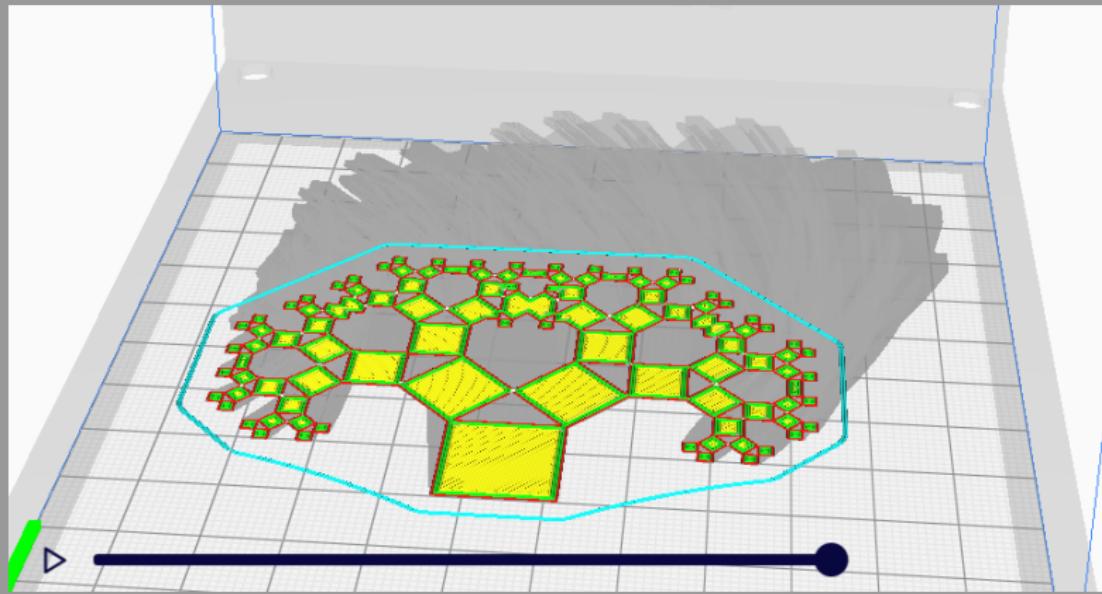
Setting	Value	Unit
Print Speed	50.0	mm/s
Infill Speed	50.0	mm/s
Wall Speed	25.0	mm/s
Outer Wall Speed	37.5	mm/s
Inner Wall Speed	50.0	mm/s
Top/Bottom Speed	20.0	mm/s
Travel Speed	120.0	mm/s
Initial Layer Speed	30.0	mm/s
Skirt/Brim Speed	30.0	mm/s

< Recommended

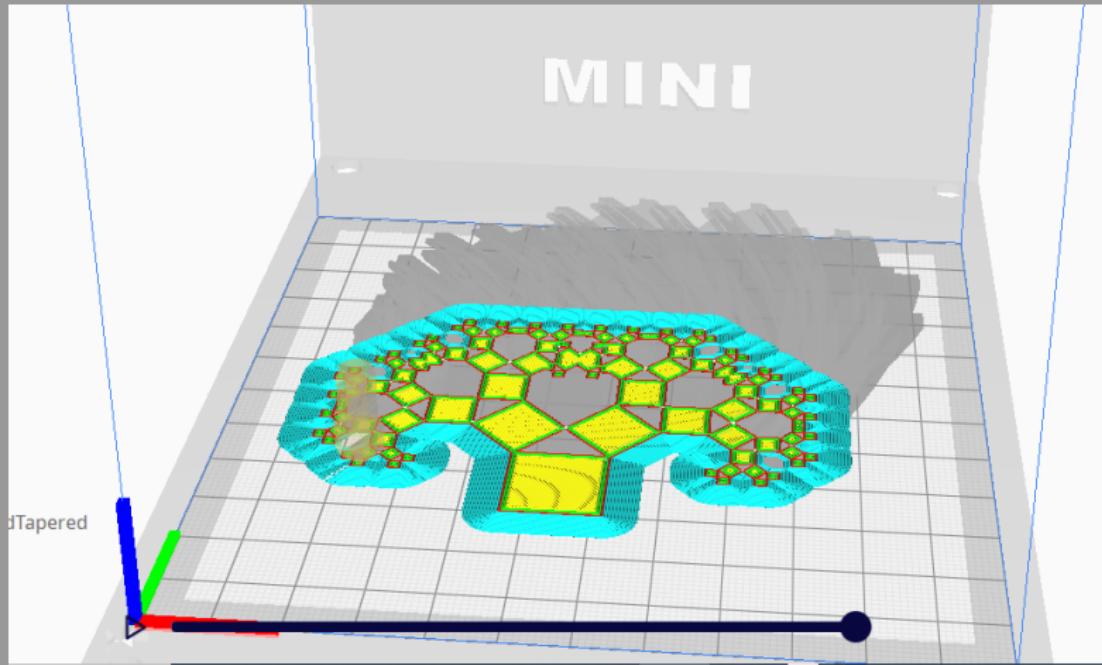
Adhesion: Skirt, Brim, or Raft



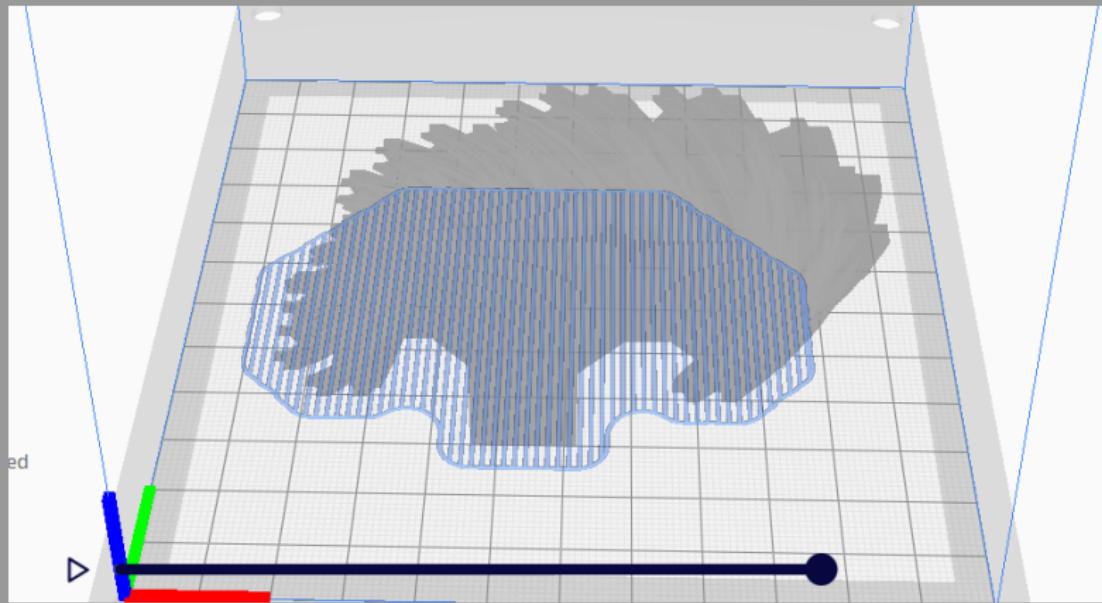
Adhesion: Skirt, Brim, or Raft



Adhesion: Skirt, Brim, or Raft



Adhesion: Skirt, Brim, or **Raft**



Thank you!

