

---

# The Un-definitive Beginners Guide to Additive Manufacturing

**Prepared by:**

Chris Craig, Deputy Director of Information Technology

Selamawit Abraha, Lab Technician

Crystal Atoz, Lab Technician

CLV 3D Maker Center

300 S. 4<sup>th</sup> St. Suite 180

Las Vegas, NV 89101

Created: 2024

---

# Table of Contents

Overview.....	6
What is Additive Manufacturing?.....	6
What is 3D?.....	6
Current Additive Manufacturing Processes .....	7
Sheet Lamination .....	7
Directed Energy Deposition.....	8
Binder Jetting .....	9
Material Jetting .....	10
Powder Bed Fusion .....	11
Vat Photopolymerization .....	12
Material Extrusion .....	13
Types of Material Extrusion Machines.....	14
Delta.....	14
Cartesian .....	15
CoreXY .....	16
Materials Used .....	17
Polylactic acid (PLA).....	17
Polyethylene Terephthalate Glycol (PETG) .....	18
Acrylonitrile Butadiene Styrene (ABS) .....	19
File Formats .....	20
Stereolithography (.stl) .....	20
3D Manufacturing File (.3mf).....	20
Wavefront OBJ (.obj).....	20
OLT Printing Fil (.oltp).....	20
Infills & Shells .....	21
Infills.....	21
Infill Density .....	21
Infill Types.....	22

---

Concentric .....	22
Honeycomb .....	22
Triangles .....	23
Lines.....	23
Rectilinear .....	24
Grid .....	24
Gyroid .....	25
Cubic .....	25
Shells.....	25
Overhangs & Supports .....	26
What is G-Code?.....	27
Syntax .....	27
Comments .....	28
G-Code Command List .....	28
G0 & G1: Linear Motion .....	28
G90 & G91: Absolute & Relative Positioning .....	28
G28 & G29: Auto-Home & Bed-Leveling .....	29
M104, M109, M140, & M190: Set Temperature.....	29
M106 & M107: Fan Control .....	29
Slicers & ideaMaker .....	30
The Maker Community.....	31
Maker Center Rules & Regulations .....	32
Steps for Printing a Model .....	37
Section 1. Create or Find a Design .....	38
1.2 Build Your Own Model .....	38
1.3 Search For a Model.....	38
a. Selecting a Model .....	38
b. Downloading the Model .....	38
Section 2. Loading a File into ideaMaker .....	39
Section 2.1 - Exporting A File (If you created your own) .....	39
Section 2.2 Importing a File into ideaMaker .....	41

---

Section 3. Understanding IdeaMaker.....	43
Section 3.1 Viewing/Panning the Model .....	43
Section 3.2 Moving the Model.....	44
Section 3.3 Rotate the model .....	45
Section 3.4 Scaling the Model .....	46
Section 3.5 Free Cutting the Model .....	47
Use of Free cut tool (dual color printing): .....	48
Sectional 3.6 Supports .....	51
Section 4. Slicing: Preparing your file for Printing .....	53
Section 4.2 Changing Print Parameters Using Simple Mode .....	54
Section 4.3 Advanced Settings (Optional).....	60
Section 4.3.1 Layer .....	60
Section 4.3.2 Extruder.....	61
Section 4.3.3 Solid Fill.....	61
Section 4.3.4 Support.....	61
Section 4.3.5 Cooling.....	61
Section 4.3.6 Temperature .....	62
Section 4.3.7 Speed.....	62
Section 4.4 Previewing Your Model.....	63
Section 5. Naming Convention for File .....	76
Section 6. Uploading Your Print to the Printer.....	76
Section 7. Instructions for Pickup.....	76
Conclusion .....	77
Glossary of Common 3D printing Terms .....	78
123.....	78
A.....	79
B.....	80
C.....	81
D.....	82
E.....	83
F.....	83

---

G.....	84
H.....	85
I.....	86
J.....	87
K.....	87
L.....	87
M.....	87
N.....	88
O.....	88
P.....	88
Q.....	91
R.....	91
S.....	91
T.....	93
U.....	94
V.....	94
W.....	94
X.....	94
Y.....	95
Z.....	95

---

## Overview

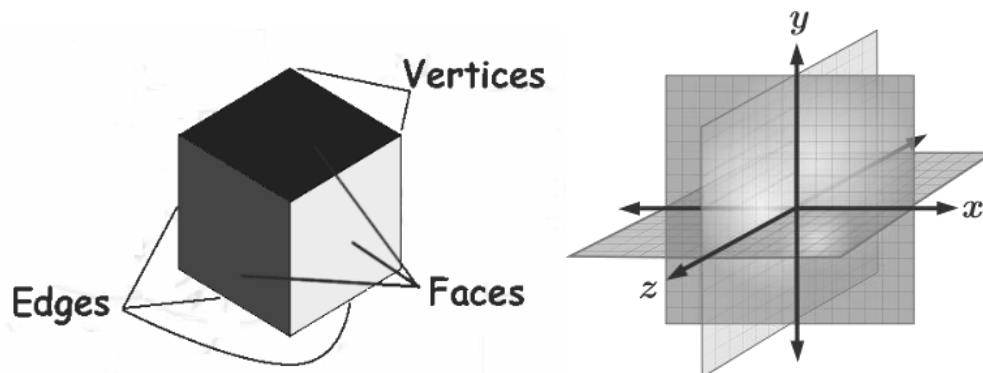
Welcome to the magical world of additive manufacturing. This un-definitive beginner guide to additive manufacturing was put together to give you a general understanding of what additive manufacturing is, how it's used, the methods involved, and ultimately how to use material extrusion additive manufacturing to print your very own models.

## What is Additive Manufacturing?

Additive manufacturing, aka 3D printing, is the process of creating an object by building it one layer at a time. This process involves a program taking the digital file, slicing it into horizontal layers, and providing instructions to the printer on how to print. This process allows users to quickly produce complex designs with great precision. You may think that this is a new technology, but it has actually been around since the 1980's and was referred to as Rapid Prototyping at the time. It is important to note that there isn't a universal solution for every problem. This guide can help you get started and gain more knowledge about 3D printing and learn to find what works best for your designs.

## What is 3D?

All 3D shapes have three measurements: length, width, and height, represented by the dimensions  $x$ ,  $y$ , and  $z$ . Along with this, there are three attributes: face, edge, and vertex. The face is each single, flat, or curved surface. The edge is the line that connects two faces, and the vertex is each corner where three faces meet.



---

# Current Additive Manufacturing Processes

Before we jump into material extrusion fundamentals, we will go over the various other types of additive manufacturing processes that exist and their applications.

The following is a short list of current additive manufacturing processes:

## Sheet Lamination

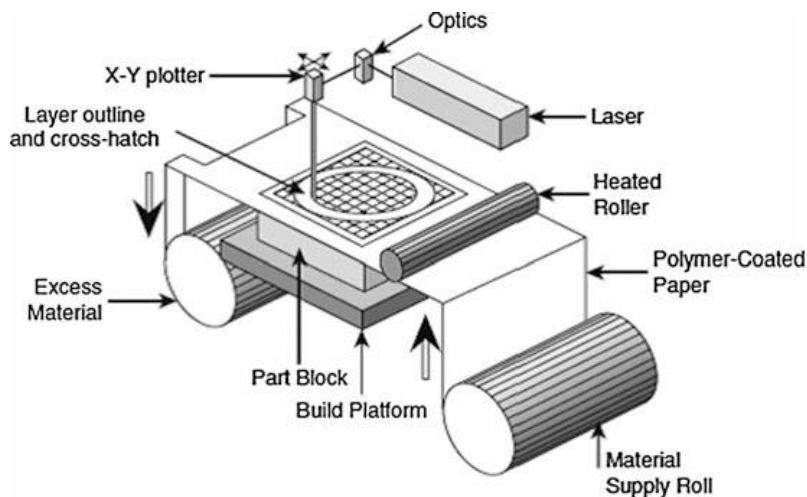
A rapid prototyping process in which sheets of material are joined together to create an object. In this process a roll of build material is introduced to the build surface. Next, a bonding adhesive is applied to the build material and a heated roller moves over the surface to bind the material. A laser beam is used to cut the sheet and remove any excess material. The next layer of the material is positioned on the print bed. The process is repeated until the object is complete.

### Materials Used:

- Metal

### Industry Applications:

- Non-functional Prototypes
- Casting Molds
- Architectural Designs



[https://www.researchgate.net/figure/Schematic-diagram-of-the-sheet-lamination-process-Gibson-et-al-2015\\_fig4\\_334169112](https://www.researchgate.net/figure/Schematic-diagram-of-the-sheet-lamination-process-Gibson-et-al-2015_fig4_334169112)

---

## Directed Energy Deposition

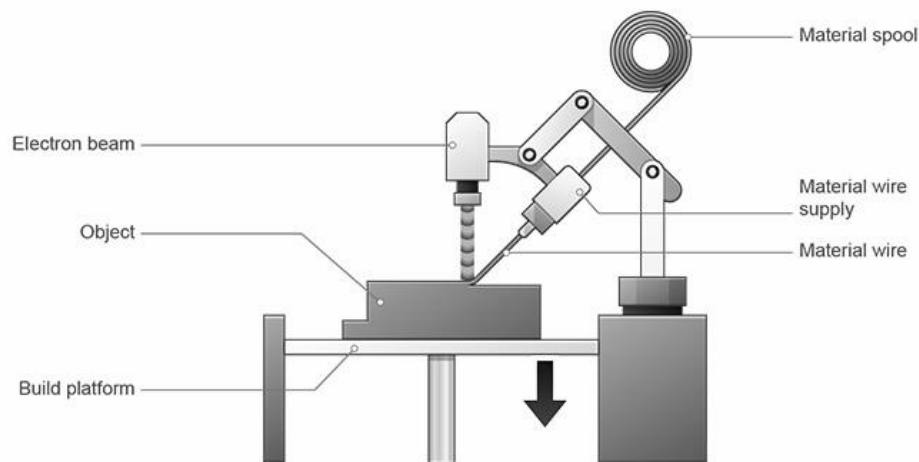
In this process, a heat source is used, such as a laser or electron beam, to melt metal powder or wire that is fed through a nozzle. This melted metal pool is then applied to a specific surface, via CAD<sup>1</sup> data, where it hardens into a solid structure. It is most commonly used to repair or add additional features to existing parts.

### Materials Used:

- Metal

### Industry Applications:

- Aerospace
- Military
- Automotive



[https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F\\_ProcessAdditive\\_DirectedEnergyDeposition@2x.png](https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F_ProcessAdditive_DirectedEnergyDeposition@2x.png)

---

<sup>1</sup> Computer-Aided Design: the use of computers to help design 3D models



---

## Binder Jetting

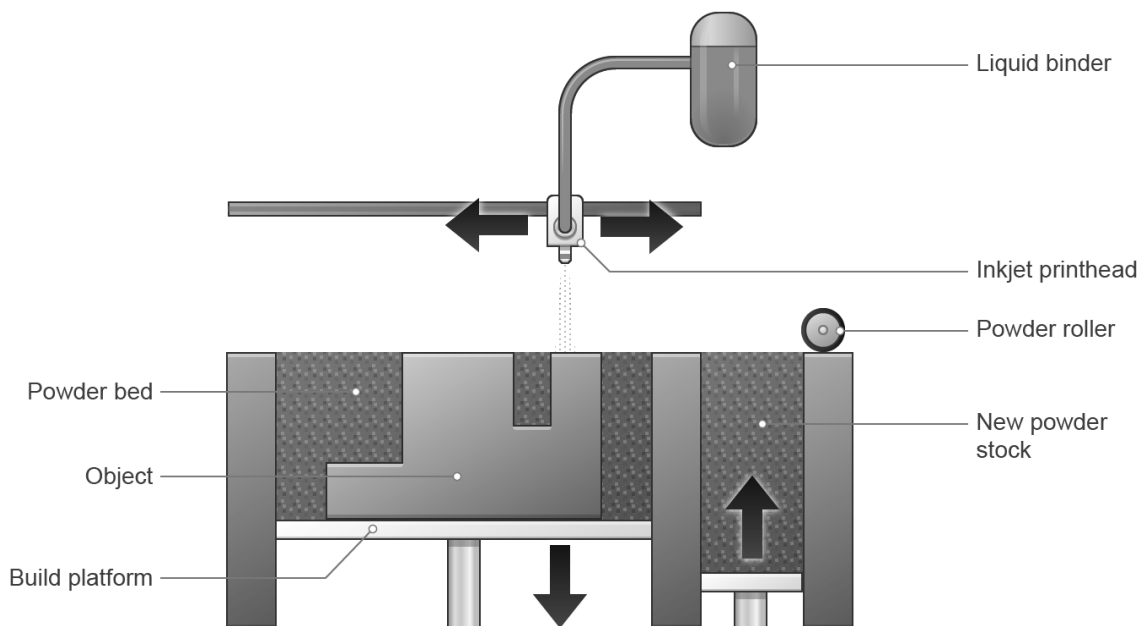
Building material is applied to the print bed. Next, the print head selectively binds the build material according to the parts 3D CAD data. The build platform is then lowered by the thickness of a single layer. The process repeats, connecting the layers of powder into a finished part.

### Materials Used:

- Ceramics
- Plaster
- Engineered Plastics
- Metals
- Sand

### Industry Uses:

- Construction
- Aerospace
- Arts



2018 © Dassault Systèmes

[https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F\\_ProcessAdditive\\_BinderJetting@2x.png](https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F_ProcessAdditive_BinderJetting@2x.png)

---

## Material Jetting

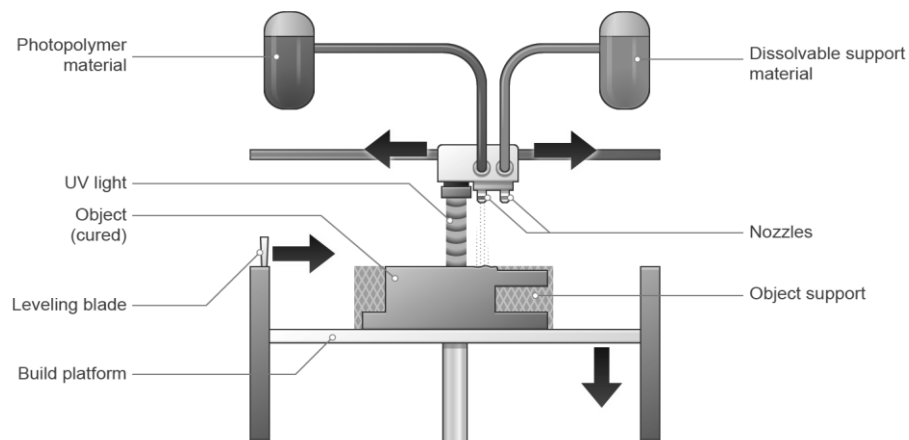
A full-color additive manufacturing technique where droplets of thermoplastic<sup>2</sup> are selectively deposited using drop on demand (DOD) technology. This process starts by the print head depositing a layer of liquid photopolymer<sup>3</sup> resin on the print bed. An ultraviolet (UV) source moves over the deposited layer, binding the build material. The build platform then moves down by the thickness of one layer, and another layer of material is added over the previous layer. The process repeats until finished.

### Materials Used:

- Ceramics
- Plaster
- Metals
- Photo Polymers
- Wax

### Industry Uses:

- Medical
- Electronics
- Automotive
- Art
- Design



2018 © Dassault Systèmes

[https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F\\_ProcessAdditive\\_MaterialJetting@2x.png](https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F_ProcessAdditive_MaterialJetting@2x.png)

---

<sup>2</sup> A class of polymer that can be softened by heating and becomes pliable at a certain elevated temperature.

<sup>3</sup> A class of light-sensitive resins that solidify when exposed to ultraviolet (UV) light.

---

## Powder Bed Fusion

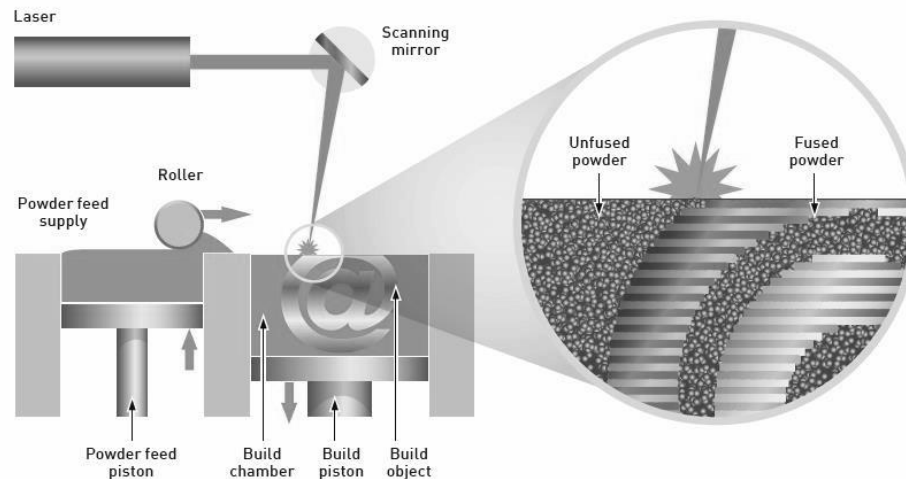
This technology uses a heat source, such as an electron or laser, to melt and join material powder to create three-dimensional objects. First, a thin layer of powdered material is spread over the overbuild stage. Next, the machine applies a heat source to selectively scan over and fuse the powder into a first cross-section of the part using the 3D CAD data. The build platform is then lowered by the thickness, and a new layer of powdered material is spread onto the previous layer. The process repeats until finished.

### Materials Used:

- Ceramics
- Engineered Plastics
- Metals

### Industry Uses:

- Industrial
- Commercial
- End-use Production Parts
- Professional Prototyping



<https://www.materials-talks.com/wp-content/uploads/2018/11/powder-bed-fusion.jpg>

---

## Vat Photopolymerization

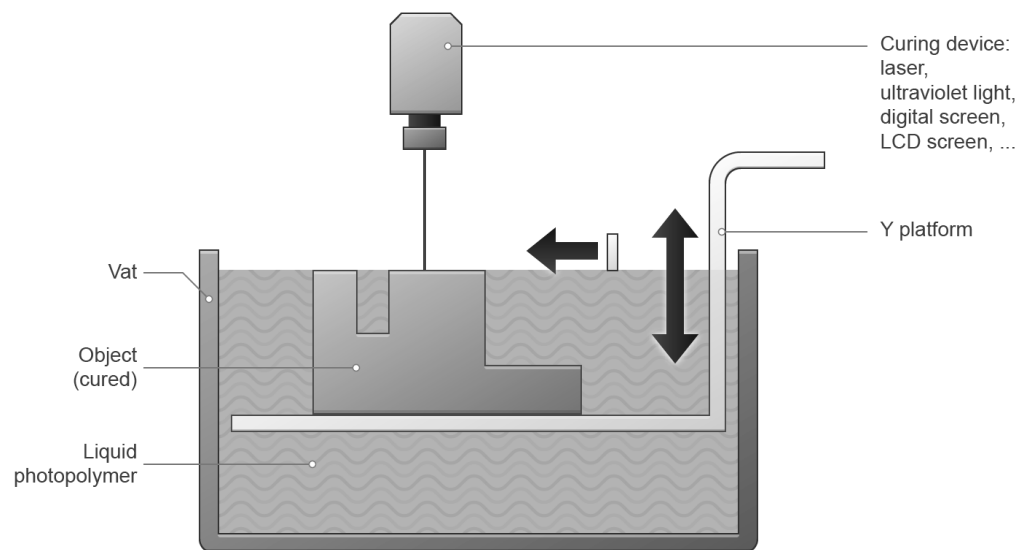
A vat or container, filled with photosensitive liquid resin, is used with a light source to create solid objects. To start, the build platform lowers from the top of the vat. An ultraviolet (UV) light source selectively targets the liquid resin, according to the pattern that is defined by the 3D CAD data, and undergoes a chemical reaction that solidifies the liquid—known as photopolymerization.

### Materials Used:

- Biomaterial
- Ceramics
- Engineered Plastics
- Photopolymers

### Industry Uses:

- Home/Hobby
- Medical
- Prototyping
- Jewelry
- Art
- Design



2018 © Dassault Systèmes

[https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F\\_ProcessAdditive\\_Photopolymerization@2x.png](https://make.3dexperience.3ds.com/resources/20230922T084823Z/en/webapps/MP3DPWikiProcessesDatas/assets/images/additive/F_ProcessAdditive_Photopolymerization@2x.png)

# Material Extrusion

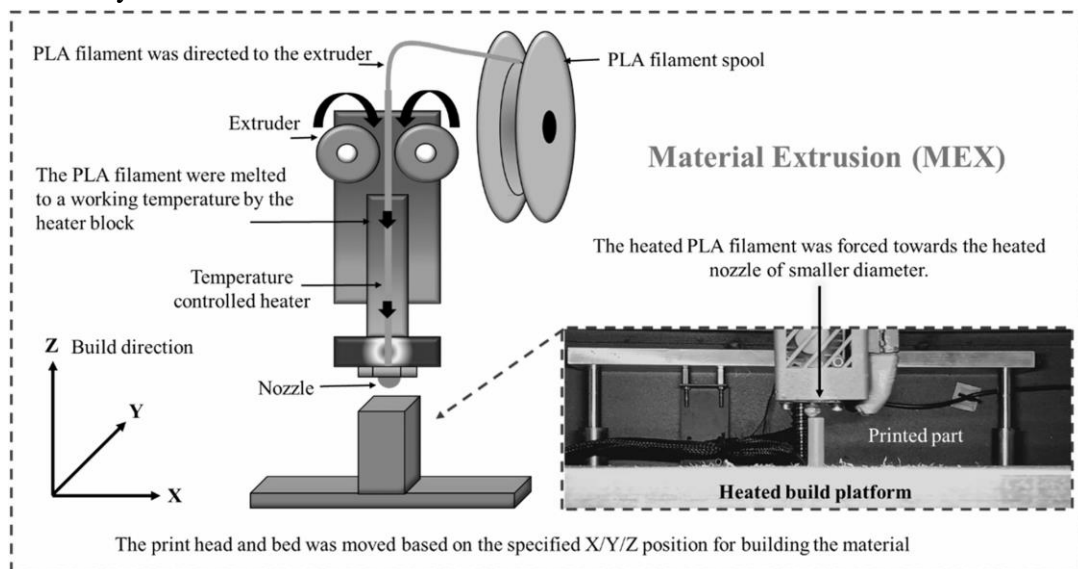
This is the most common type of 3D printing in home applications. This is the method our printers in the lab use. In this method, a thermoplastic material is pushed through a heated extrusion nozzle and deposited layer by layer to build an object. The most commonly used additive material extrusion process is fused deposition modeling (FDM). In FDM physical objects are made by building up successive layers of material with an extruded thermoplastic material.

## Materials Used:

- Concrete
- Biomaterial
- Ceramics
- Engineered Plastics
- Metals
- Photopolymers

## Industry Uses:

- Home/Hobby
- Medical
- Prototyping
- Art
- Construction
- Aerospace
- Automotive
- Electrical
- Jewelry
- Culinary



[https://media.springernature.com/full/springer-static/image/art%3A10.1038%2Fs41598-023-33510-7/MediaObjects/41598\\_2023\\_33510\\_Fig1\\_HTML.png?as=webp](https://media.springernature.com/full/springer-static/image/art%3A10.1038%2Fs41598-023-33510-7/MediaObjects/41598_2023_33510_Fig1_HTML.png?as=webp)

---

## Types of Material Extrusion Machines

There are various types of Material Extrusion machines on the market that vary in size and shape. While there are striking differences in the machines and capabilities, they all fundamentally operate the same way by extruding a given material through a nozzle layer by layer to create a 3D object. Below are a few machine designs that you may see on the market.

### Delta

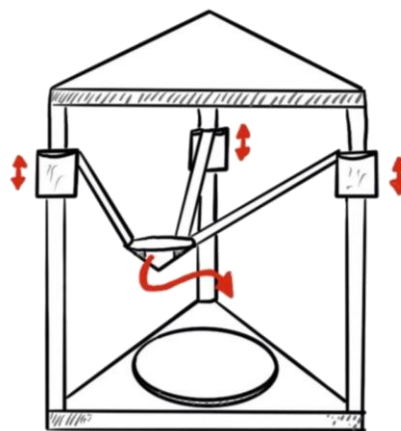
Delta style printers have a round static base with three arms that move the print nozzle on the X, Y, and Z axis. The print head is mounted on three movable arms that push or pull the print head to control its position.

#### Pros:

- Fast speeds
- Ability to build tall objects
- Greater precision and quality

#### Cons:

- Takes up a lot of vertical space to accommodate the arms
- Sometimes requires assembly
- Difficult to calibrate, fix, and upgrade
- Small build area



---

## Cartesian

Otherwise better known as “bed slingers”, these are the most common style of material extrusion printers on the market. They typically have a print bed that moves forward and backward on the Y-plane while the nozzle moves right and left on the X-plane and up and down on the Z-plane.

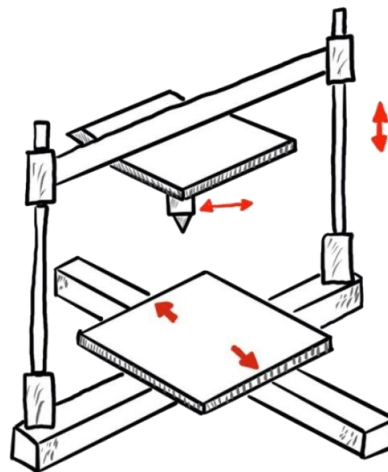
There are an endless number of manufactures, sizes, and designs available from name brands to build your own where the options are almost unlimited. Some printers have multiple nozzles for dual and multi-color printing. While less expensive than most printers on the market, Cartesian printers can print in relatively high detail for a Material Extrusion printing.

### Pros:

- More affordable
- Easy to learn

### Cons:

- Low print speed
- Vibration and material separation risk due to movement of the print bed
- Large and heavy frame
- Limited Print height



---

## CoreXY

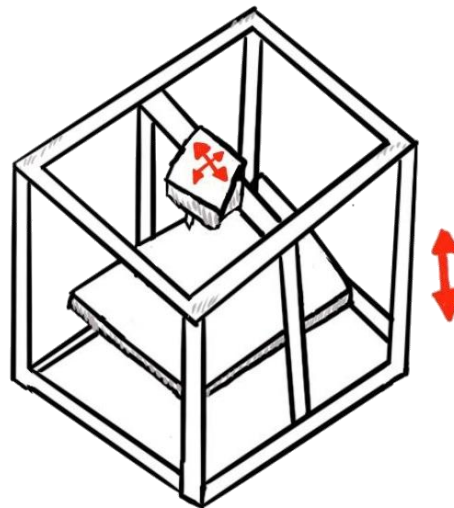
These are the types of printers that we have here at the Maker Center. They are relatively new, first made in 2013. It operates with the nozzle moving right and left in the XY-plane while the print bed moves up and down in the Z-plane. This operation incorporates a system of belts and pulleys to move the print head along the XY-plane. There are two motors that move one belt each to position the print head.

### Pros:

- Large print area
- Controlled environment for heating and cooling
- Few vibrations
- Relatively high accuracy

### Cons:

- Relatively expensive
- Higher risk of print failures





---

## Materials Used

Science and the advances in chemical technology has really opened the market up for material extrusion printers providing makers with a wide range of colors, material compounds, and textures. This allows our minds to be more open to possibilities creating an endless platform for fun and creativity.

While various materials, including chocolate, cement, paste, rubber, and different types of thermoplastics, can be used in material extrusion printing, exotic and expensive materials are typically unnecessary for prototyping or creating playful items. 3D printers use filament—a raw material that is spooled and fed into the printer to be melted and deposited onto the print bed. Measured by diameter in millimeters, there are two sizes of filament: the less common and almost defunct 3mm, and the more common industry standard 1.75mm diameter filament.

There are seemingly endless options of colors and finishes such as gloss, matte, satin in the most basic filament materials. For the purposes of this training, we will cover the three most common filament materials used in material extrusion printing.

### Polylactic acid (PLA)

This material is used by the printers in the Maker Center. It is by far the most common and widely used filament used by makers. It is sourced from natural and renewable resources.

#### Pros:

- Low cost
- Full spectrum of colors that include bi and tri colors in one strand
- Matte, gloss, and silk finishes
- Can have other materials added such as wood, metal, rock for different textures and capabilities such as polishing or staining
- Relatively safe to work with due to its low VOC<sup>4</sup> and being odorless
- Can be printed with low or no heat bed temperature
- Safe to sand during post processing, although PPE<sup>5</sup> is always recommended
- Easy to paint/finish and bond through plastic welding and other adhesives
- Low melting point: 90°-210°Celsius
- Good print quality

#### Cons:

- Lower strength and durability

---

<sup>4</sup> Volatile Organic Compounds that are emitted as gasses from certain solids or liquids.

<sup>5</sup> Personal Protective Equipment used to minimize exposure to hazards

- 
- Poor heat resistance due to low melting point
  - Moisture sensitivity
  - Resistant to acetone and other chemicals therefore not as easy to smooth during post processing
  - Surface is somewhat soft and prone to abrasion

## **Polyethylene Terephthalate Glycol (PETG)**

PETG is an amazing material for makers who need a material that is stronger and can withstand higher environmental temperatures. Second only to PLA, PETG is very common and also comes in many colors and variations to meet the maker's needs.

### **Pros:**

- Relatively high impact strength less prone to fracture
- Thermally stable
- Hard by touch but elastic from a shock load perspective
- Maintains a hardness that results to being less abrasive than softer filaments like PLA
- Almost odorless
- Easy to bond with plastic welding and other adhesives
- Higher melting point
- Inexpensive
- Less potential for warping
- Approved by the FDA<sup>6</sup> as food-safe
- Fairly low printing temperature around 230°-250° Celsius
- Waterproof

### **Cons:**

- Still can warp or crack
- Post Processing can be more difficult
- Prone to oozing and stringing
- Harder to store in high humidity environments because of its high hygroscopicity<sup>7</sup>
- Not great at bridging

---

<sup>6</sup> Food and Drug Administration responsible for protecting the public health by ensuring safety standards.

<sup>7</sup> The material's ability to absorb moisture from the environment.

---

## Acrylonitrile Butadiene Styrene (ABS)

ABS has fallen in popularity over the years with all the new filaments hitting the market that are easier to work with, less harmful to the environment and the maker. ABS is still used today and some makers still like to use it. However, with advances in other materials like PETG we are not seeing ABS being used as much as it once was.

### Pros:

- Post processing smoothing and binding can be easily done with Acetone
- Sturdy and hard once printed
- Can be used in many applications due to its hardness and higher melting point
- Longer lifespan

### Cons:

- Difficult to print
- Does not stick well to the print bed
- Requires a high printing temperature
- Requires a heated print bed and enclosed printer
- Highly susceptible to warping due to shrinkage when cooling
- Emits harmful fumes during printing
- Bad for the environment as it is made from oil and requires PPE during post processing
- Few options for color and finishes

As mentioned at the start of this section this is nowhere near an exhaustive list of the types of filaments available on the market, and we could easily spend several days just discussing the various types of materials that are available. The list of material options continues to grow daily which is very exciting. Every new product that hits the market creates more opportunities for makers to create something new and astonishing to the rest of the world.

---

## **File Formats**

When importing a model into the slicing software, there are several different file formats that are accepted. At a high level each file is made up of a series of linked triangles that describe the surface geometry of the 3D model. Any of the following file formats will work fine for most 3D printing in the Makers Center, especially for single color, single component, non-complex designs/models.

### **Stereolithography (.stl)**

Although popular and rather universal in the 3D printing world, they are not very flexible files and are known to be rather large for the little informational data they can actually maintain.

### **3D Manufacturing File (.3mf)**

They are very robust, can store not only triangle data but also colors and material information. Despite their smaller file size, they deliver full fidelity models for 3D manufacturing. If you can, use .3mf.

### **Wavefront OBJ (.obj)**

This is another excellent file supporting the 3D manufacturing process as it stores multiple colors, texture information, and also the actual geometry of the given 3D object.

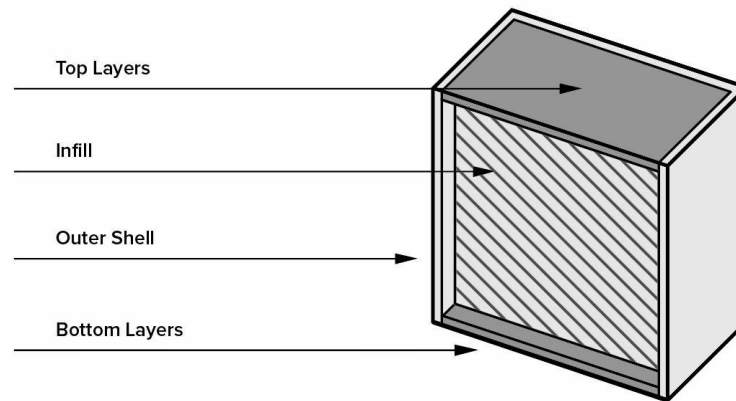
### **OLT Printing Fil (.oltp)**

This is a less common file type, but it is still accepted by our slicer, ideaMaker.

---

## Infills & Shells

A print can be broken down into 4 sections: shells, bottom layers, top layers, and infill. All of these sections can be altered. These alterations can affect the speed of the print, amount of material used, and the overall performance. Overall, shells and infill play an important role on the quality, appearance, and function of your print.

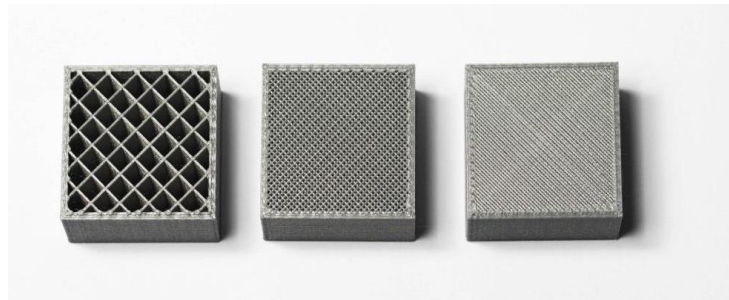


## Infills

The infill is the internal structure of the print. Rarely, you find models that are completely solid. Infills can have varying structures and shapes that can overall affect the print time, weight, strength, material usage cost, and flexibility of a print.

## Infill Density

The strength of a model is determined by its infill density in form of a percentage value. Understanding the application of a print can help the designer to find an optimal infill percentage. Usually, a value between 15% and 20% should be acceptable for most applications.



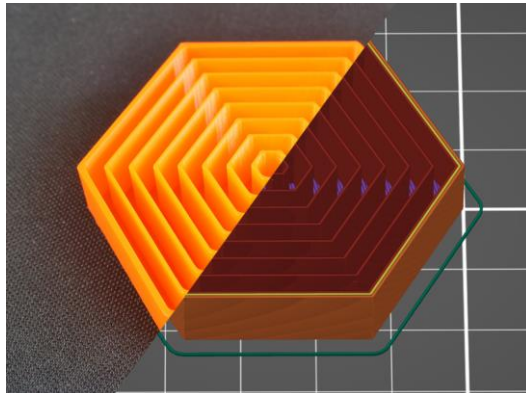
---

## Infill Types

The following are different infill patterns specifically available on ideaMaker at the lab for makers to use.

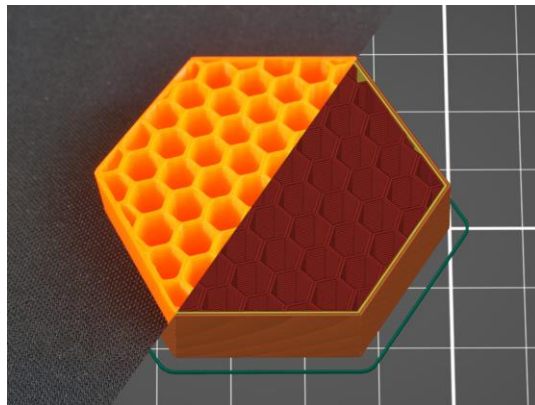
### Concentric

In this pattern, the infill traces the model perimeter lines and makes them smaller as it moves towards the center. This pattern can be useful for transparent parts or flexible parts. Although, print time can be higher.



### Honeycomb

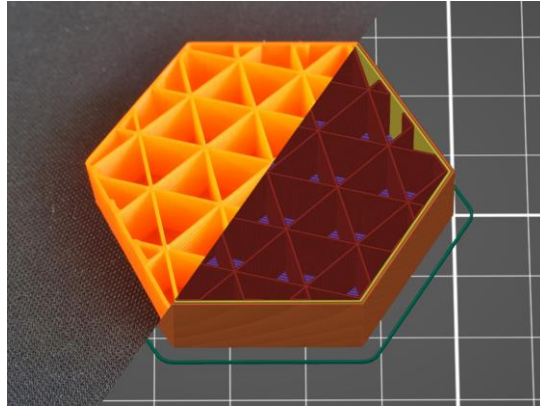
This is one of the more popular patterns. It has a quick print time, very strong, and can provide strength all-around.



---

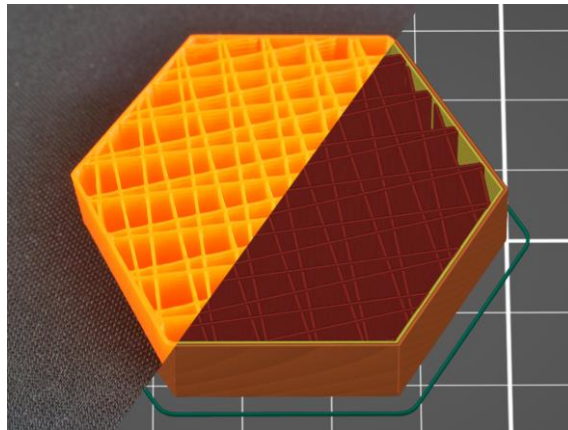
## Triangles

This pattern is used when strength is needed in the direction of the walls. It takes a little more time to print.



## Lines

This infill does not have any crossing paths in one layer. These paths are similar to rectilinear, but they are not parallel<sup>8</sup> and instead print at an acute<sup>9</sup> angle.



---

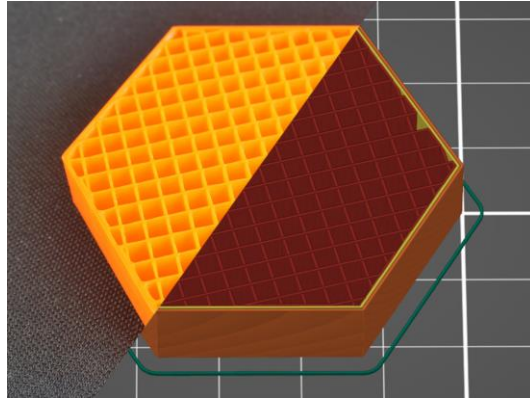
<sup>8</sup> Lines that never cross.

<sup>9</sup> An angle less than 90 degrees.

---

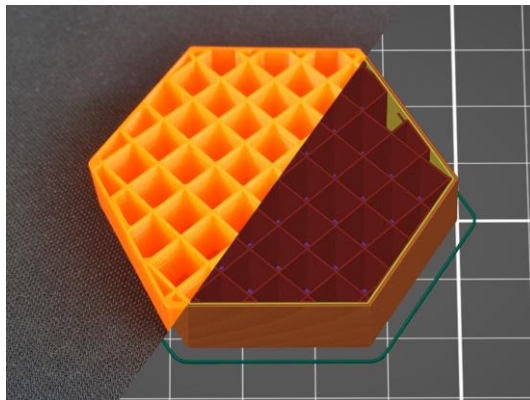
## Rectilinear

This is the standard infill pattern. It is strong in all directions, fast print time, saves material, and requires the least amount of bridging<sup>10</sup> across the infill pattern. Compared to Grid, it has double the support.



## Grid

This is one of the simplest and fastest infills. It is printed in both directions and rotated by 90° in each layer. Is more solid than rectilinear, but can have a risk of print failure due to crossings.



---

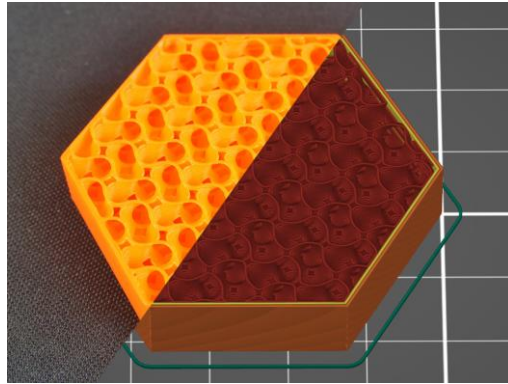
<sup>10</sup> Plastic that needs to be extruded between two points without any support from below.



---

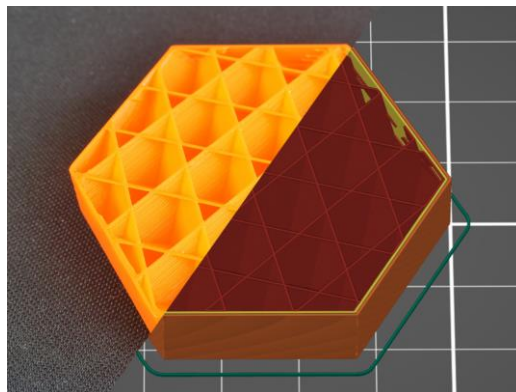
## Gyroid

This is our default here at the Maker Center. This is one of the best infills because it provides great support in every direction, can be printed fast, saves material, and doesn't cross itself. On another note, it looks great!



## Cubic

This pattern creates paths that cross with each other within one layer and create cubes that are oriented with one corner facing down. This results in creating numerous air pockets that can maybe serve as heat insulation or the ability to float.



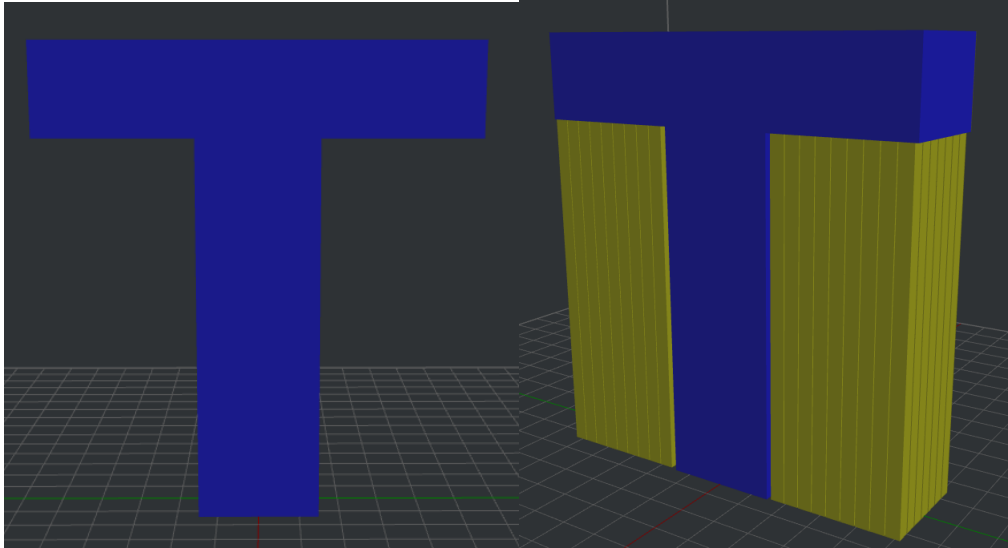
## Shells

There are two types of layers on the outside of a print: the bottom layers and the top layers. The bottom layers are exposed to the outside of the model and are facing the build plate. The top layers are exposed to the outside of the model, facing upwards, towards the nozzle, and will have the best finish. There are several key design considerations for optimizing shells. You can add strength to your model by increasing the shell thickness. Added, increasing the shell thickness is necessary if you plan on sanding or chemical smoothing during post processing. On the downside, increasing its thickness can increase the build time and materials required to print.

---

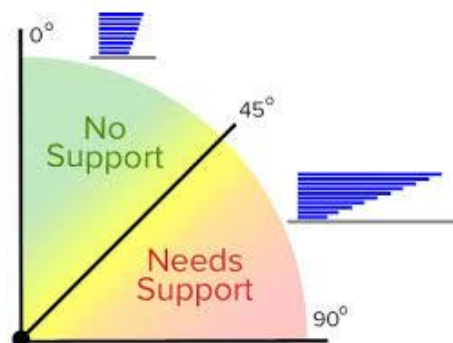
# Overhangs & Supports

Overhangs are a part of something that sticks out over another thing. If your overhang is too steep in your model, you may have issues like drooping and curling. Drooping occurs when the plastic is pulled down by gravity before it solidifies. Curling occurs where the plastic is cooled insufficiently and starts to bend upwards.



To prevent issues caused by overhangs, temporary structures, called supports, can be printed underneath to prevent them from collapsing during printing. Once the build is complete, the supports must be manually removed during post-processing.

Supports are needed when the overhang is greater than  $45^\circ$ . Although, some printers are capable of good print quality at steeper angles. A general rule of thumb is to add supports above  $45^\circ$ . Our Slicer, ideaMaker can auto-detect and auto-generate supports. This is covered further in the how to print steps.



Supports can use a lot of material and after removal, can leave blemishes on your model. To avoid both of these, you can orient your model a different way, so they don't require supports. Otherwise, minimize the use of supports and add them only where necessary.

---

## What is G-Code?

Before we can print our freshly downloaded or designed 3D file, we first have to put it in a format the printer can understand. We have to convert the .stl, .3mf., .oltp, or .obj file to machine readable geometric code or .gcode. Unlike the other file formats that contain 3D model shape, size, geometry, and color, .gcode provides instructions to a machine controller that tells the motors where to move, how fast to move, and what path to follow. This process is also known as “slicing”.

At a high level .gcode instructs the machine where to move on the X, Y, and Z planes. It also can instruct the machine to adjust other various settings such as fan speed, layer height, travel speed, how much material to extrude, temperature, and how much to retract before moving to the next X, Y, and Z location.

For the structure of the code, there are 3 main sections: Initialization, Printing, and Resetting the Printer.

While it is important to understand the basics of GCode and what it does until you get into more advanced Additive Manufacturing it is not necessary to dive too deep into every nuance of .gcode. Modern slicing software for the most part will handle all you need to prepare the file for printing by generating and managing the creation of .gcode for you. The following sections go over a brief overview of .gcode for those who want to learn a little more about what is happening behind the scenes.

## Syntax

Every G-Code command line follows a certain syntax. Each line corresponds to only one command. Thus, the code can result to be quite long. In the example below, you can see that the first argument must be a ‘G’ or ‘M’ code type followed by a command number that identifies the command. Next, we have the parameters that more accurately detail the command. Each command has its own set of parameters to follow.



<https://all3dp.com/2/3d-printer-g-code-commands-list-tutorial/>

---

## Comments

When reading the code, you'll see semicolons after a command that explains when the code does. Comments can help give readers an explanation of what the code is doing. Here is an example of a command line with a comment:

*G1 X25 Y5; I am a code comment!*

## G-Code Command List

If you are curious about understanding the code the machine uses as instructions or advanced enough to change the GCode, here is a list of general GCode commands to reference. Otherwise, you may leave the GCode alone by default as the slicing software will handle everything for you.

### G0 & G1: Linear Motion

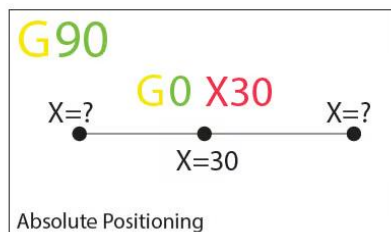
The 'G0' and 'G1' commands both perform linear movements. 'G0' is used for non-extrusion movements such as initial or travel moves, while 'G1' is all the extruding linear motion. For both of these commands, they function the same and have the same parameters that include the desired final positions for the X, Y, and Z-axes, the amount of extrusion during the move, the speed, and the feed rate in the set units. It is noted that most linear moves are performed within one layer and thus the Z coordinate is usually removed from the command line.

**G0** X<pos> Y<pos> Z<pos> F<rate> E<pos>

**G1** X<pos> Y<pos> Z<pos> F<rate> E<pos>

### G90 & G91: Absolute & Relative Positioning

The 'G90' and 'G91' commands tell the machine how to interpret the coordinate movements. 'G90' is the "absolute positioning" and "G91" is for the "relative positioning". Neither of these commands require any parameters and if you set one, it automatically cancels the other.



---

## G28 & G29: Auto-Home & Bed-Leveling

The 'G28' print command is used to perform the process of "homing" the machine. This process is setting the physical limits of all movement axes. This is important for the machine to orient itself and prevent the printhead from moving outside the boundaries of the print bed. The "G29" command starts the automatic bed leveling sequence. This method is usually set by the firmware.

Auto-Home	G 28	[X] [Y] [Z]
Bed Leveling	G 29	

## M104, M109, M140, & M190: Set Temperature

These are all miscellaneous commands that do not involve movement. 'M104' sets the target temperature for the hot end to reach and to keep it there until otherwise told. Parameters for this include the actual temperature value (S) and the which print head to heat (T) if there are multiple extrusions. 'M109' waits for the temperature target to be reached before moving to the next line. 'M140' and 'M190' are very similar and control the bed temperature instead.

HOTEND	M 104	S<temp> I<index>
	M 109	S<temp> R<temp> T<index>
BED	M 140	S<temp> I<index>
	M 190	S<temp> R<temp> T<index>

## M106 & M107: Fan Control

These controls provide fan control. 'M106' turns the fan on and sets its speed. This is particularly useful for the cooling fan. The speed parameter (S) must be between 0 and 255. A 255 value equates to 100% power. The (P) parameters control multiple fans and an index is assigned by the firmware. 'M107' will power off a specific fan, usually the cooling fan and thus only has the (P) parameter.

M 106	S<speed> P<index>
M 107	P<index>

---

## Slicers & ideaMaker

The slicer or slicing software is the primary tool we use when 3D printing files. Slicers are not for designing 3D files; the one main purpose for the slicer is to “slice” the 3D file into horizontal layers by creating .gcode<sup>11</sup> to be sent to the printer. There are many very popular universal slicers like Cura or Orca slicers that work with many brands and types of printers, but for the Maker Center, we will be using ideaMaker.

Universal or non-manufacture specific slicers require the user to set up the slicer for their given printer or printers. Universal slicers are nice especially if you operate many different, brand, size, or type of printers and want to only learn and operate only one set of tools to operate all of them. That being said, many manufacturers produce their own slicers that are pre-setup for their specific machines, such as Prusa Slicer for Prusa printers, or like we have here in the Maker Center: ideaMaker for the Raise3D printers.

For this guide, we will not go deep into the settings of ideaMaker and focus more on the basics to get you up and ready to print your first print. When you get more advanced, want to reference anything, or are just curious to learn, you can find the link to download the ideaMaker manual below.



For reference, you can access and download the IdeaMaker Manual:  
<https://support.raise3d.com/ideaMaker/download-manual-15-524.html>

---

<sup>11</sup> Refer to section: What is GCode?

---

## The Maker Community

One of the most beautiful things about 3D printing is the amazingly supportive community of makers that exist. The maker community is full of extremely talented artists, sculptors, modelers, designers, computer scientists, entrepreneurs, hobbyists, experts, newcomers, students, educators and everyone in-between.

As you dig deeper into the 3D printing maker world you will discover a very supportive, sharing based community of people who are always willing to help and share their ideas, files, and experiences. Some have created businesses around 3D printing and designing files for print, others just share their work in what is known as creative commons<sup>12</sup>. This sharing mentality is what has really opened the doors to everyone to have the ability to enjoy 3D printing without ever having to learn to create a 3D file; just download, slice, and print.

There are many file repositories out there for the purposes of this training we will focus on the most common in the maker community, you will find shortcuts to these sites on our Maker Center computers. [Printables.com](http://Printables.com), [Thingiverse.com](http://Thingiverse.com), and [Cults3d.com](http://Cults3d.com).

After finding a model that you are interested in, you must review the license rights to ensure it comes with a creative commons license. The license rights with files posted online and offered for free is ultimately based on an honest system. On any of the websites on the computers in the lab, you can click on a model you like and look for the license requirements. You can reference the following site, to get an overview of the different license options.



<https://creativecommons.org/share-your-work/ccllicenses/>

Few makers have enough resources or power to track down and bring litigation against someone selling their files. That being said, this is something that should be honored and supported 100%. There is a zero-tolerance policy in the Maker Center for the misuse of files or violations of the creative commons license.

The first time you put your blood, sweat, and tears into the design of a file you will understand and highly respect what it takes to create magic through 3D design and do all you can to protect that work, trust me on this one. The best way to support the 3D maker community and continue to drive the hobby and lifestyle forward is to respect, support, and defend the creative commons license when using and sharing files.

---

<sup>12</sup> Gives everyone a standardized way to grant the public permission to use their creative work under copyright law.

---

## **Maker Center Rules & Regulations**



# CLV Maker Center

## GENERAL RULES & REGULATIONS



### ASK

If in doubt, please ask the technicians for help



### BE RESPONSIBLE

All minors under 18 must be supervised by an adult who is at least 21 years old at all times.



### NO EATING/DRINKING

Do not eat or drink. It is only permitted in the lobby



### TIDY UP

Make sure to clean your workspace after use.



### REPORT THE GOOD STUFF

The larger the community the more knowledge we can gain. If you had fun, made friends, learned something new, share with your friends!



### BE MINDFUL

Be respectful to your neighbors by talking at an appropriate volume, no music, or phone calls.



### DOORS

Please keep front doors closed at all times and do not let people in.



### BE UNDERSTANDING

We cannot guarantee item quality, stability, or a specific print time.

### CITY VALUES

**Kind:** Caring, empathetic, courteous, service oriented, patient

**Committed:** All in, dedicated to excellence, determined to succeed

**Smart:** Not book smart only! Showing good judgment and decisions and actions, doing the right thing at the right time, having situational awareness, maintaining healthy emotions



#### DISCLAIMER:

COLOR OF FILAMENT IS MANAGED AT STAFF DISCRETION AND SPECIFIC COLORS ARE NOT GUARANTEED.



# CLV Maker Center DESIGN COMPUTER LAB RULES



## CREATIVE COMMONS

All projects created in the maker center must respect the individual property of others and follow all U.S. copyright laws.



## SHARING IS CARING

Be conscientious of your build materials. The material cost is free for everyone and we want everyone to have a chance to create something.



## NO WEB SURFING

No web surfing unless relevant to the development of your project.



## BE PREPARED

Bring your own headphones or USB thumb drive.



## NO .GCODE

Do not bring your own G-code. Accepted file formats are .stl, .3mf, .obj, .oltp, and image files.



## APPROPRIATE MODELS

All models must be family appropriate and have no malicious intent.



## TIDY WORKSPACE

Do not move the computer towers or monitors.



## RESTART COMPUTERS

When done, please restart the computers for the next person to use and to wipe any personal information that may have been stored.



## FILE NAMES

Please rename your print files to:  
**lastName\_firstName\_month-day**  
Ex. Jones\_Alice\_03-02



## APPROVED PRINTS

When done verifying your print, please ask a technician to approve first and help you print.

# CLV Maker Center

## TRAINING/PROTOTYPE/POST PROCESSING ROOM RULES



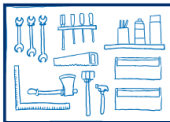
### TOOLS

Tools must be in the training room at all times and be returned after usage.



### SAFETY FIRST

Wear safety goggles, masks, and/or gloves during post-processing.



### CLEAN UP

Make sure to leave workbench or cut mats clean for the next user.



### BE ALERT

Power down tools when not in use. During use, keep an eye on it at all times.



### FIRST AID

In case of emergency, please alert one of the technicians to help guide you to the necessary aid.



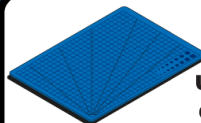
### NO MINORS

No one under 18 can operate any of the tools themselves.



### TOOL HELP

A technician is able to help explain the usage of the tools but any finishing steps is responsibility of the maker.

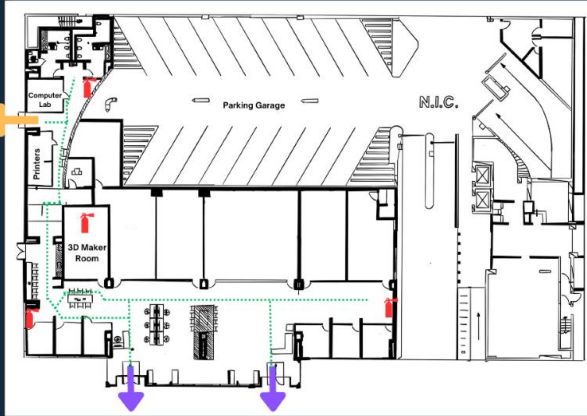


### USE CUT MATS

Use cut mats when conducting any postprocessing on tables.



# FIRE EVACUATION PLAN



## FOR YOUR SAFETY

**DO NOT** stop to collect personal belongings.  
**DO NOT** re-enter the building unless told it is safe to do so.

## IN CASE OF FIRE

### 1. Report



Alert a staff member

### 2. Extinguish



Use extinguisher if it is safe to do so

### 3. EVACUATE



Follow exit routes to move away from the building

### 4. Go to Safe Refuge Area (SRA)



Exit to Lewis, go through alley to SRA



Exit on 4th, turn left on Bridger toward SRA



---

## **Steps for Printing a Model**

---

## Section 1. Create or Find a Design

There are two ways of retrieving a model. You may import your own model from Blender or TinkerCAD, or find a model on one of the provided websites on the lab computer.

### 1.2 Build Your Own Model

You can create your own model using Blender or TinkerCAD.

### 1.3 Search for a Model

You can find a model on [Printables.com](http://Printables.com), [Tinkerverse.com](http://Tinkerverse.com), or [Cults3D.com](http://Cults3D.com). Keep in mind that most of the models on Cults3D are pay to own and it is at your own discretion if you want to pay for a model. All of websites are added on the desktops for your convenience.



#### a. Selecting a Model

When selecting a model, it is important to check the Creative Commons licensing (p.31).

#### b. Downloading the Model

Referring back to the File Formats section, you want to make sure the format is accepted. You can find the accepted file formats in a previous section (p.20). Make sure to remember where you downloaded the file. By default, it will download to your 'Downloads' folder.

---

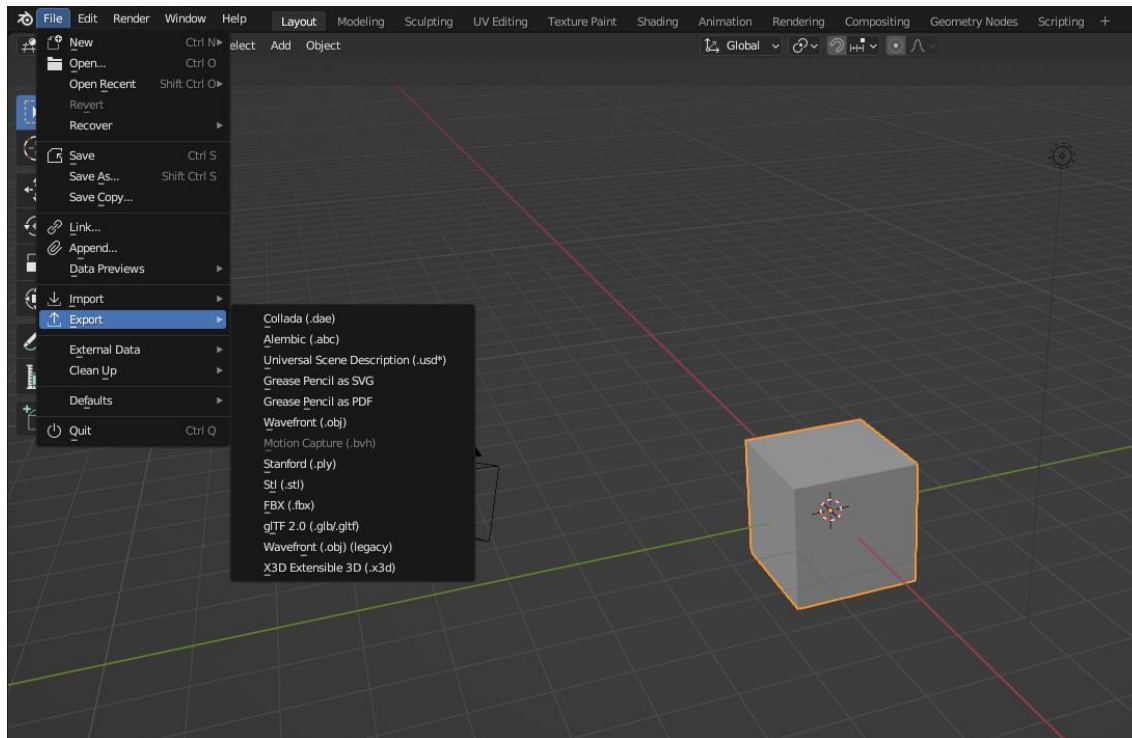
## Section 2. Loading a File into ideaMaker

Our printers use the ideaMaker software to load a model for printing.

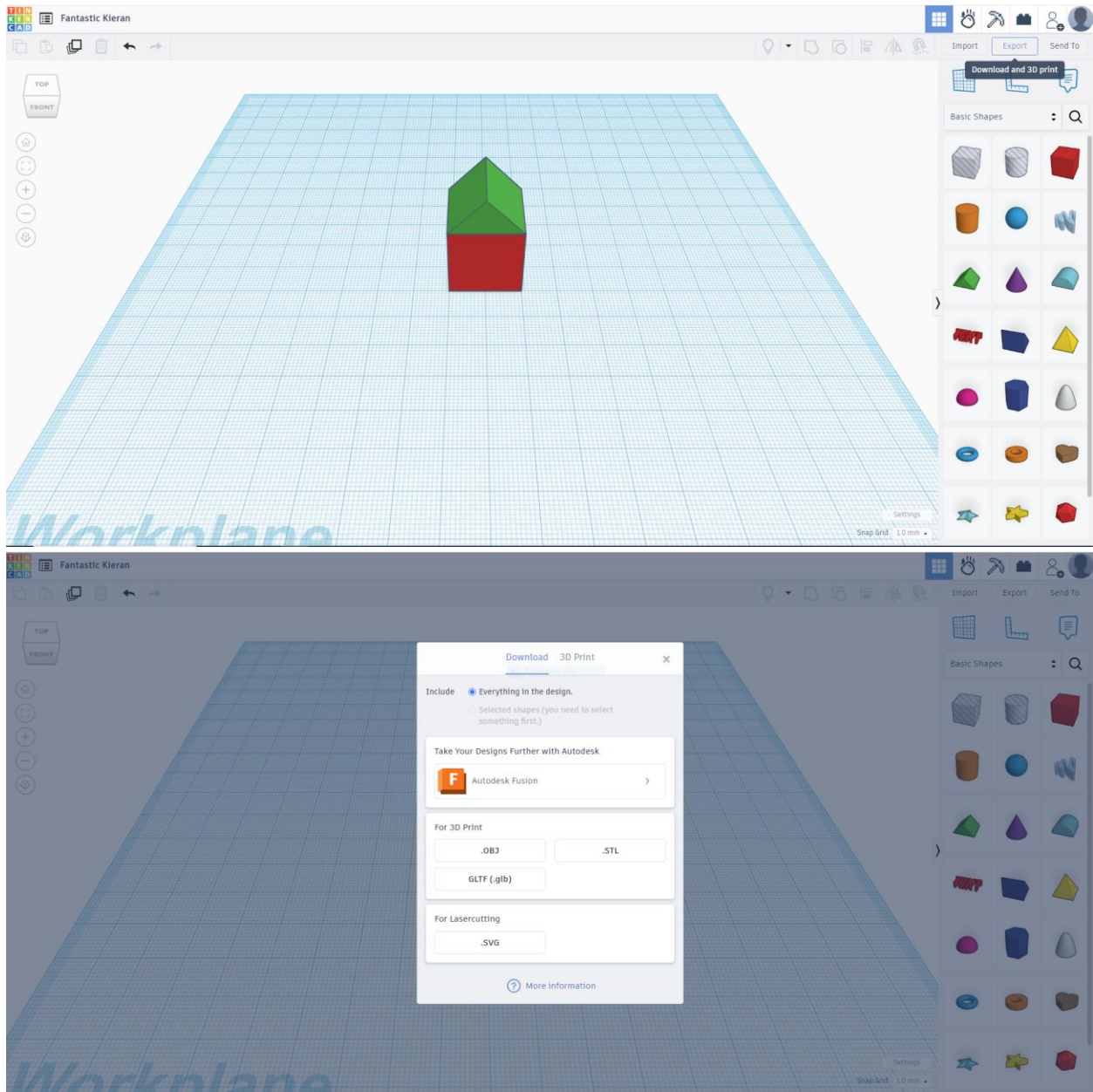
### Section 2.1 - Exporting a File (If you created your own)

If you created your own model in Blender or TinkerCAD, the file must be exported to a different file format in order for ideaMaker to accept it. Accepted file formats are stereolithography (.stl), wavefront OBJ(.obj), 3D manufacturing file (.3mf), OLT printing file(.oltp) and any image file.

To export from Blender simply click File > Export > Stl (.stl)



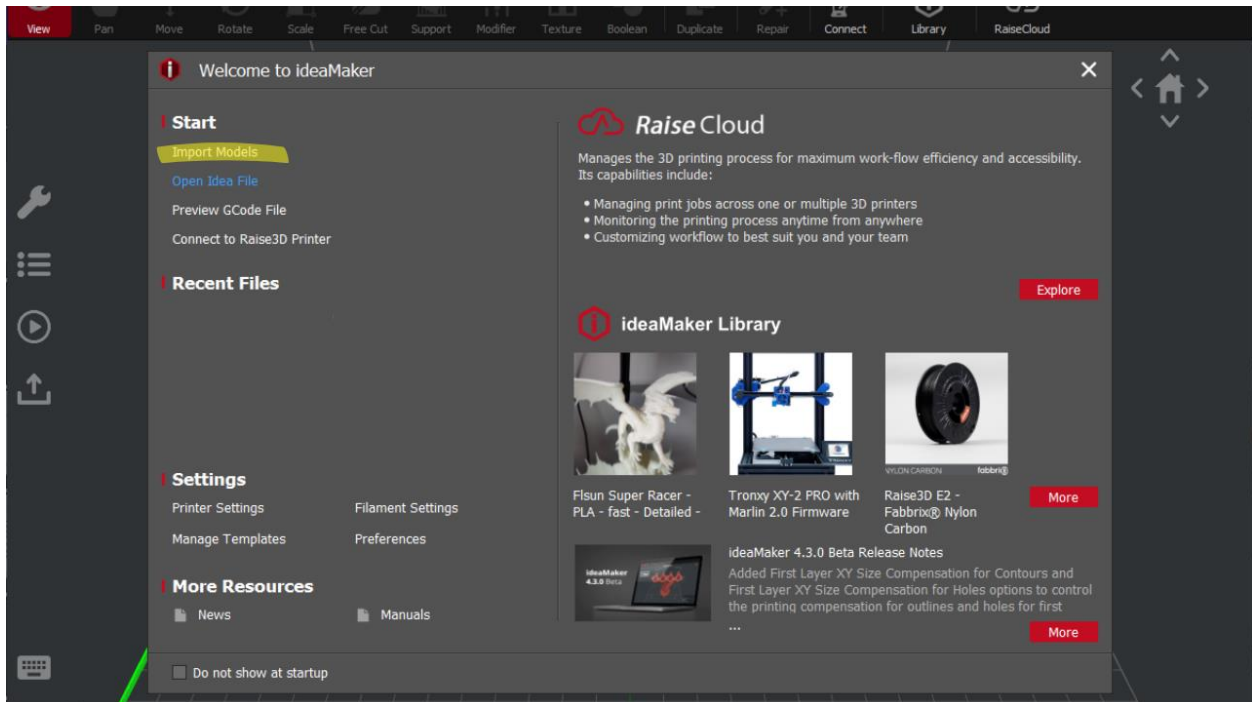
To export from TinkerCAD simply click Export > and choose your preferred file format under 3D print.





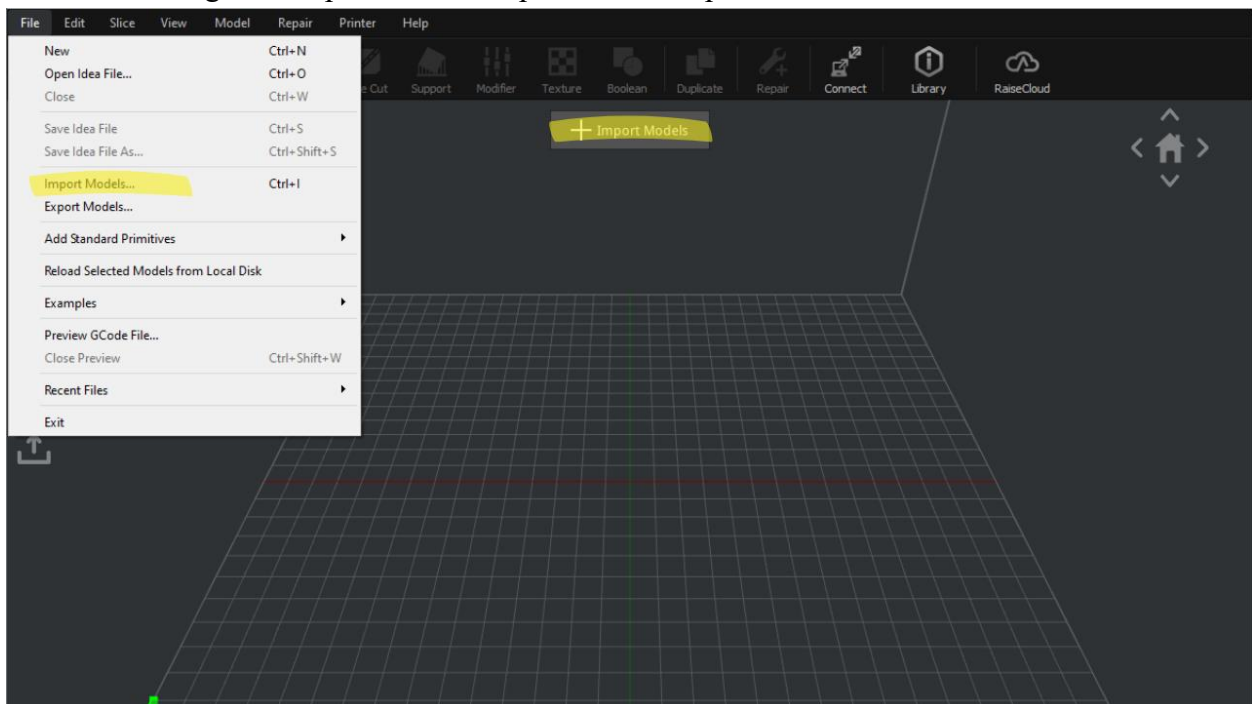
## Section 2.2 Importing a File into ideaMaker

After opening ideaMaker a welcome window will open, from this window select “Import Model”



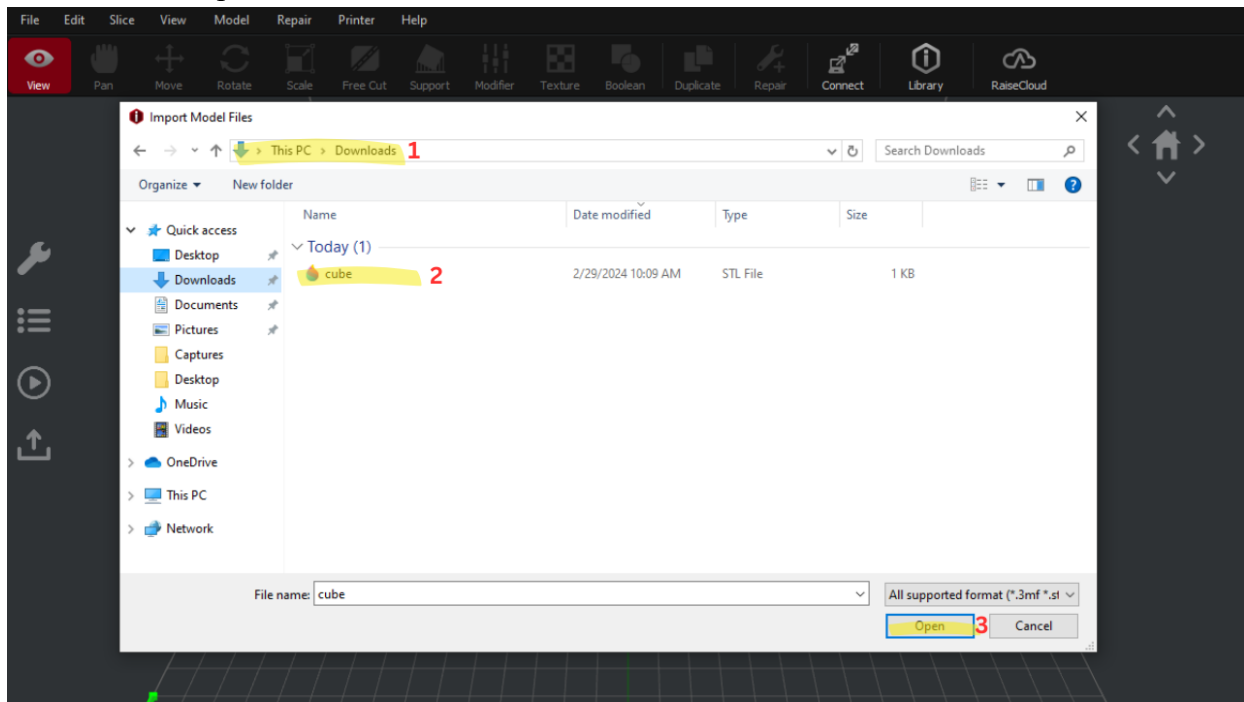
Open IdeaMaker and import your model by:

- Select the “File” dropdown on the top left of the screen and select “Import Model”.
- Selecting the “Import Models” option on the top center of the screen.

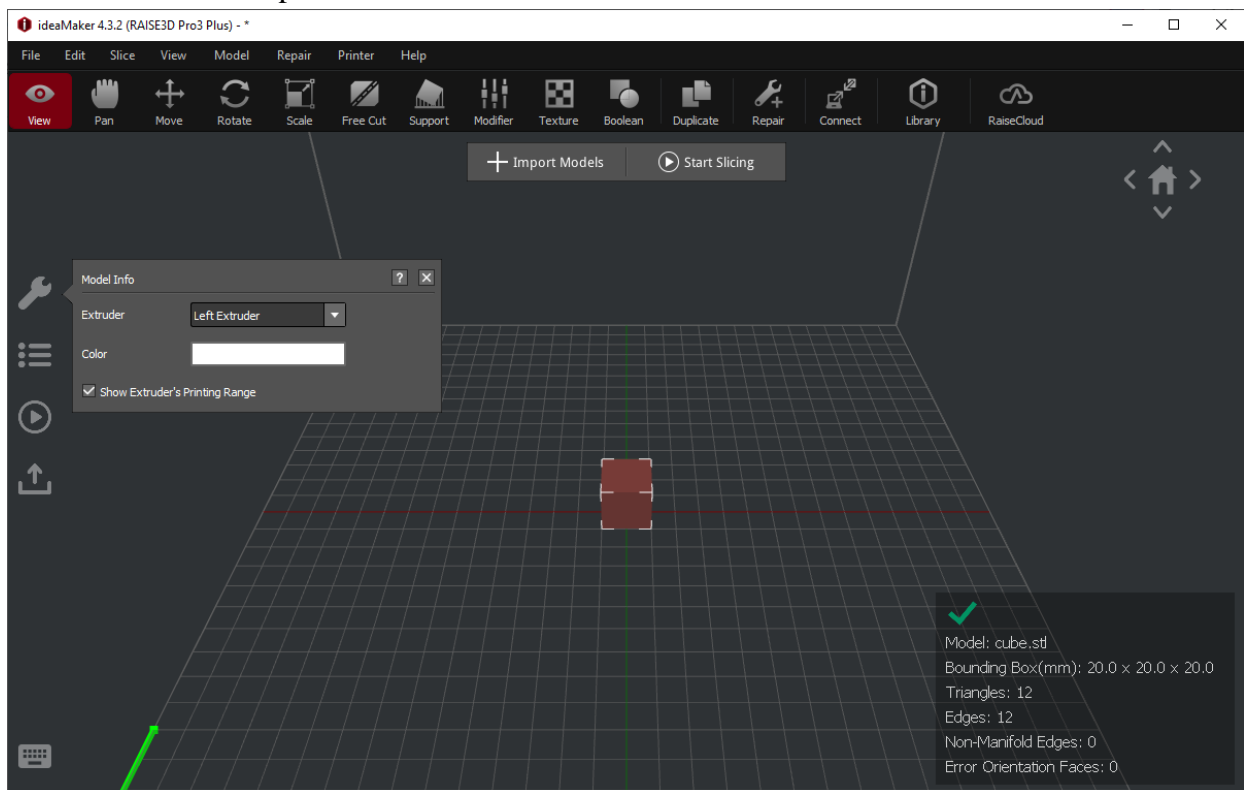


Locate the file that you want to print and load it in.

1. Open File Explorer and navigate to the location of your model.
2. Click on your model.
3. Click “Open”.



The model is now imported into ideaMaker.



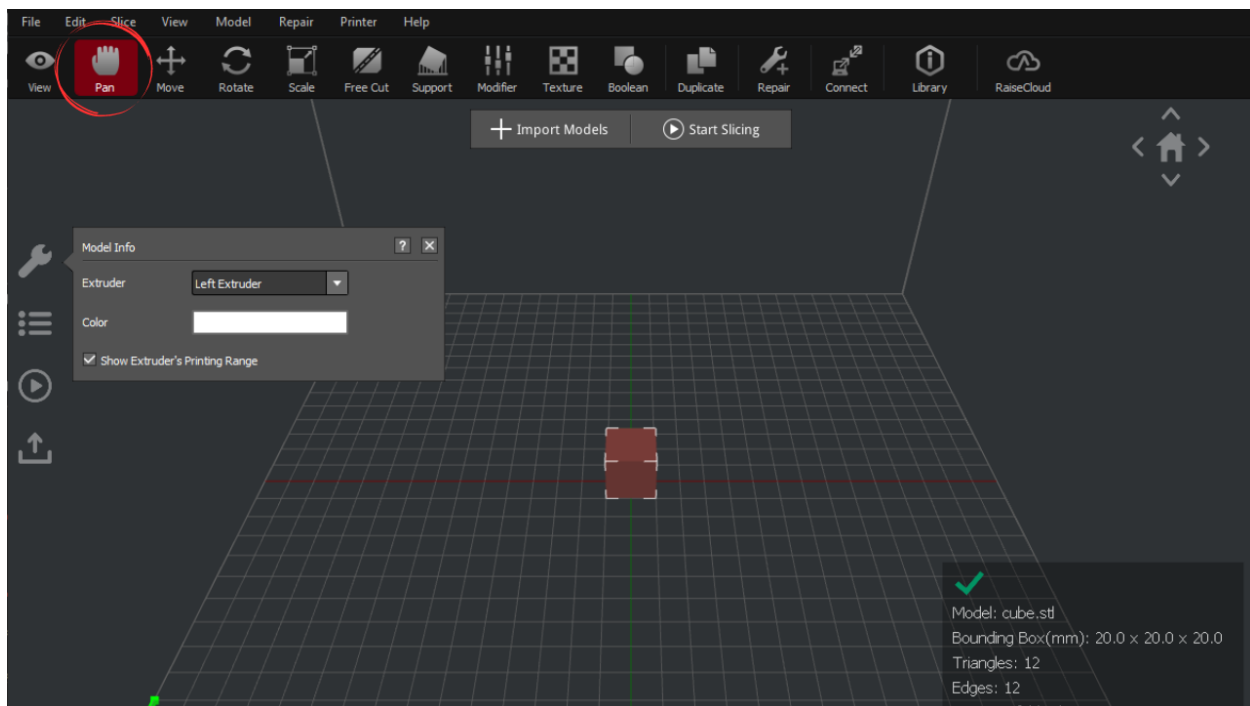
## Section 3. Understanding IdeaMaker

IdeaMaker is where you will view your model in the printer size, change any parameters before printing it, and slice your model. The following will go over different tools available for you to use.

### Section 3.1 Viewing/Panning the Model

Different ways to change the Perspective:

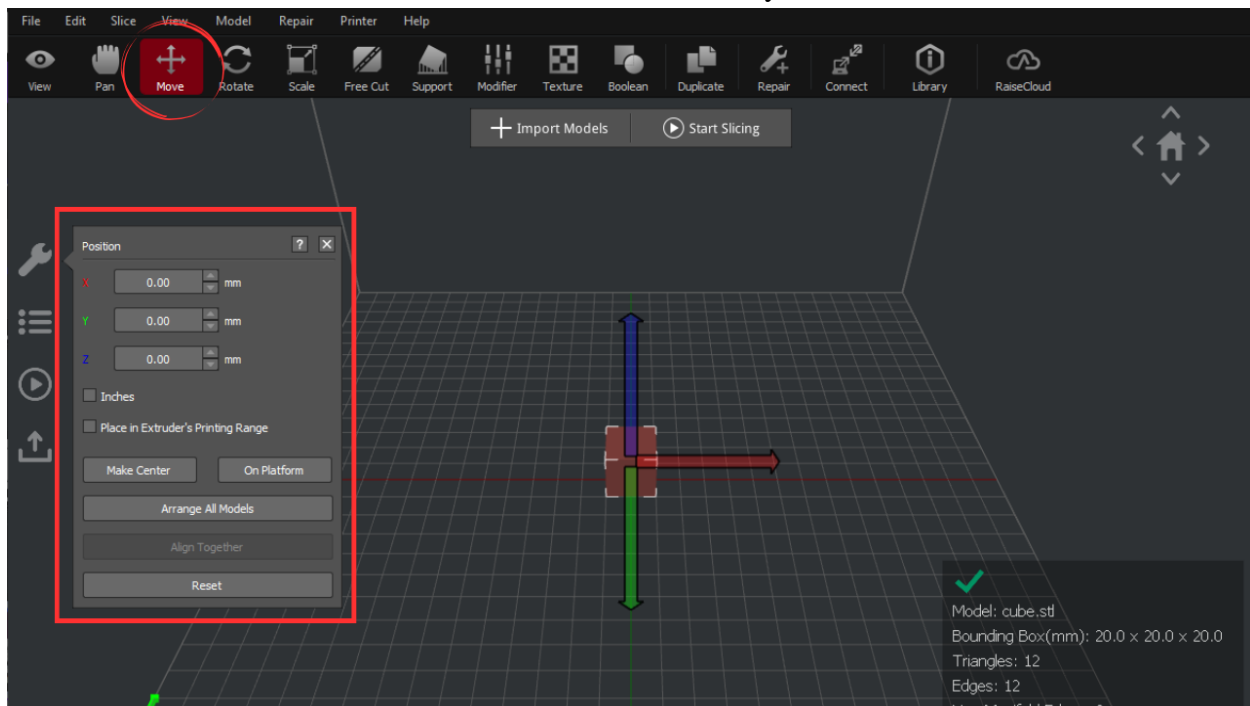
- By right clicking and dragging the mouse in the desired direction, you can rotate your perspective.
- By left clicking and dragging the mouse in the desired direction, you can move your build plate.
- By using the scroll wheel, you can zoom in and out.
- On the top right-hand side of the screen, there is a home icon with arrows pointing in every direction. Clicking on any of these arrows will change the perspective in that direction.



## Section 3.2 Moving the Model

Select “Move” from the top panel. There are two ways to move the model:

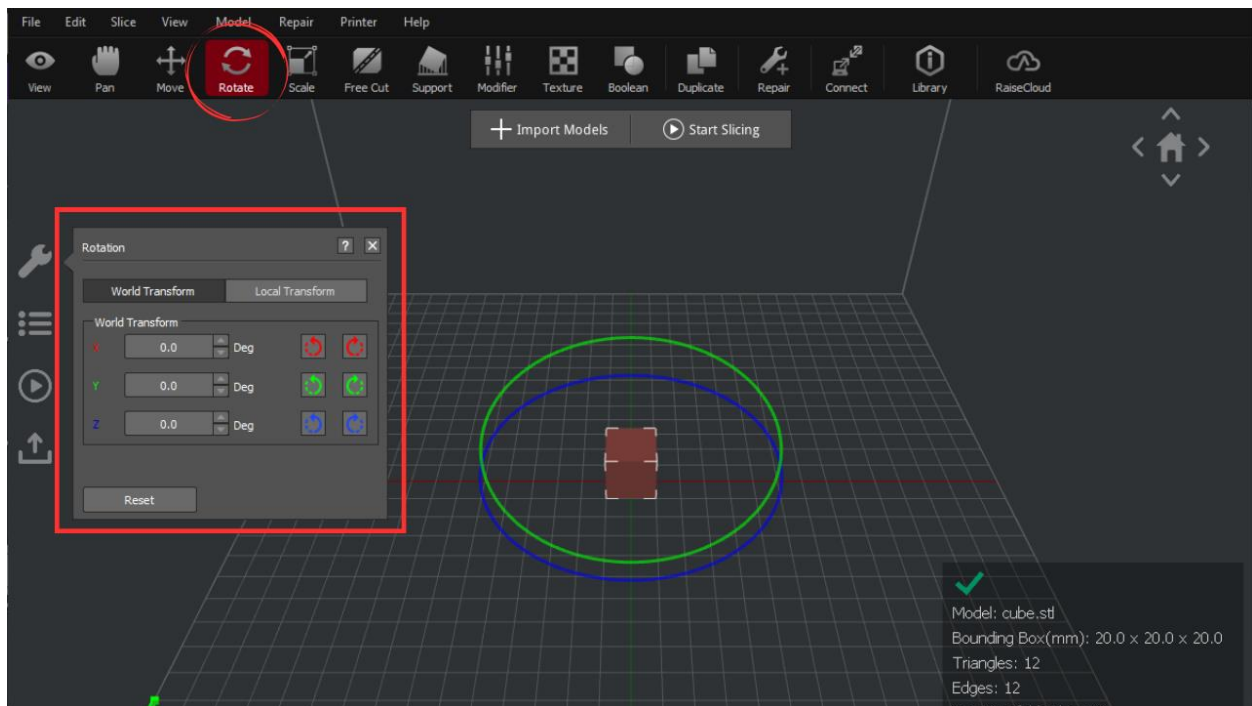
1. Click on the model and drag your mouse in the direction you want to move it.
  - Click on the red arrow to move the model only along the X axis.
  - Click on the green arrow to move the model only along the Y axis.
  - Click on the blue arrow to move the model only along the Z axis.
2. On the left-hand panel, click the wrench logo to adjust the position. Simply enter the desired coordinates and the model will automatically be relocated.



## Section 3.3 Rotate the model

Select “Rotate” from the top panel. Once the model is selected, there are two ways to rotate it:

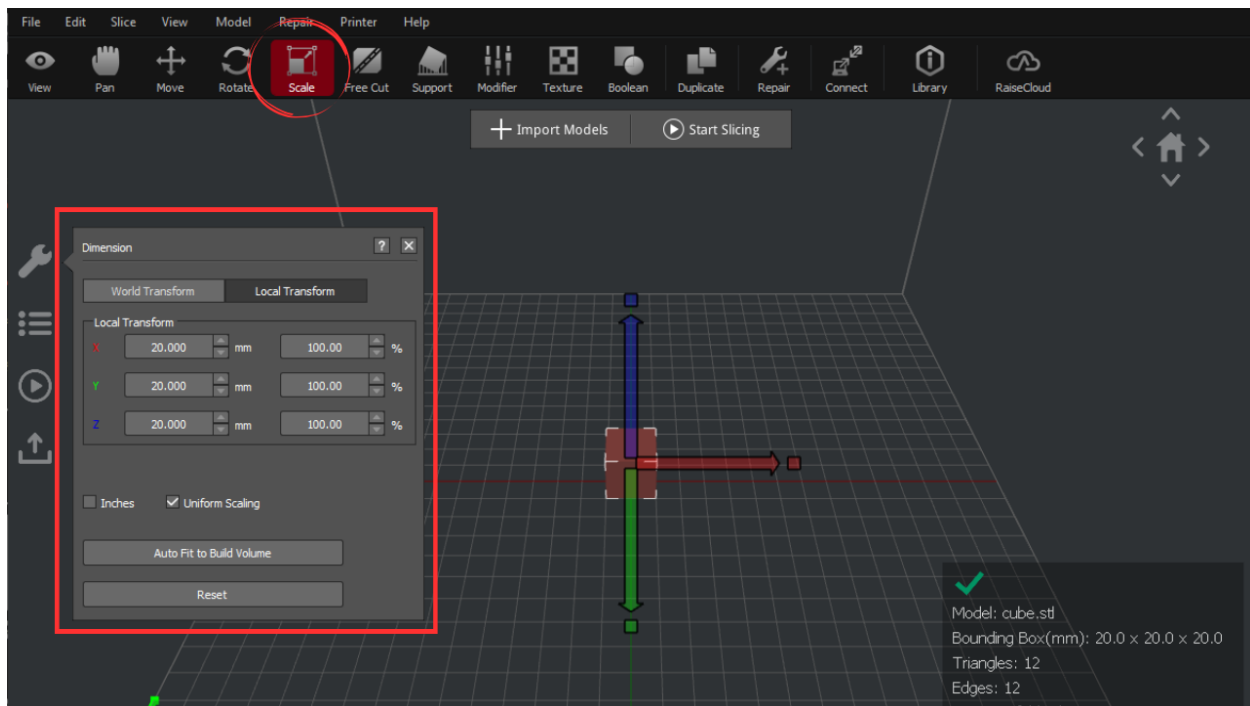
1. Click and hold the desired plane in which to rotate the model. As the mouse is dragged, the model will rotate along the selected plane.
  - Red is the X plane
  - Green is the Y plane
  - Blue is the Z plane
2. On the left-hand panel, click the wrench icon to adjust the angles. Simply enter the desired angle and the model will automatically rotate.



## Section 3.4 Scaling the Model

Select “Scale” from the top panel. There are a few options available:

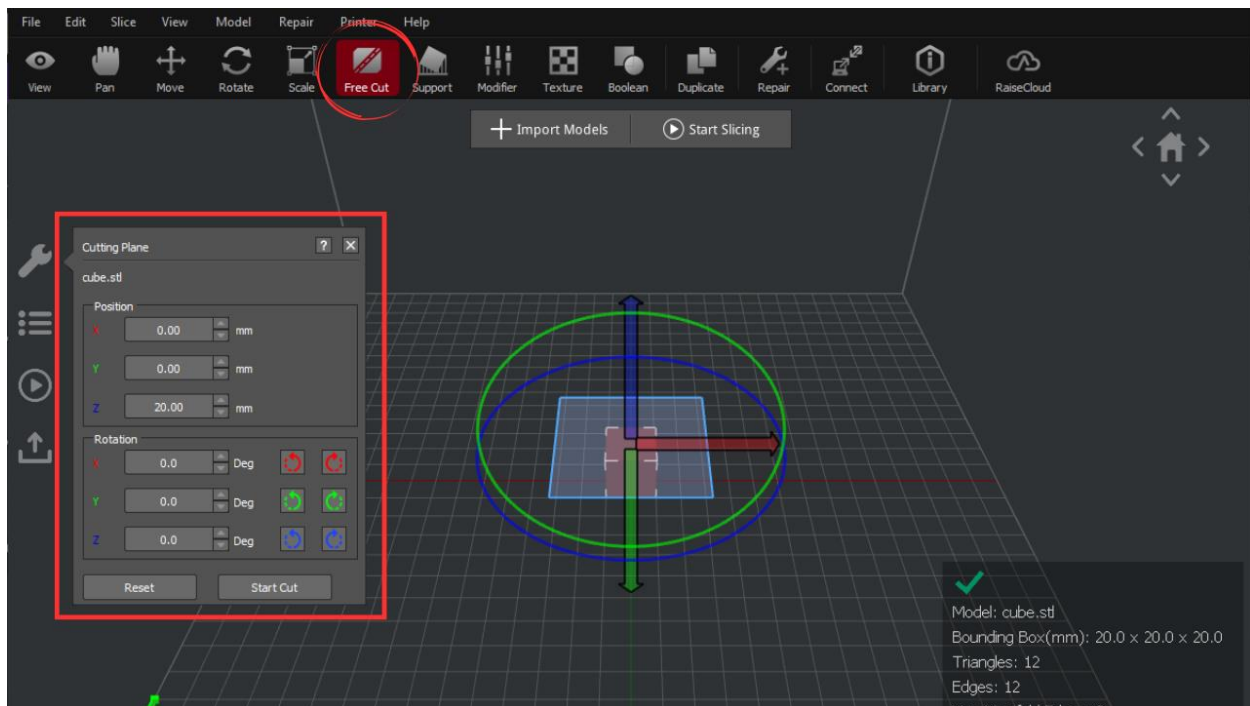
1. Uniform Scaling (default): Click on the model, select any plane and drag towards the model to decrease the size or away from the model to increase the size.
2. Non-uniform scaling: Click on the model and open the wrench logo on the left-hand panel. Deselect the “Uniform Scaling” checkbox. The model can now be scaled along one plane by moving the mouse towards the model to scale down and away to scale up.
  - Use the red arrow to scale only along the X axis.
  - Use the green arrow to scale only along the Y axis.
  - Use the blue arrow to scale only along the Z axis.
3. On the left-hand panel, click the wrench logo to adjust the model size. Simply enter the desired dimensions and the model will automatically size.



## Section 3.5 Free Cutting the Model

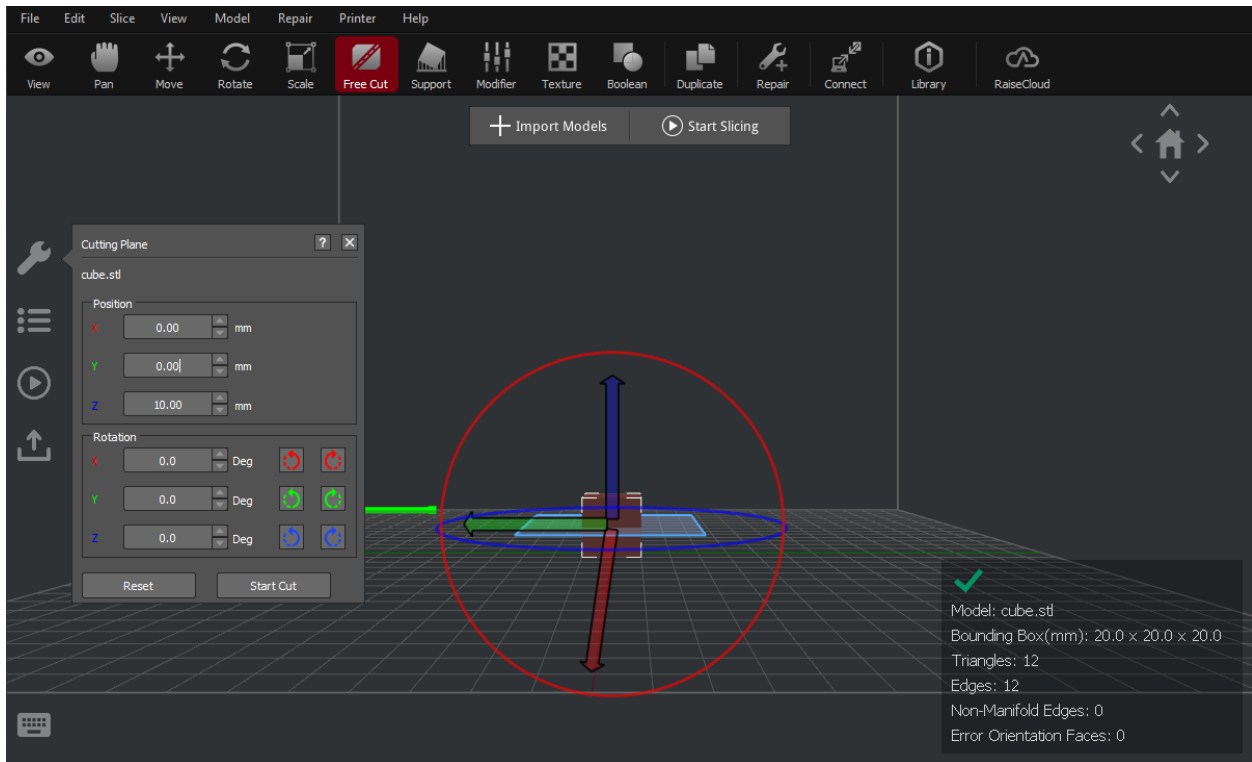
Select “Free Cut” from the top panel. Once the model is selected, there are two ways to cut it:

1. Click and hold the desired plane in which to cut the model. As the mouse is dragged, the blue plane will move along the selected axis. After moving the blue plane to the desired location, select “Start cut” on the wrench logo.
2. In the wrench logo, input the X, Y and Z coordinates of the blue plane as well as any desired rotation to apply to the plane. Once the blue plane is how you want it, select “Start cut”.

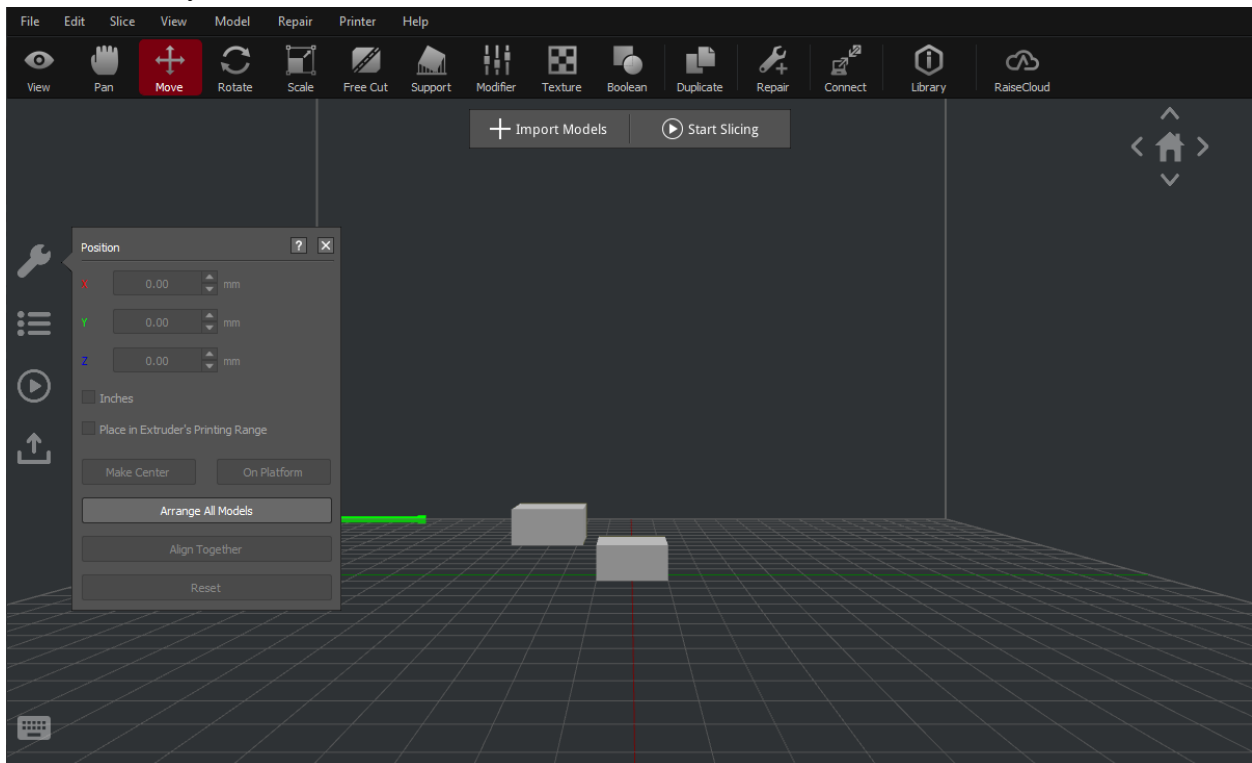


## Use of Free cut tool (dual color printing):

Free cut the model:

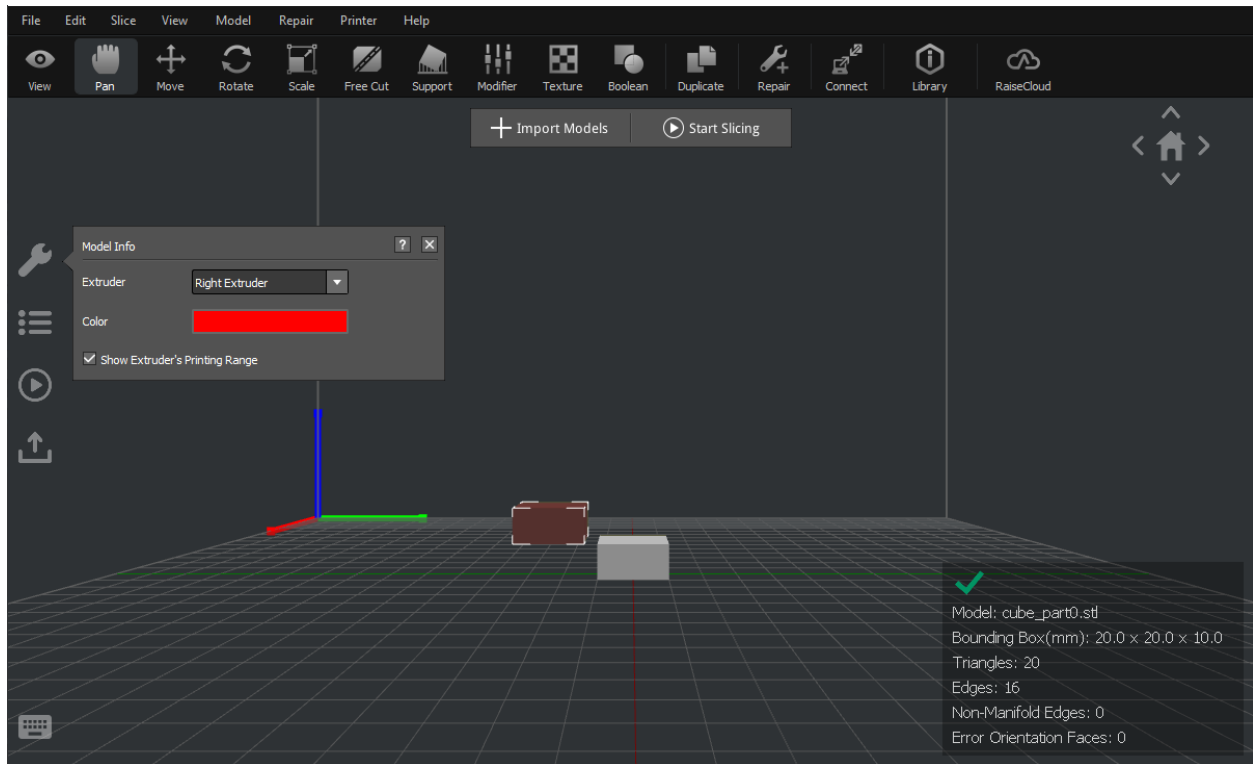


After free cut you should have two models.



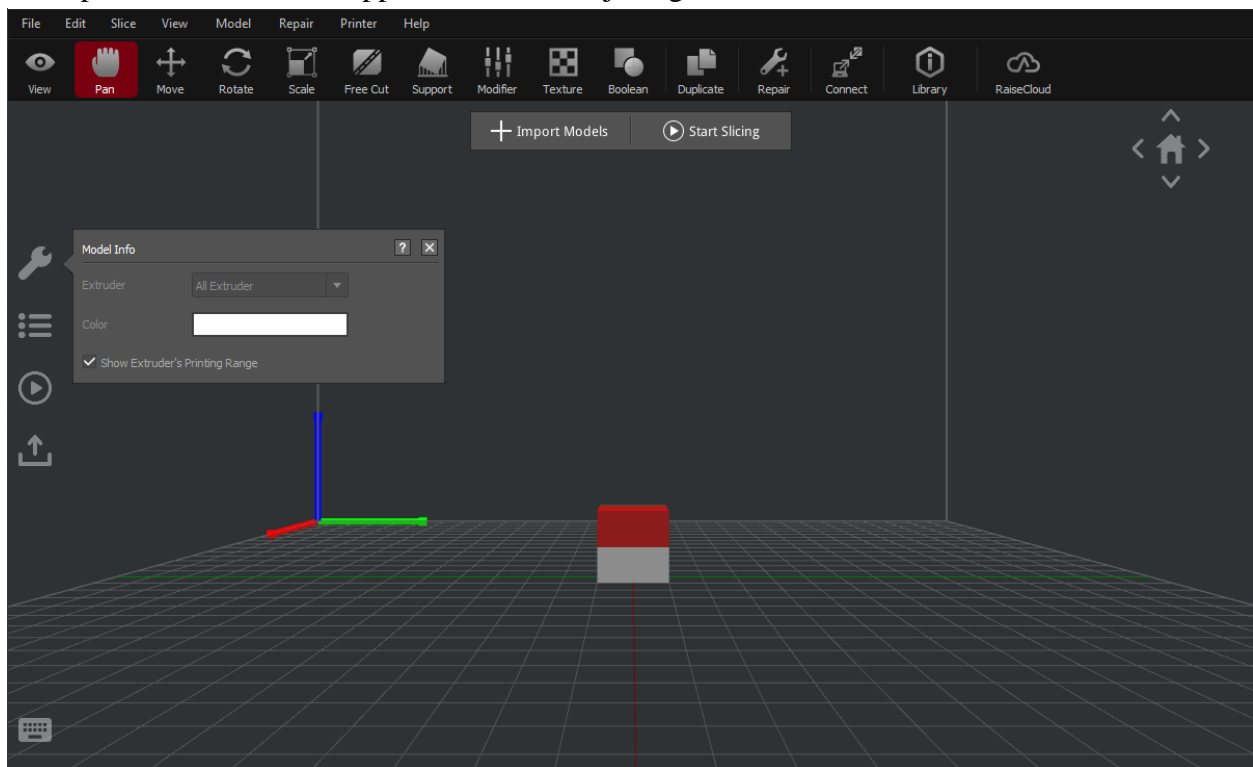


Select whichever model you would like to change color of and change from free cut to pan mode. In the wrench you can change the extruder to use:



Now the desired half of your model will be a different color.

It is important to align the models back together so they can print properly. After slicing these two models together, it is normal to get a warning saying that one of the models is off of the build platform and needs support. In this case, just ignore and continue to slice.



---

## Sectional 3.6 Supports

If the model has overhang with angle less than 45 degrees, the print does not need support. If the overhang angle is greater than 45 degrees, however, support is required to avoid print failure. This is discussed more on page 26. Most of the time, ideaMaker will give you a warning if a model needs supports before printing.

To add support, select the model and choose support from the features. If the wrench is not selected, select it to open settings.

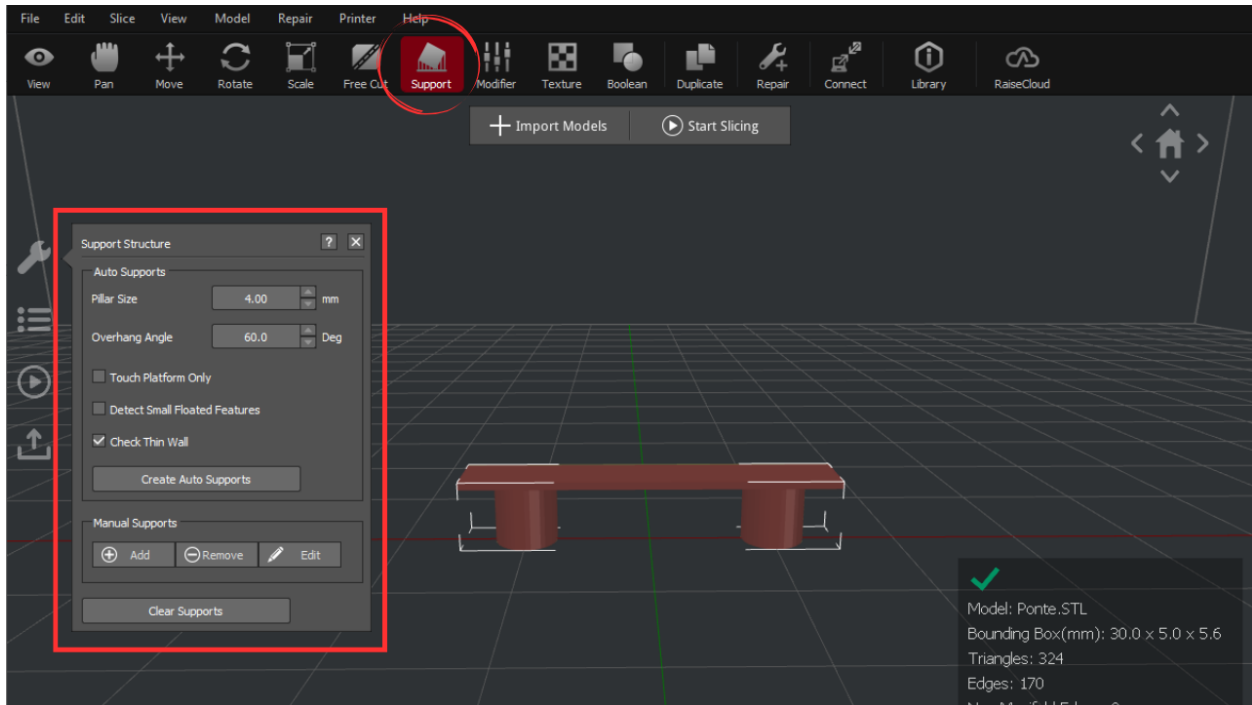
The first method of adding is by allowing ideaMaker to auto generate it by changing these settings and selecting “Create Auto Supports”:

- **Pillar size:** The desired size of pillar supporting the model.
- **Overhang Angle:** ideaMaker will add support to all parts of the model that have an overhang angle of this chosen degree.
- **Touch Platform Only:** ideaMaker will generate supports which will only touch the build platform
- **Detect Small Floated Features:** ideaMaker will automatically detect and preserve smaller details in the model that are not connected to the build platform and produce appropriate support structure.
- **Check Thin Wall:** ideaMaker will generate support for points where the model has thin walls
- 

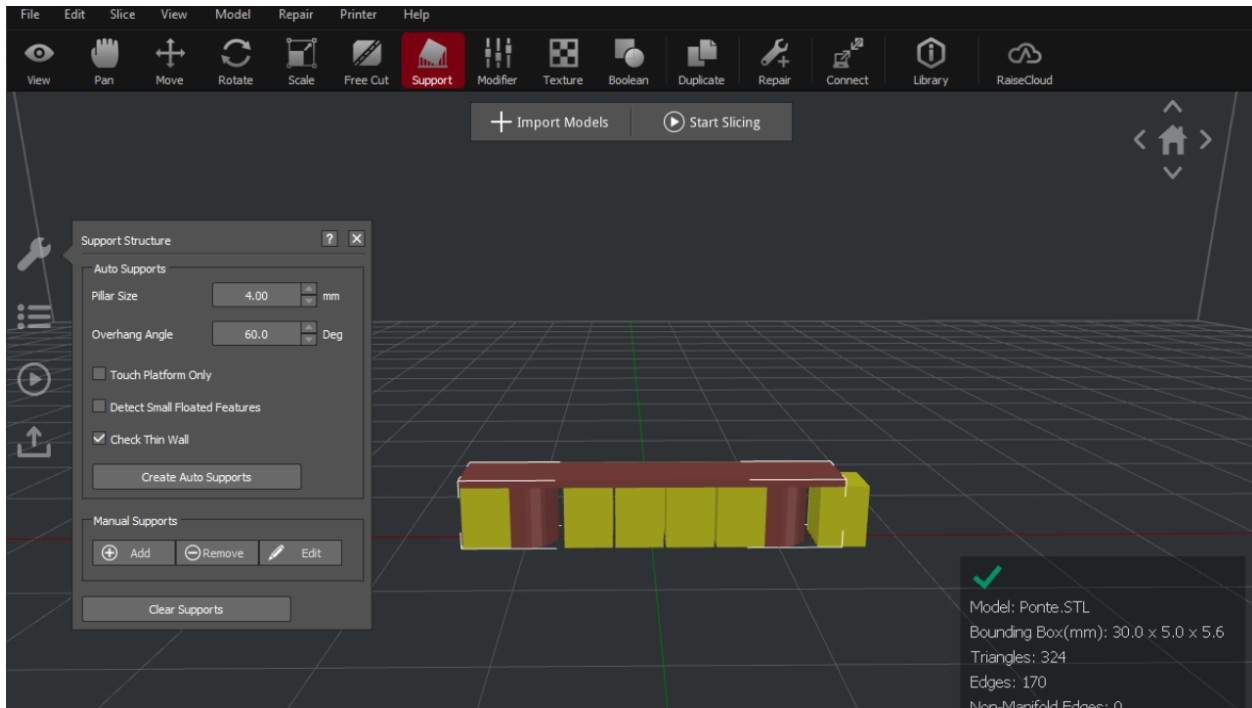
The second method of adding supports is by doing it manually:

- **Add:** will add supports
- **Remove:** will remove supports
- **Edit:** by selecting a pillar and choosing edit, you can edit its Z position as well as its height.

Before adding support:



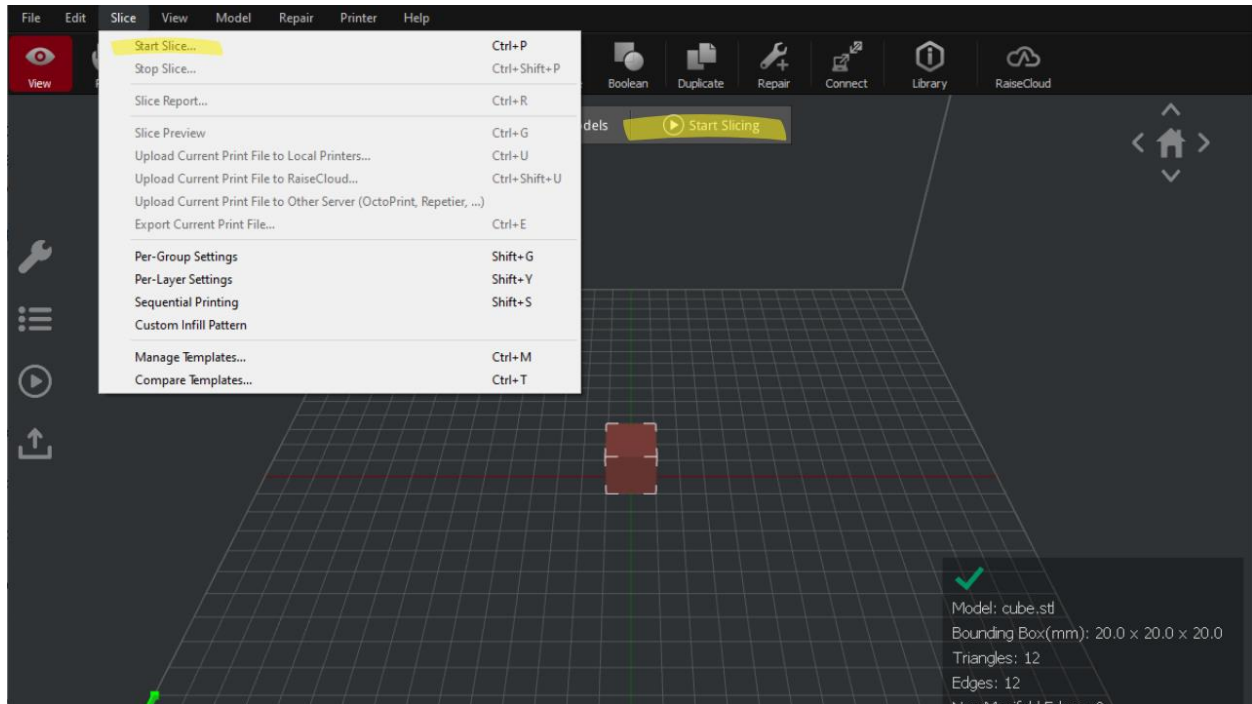
After adding support:



After your print is complete, you can remove the supports during post processing via the provided tools.

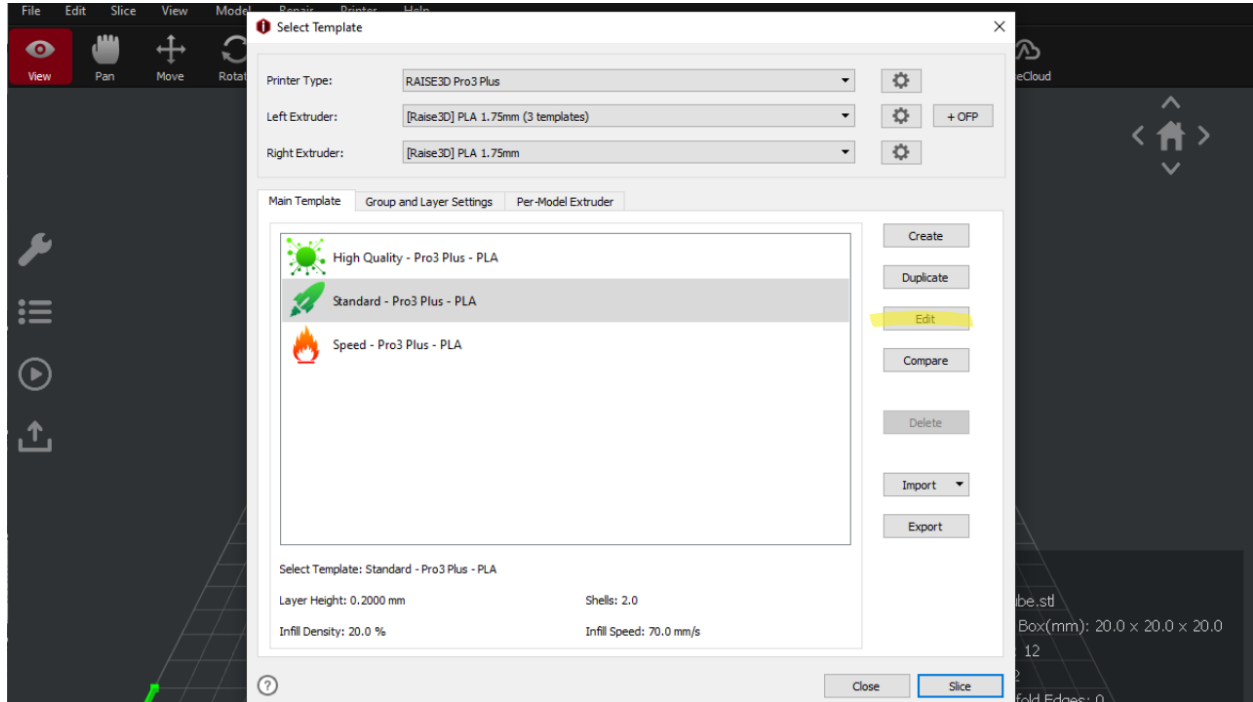
## Section 4. Slicing: Preparing your file for Printing

Before sending your model to the printer we need to convert it to instructions for the printer to know how and what to print. These instructions are called GCode (pg.27).

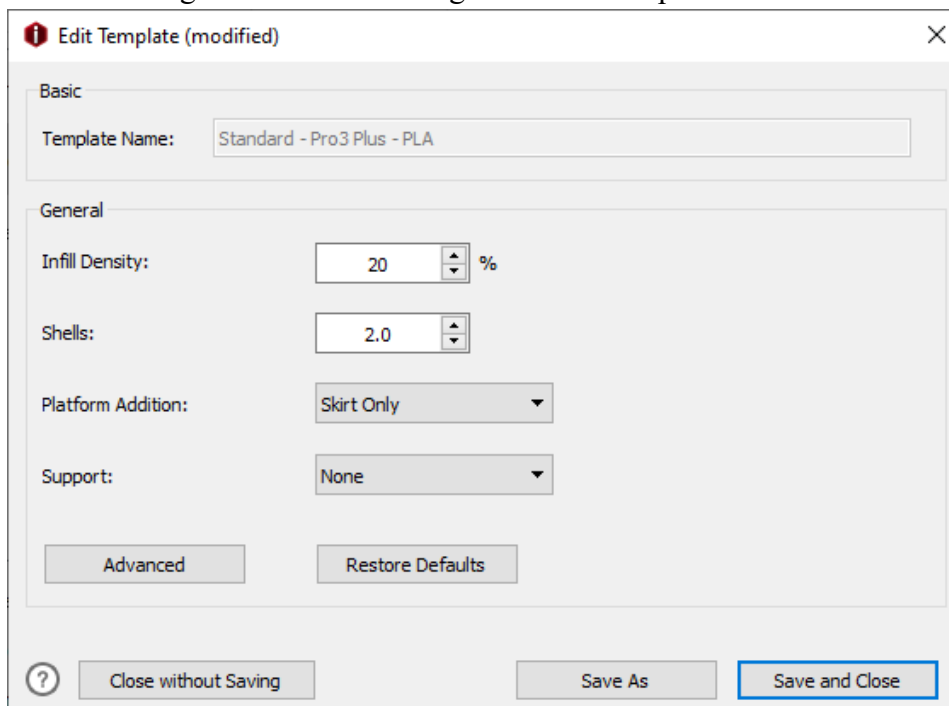


## Section 4.2 Changing Print Parameters Using Simple Mode

After selecting “Start Slicing”, the following window will appear. To edit the settings of your print select edit:

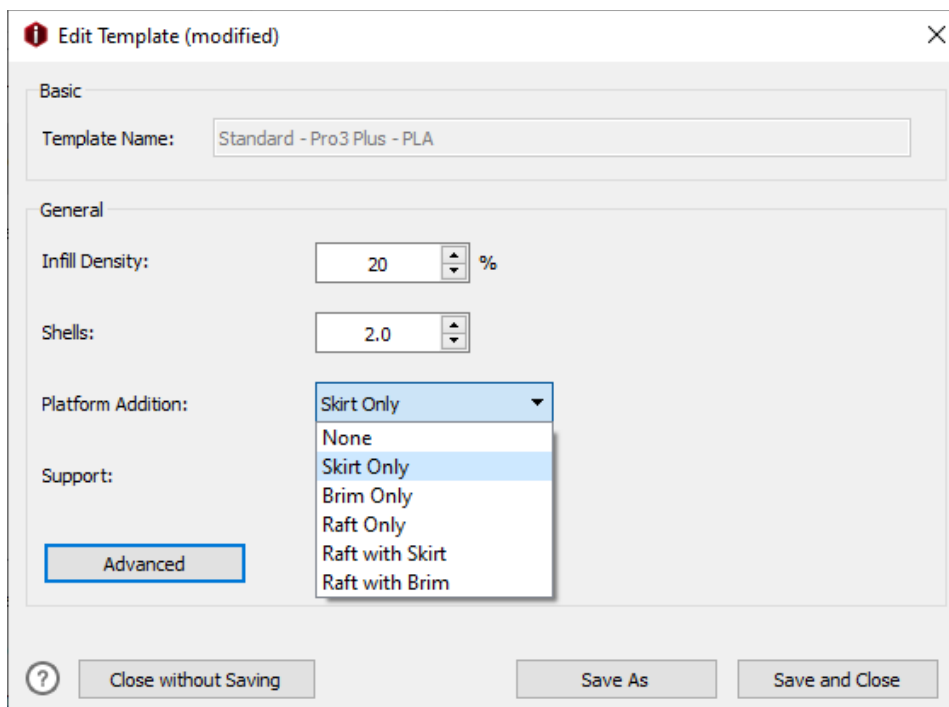


After selecting “Edit” the following window will open:

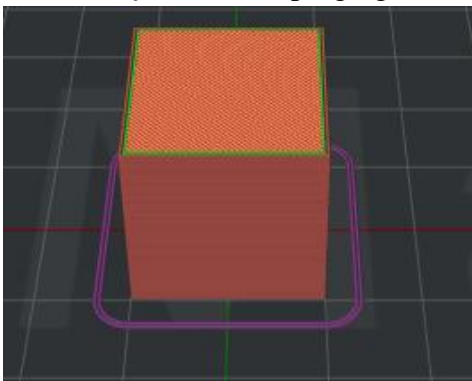


From this window the settings you can change are the infill density, the number of shells, platform editions, and support.

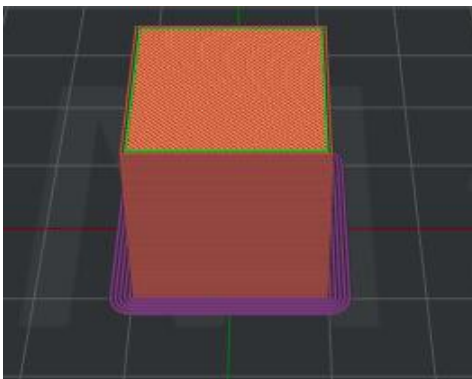
1. **Infill density:** the density of your infill (page number explaining infill)
2. **Shells:** how many shells your model should have (page number)
3. **Platform Addition:**



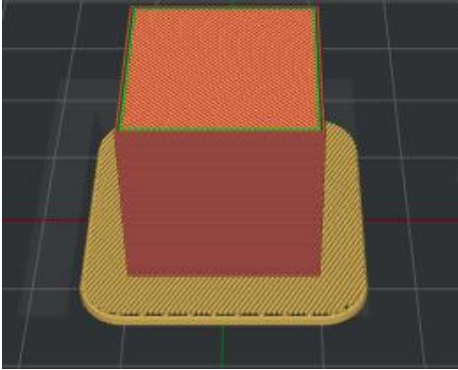
- **None:** there will be no additional platforms
- **Skirt Only:** Good for purging the nozzle before printing



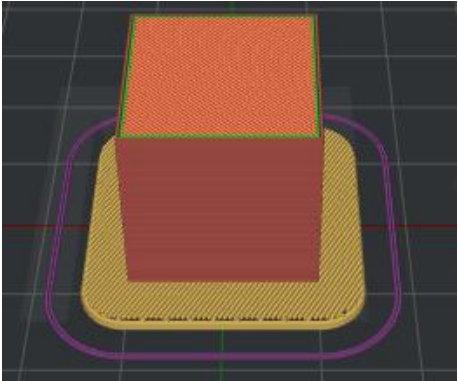
- **Brim only:** Good for increasing your model's adhesion to the bed



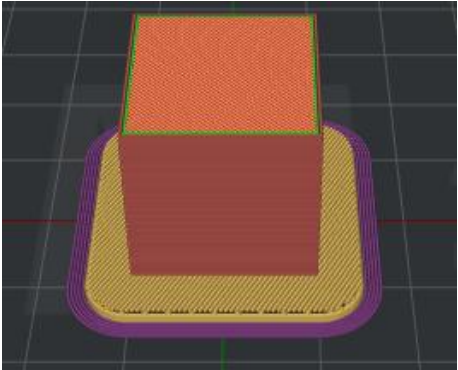
- 
- **Raft:** Good for clean first layer of model



- **Raft with Skirt:** Good for purging the nozzle and creating a clean first layer of the model

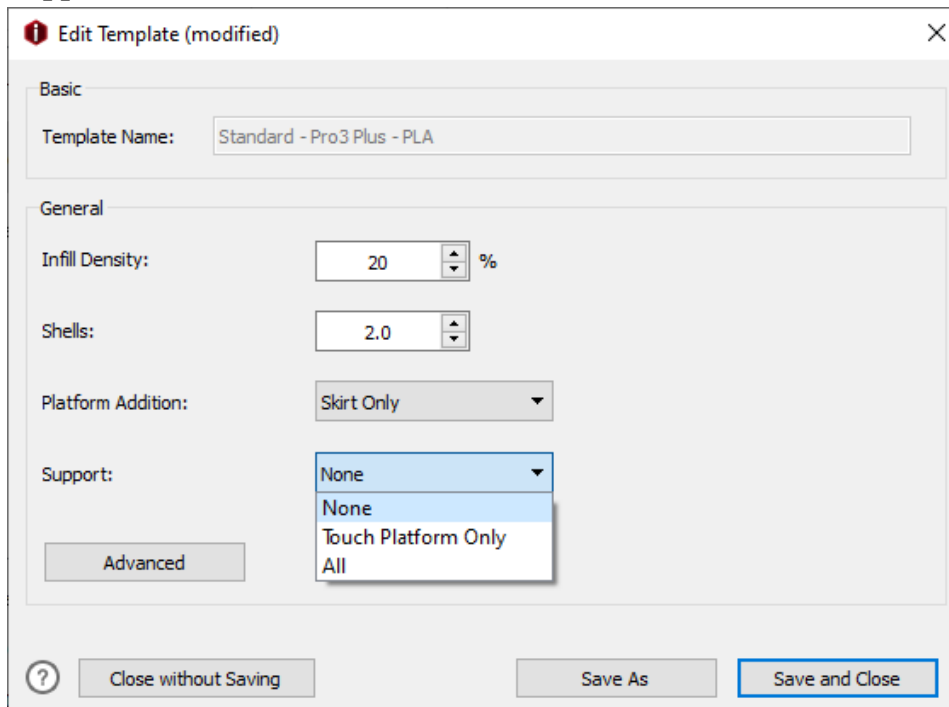


- **Raft with Brim:** Good for adhesion and clean first layer of the model





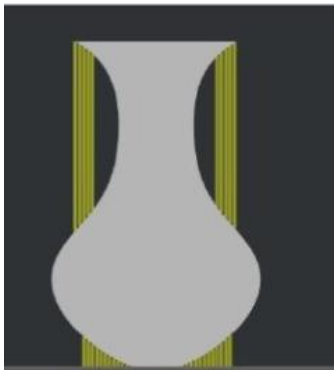
## Support:



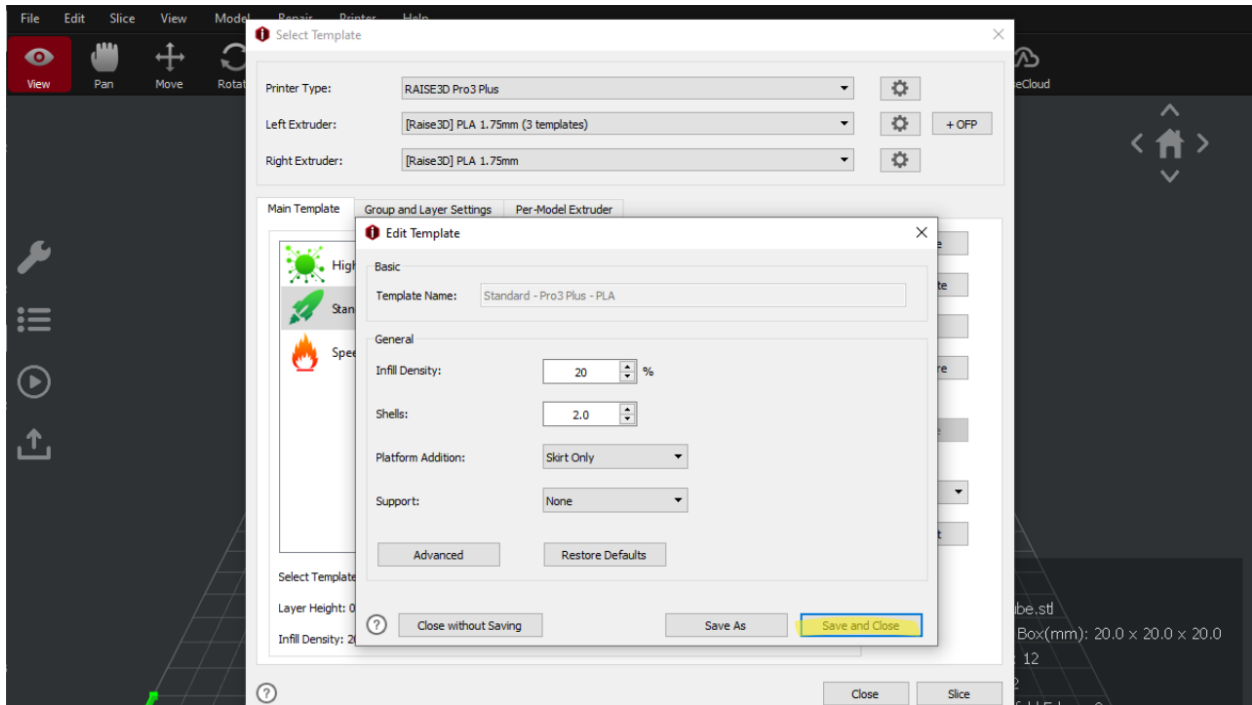
- **None:** do not add support to the model
- **Touch Platform only:** only generate supports which will touch the build platform



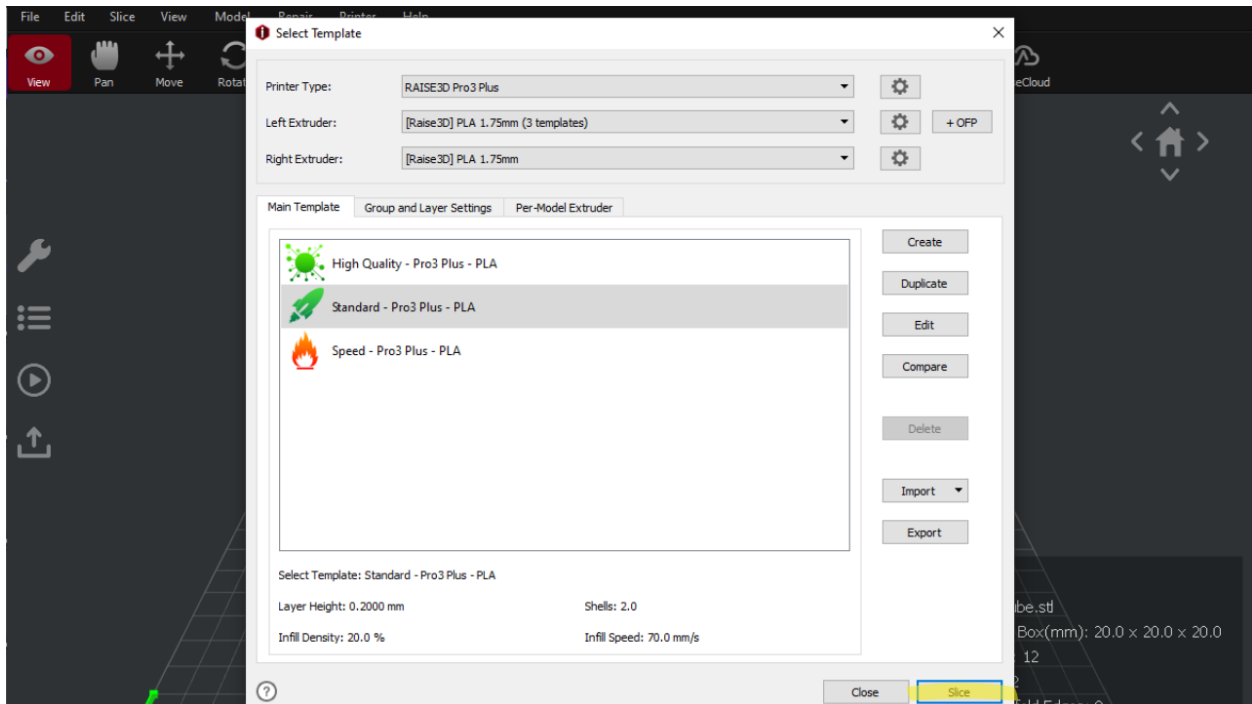
- **All:** add supports to all suspended and overhang structures of the model



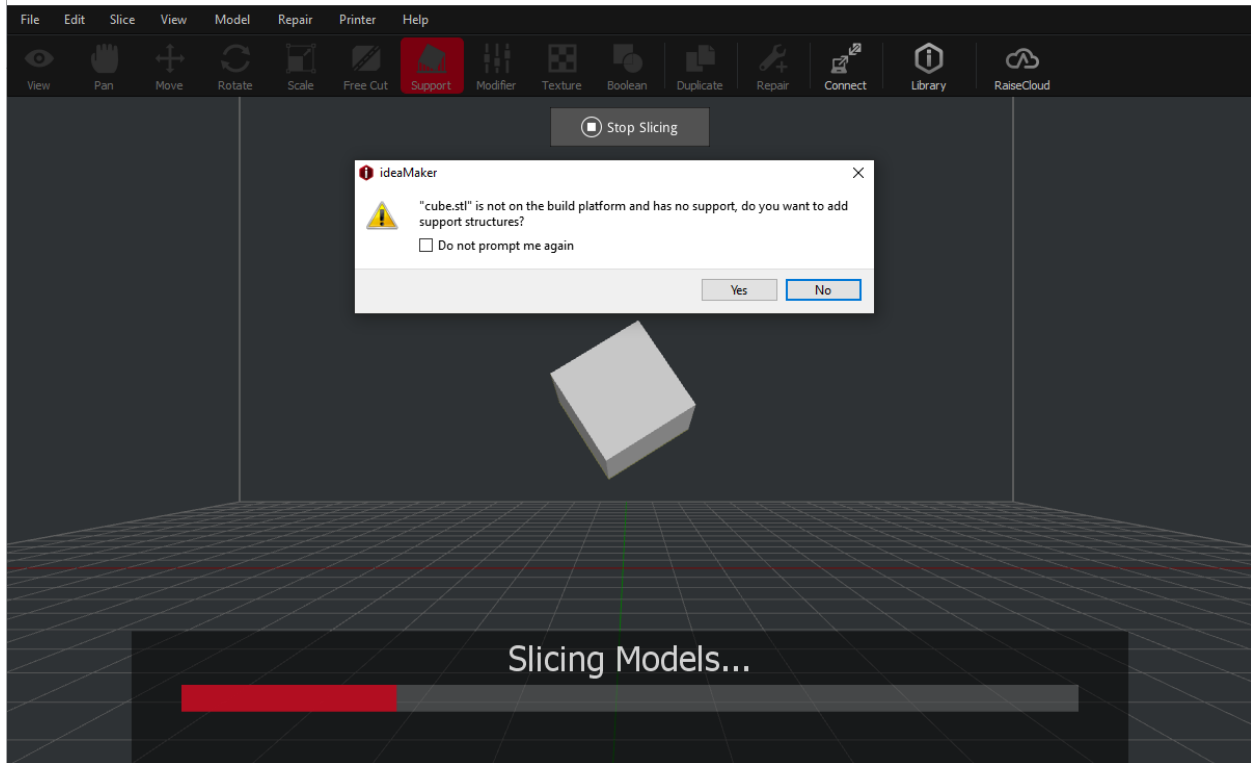
After changing to desired settings, select “Save and Close”



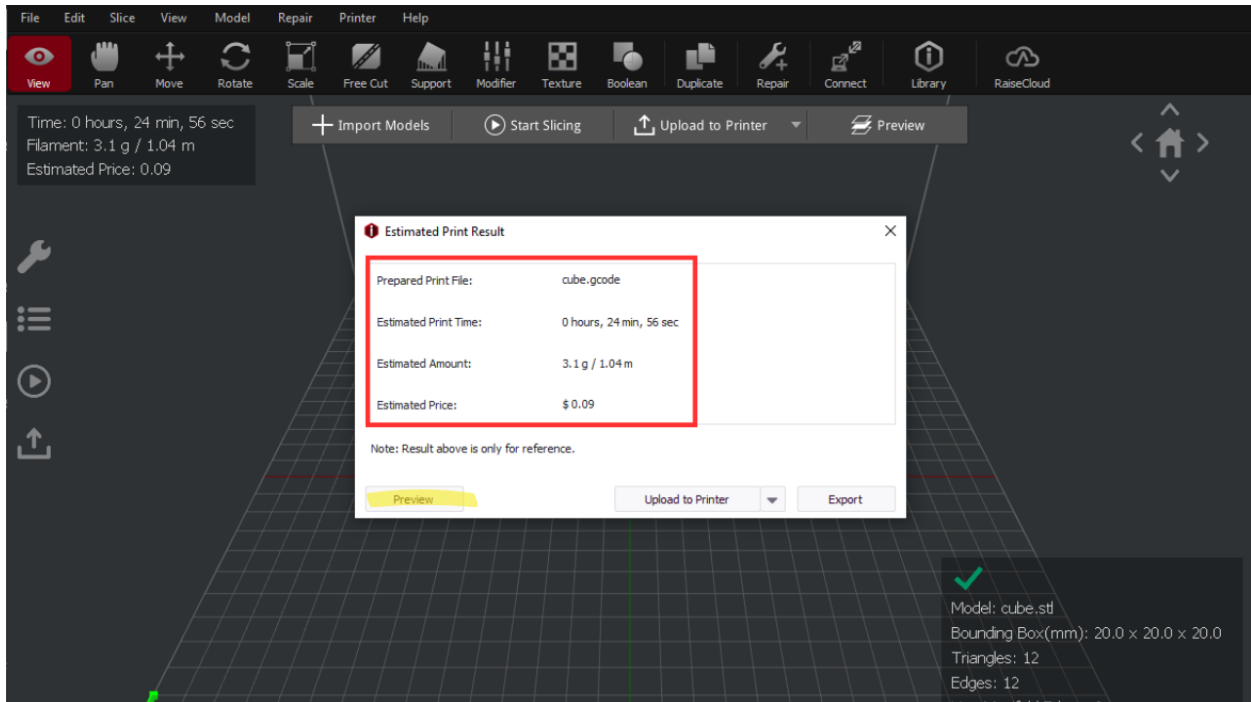
You will be taken back to the previous window. You can now slice:



If the model needs support but you did not include any, you will get a warning when you slice the model. By selecting yes, ideaMaker will generate the necessary support for your model.

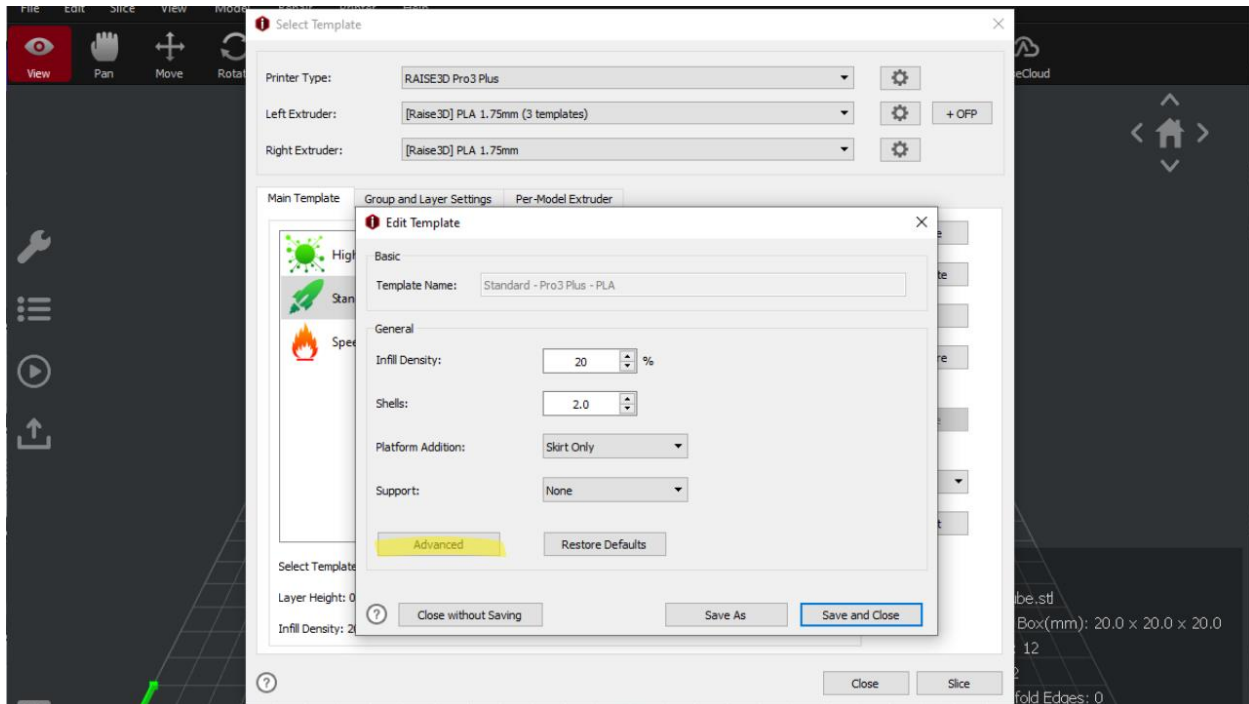


The “Estimated Print Result” window will populate. Here you can see the estimated print time of your model along with the estimated amount of filament it will use and the estimated price of your model. Click “Preview” to preview how your model will be printed.



## Section 4.3 Advanced Settings (Optional)

In the “Edit Template” window, select “Advanced”:



Due to the depth of the advanced settings, it is highly suggested to refer to the ideaMaker manual if you want to learn more: <https://support.raise3d.com/ideaMaker/download-manual-15-524.html> Otherwise, the following advanced settings are optional.

### Section 4.3.1 Layer

This is the exact height of each layer of plastic extruded by the printer. Our default is 0.2mm. Changes to this setting can increase your print’s speed, resolution, and smoothness. Layer height is typically measured in microns, where one micron is equal to 0.001mm. A larger layer height will result in a faster print, but this can decrease the resolution and quality of your print. Thus, it is only suggested to increase the layer height for larger prints. A smaller layer height can help smooth out your prints but will take longer to print.

When deciding on a layer height, the “Magic Number Theory” is often used: Set layer heights as a multiple of your stepper motors defined step distance. The max layer height for our 0.4 nozzles is 0.32mm. If the layer height is set too high for a small nozzle, this can cause layer separation and splitting. To prevent this issue, choosing a layer height that is 20% smaller than the nozzle diameter can allow the layers to properly bond together.

The initial layer height is important to ensure the first layer adheres to the print bed properly. Our default is 0.3mm. If this setting is too low, it can result in squishing or otherwise known as

---

“elephant’s foot. A good rule of thumb is to set the initial layer height slightly higher than the rest of your print layers.

Adding more shells can create thicker walls that result in a more durable print. Too thin of shells can have the infill peak through. Although, for intricately designed and decorative models, they do better with thinner shells to not distort the design.

Spiral Vase Mode is used to fix the “Z-Scar” on the outside and allows the model to print in one continuous line from the bottom to the top and removing a single start and stop print. Useful for vases.

### **Section 4.3.2 Extruder**

The extruder settings can be changed to prevent stringing. If you see strings or wisps on your print, you should think about changing the retraction settings. The retraction determines how much and how fast filament is pulled back into the nozzle to prevent material from oozing out when it’s not being extruded. This is controlled by the retraction distance and retraction speed. In some cases, stringing can be a result of the nozzle temp. Before changing that setting, adjust the retraction settings in small intervals first before trying to lower the nozzle temp. Too much retraction can cause nozzle jams.

### **Section 4.3.3 Solid Fill**

The top layer thickness prevents any pillowing or sagging. The bottom layers influence the strength and stability of your base of your print. To prevent any issues with your top and bottom layers, they should be 6 to 8 times greater than the layer height.

Ironing is used to help achieve a smooth top surface. This is done by after finishing the topmost layer, the nozzle goes over the surface at a slow speed with a low flowrate and uses the heat from the nozzle to fill any gaps and smoothing out any bumps.

### **Section 4.3.4 Support**

If a bridge is less than 10mm in length, the printer may be able to print it without supports

### **Section 4.3.5 Cooling**

This is very important to change if there are large unsupported overhangs or bridges being printed and the layers need to be cooled quickly. The fan is usually turned off when printing the first layer to improve adhesion.

---

### **Section 4.3.6 Temperature**

Temperature can play an important role in 3D printing. Too high of a nozzle temperature can cause over extrusion and blobs on the model. Too low of a nozzle temperature can cause under extrusion and gaps in the layers. A low bed temperature can cause warping and no adhesion. A too high bed temperature can cause deforming. With that in mind, a good rule of thumb is between 50°C-60°C.

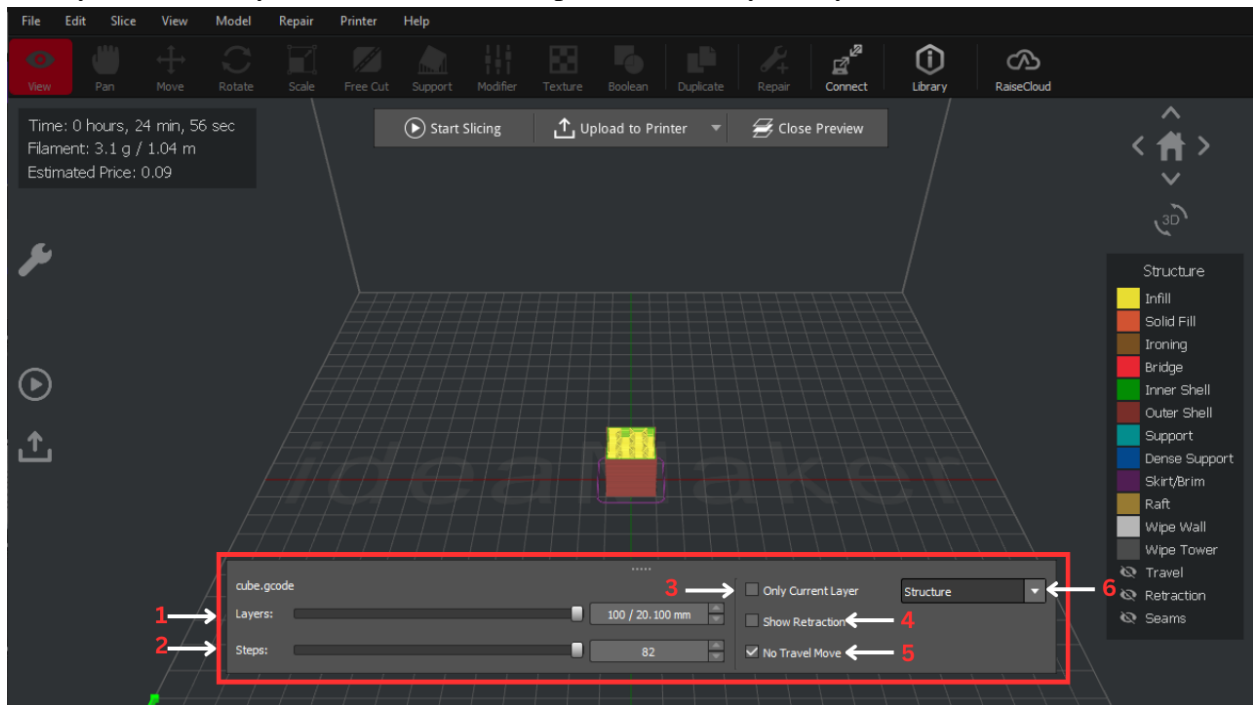
### **Section 4.3.7 Speed**

It is a good note to leave any of the speed settings along and only adjust the default speed if necessary. If you are having print quality issues, you may want to reduce the speed. The slower speeds can also be used to identify which settings is causing problems.

The other advanced settings categories rarely need to be changed.

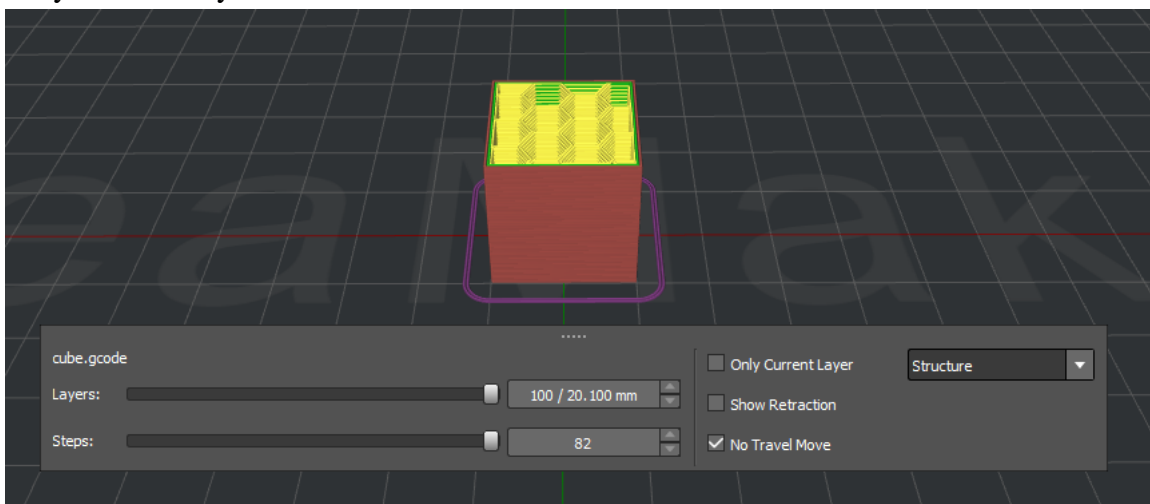
## Section 4.4 Previewing Your Model

Here you can view your model before it is printed to verify how you want it.

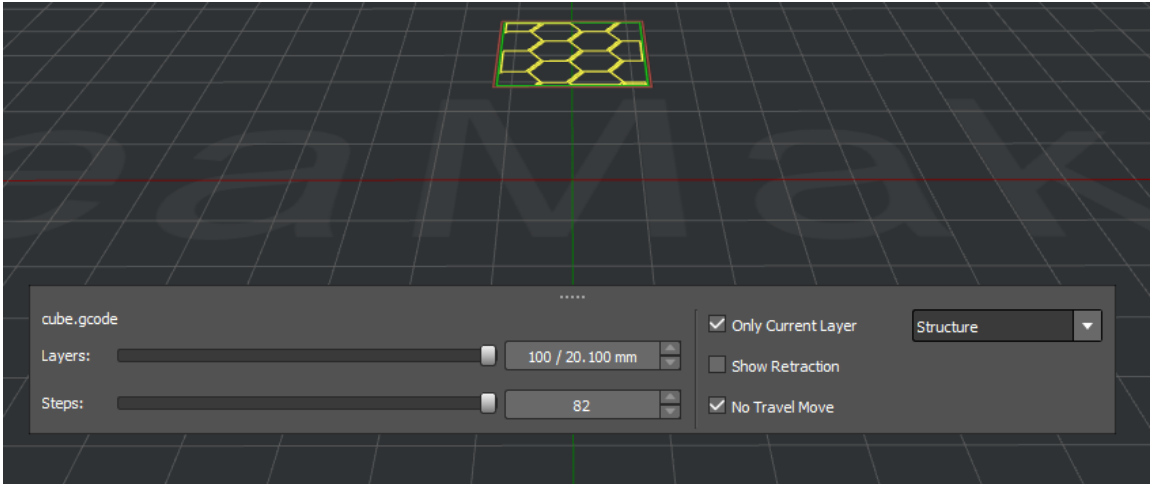


- **Layers:** by clicking and dragging the slider, you can see what each layer of your print will look like
- **Steps:** by clicking and dragging the slider, you can see what each step of the current layer will look like.
- **Only Current Layer:** if this is deselected, you will see all previous layers including the current layer you are on. If it is selected, however, you will only see the current layer you are on.

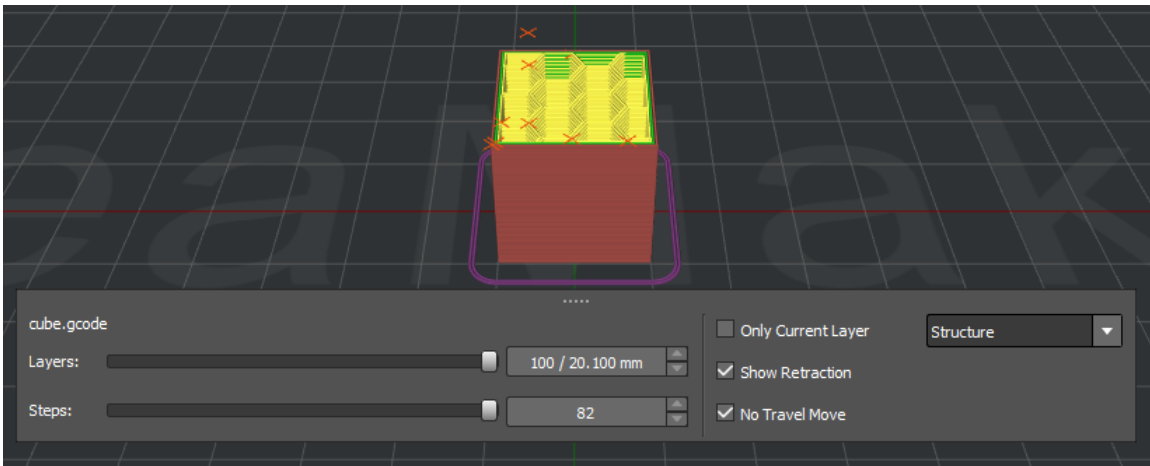
Only Current Layer deselected



Only Current Layer selected:

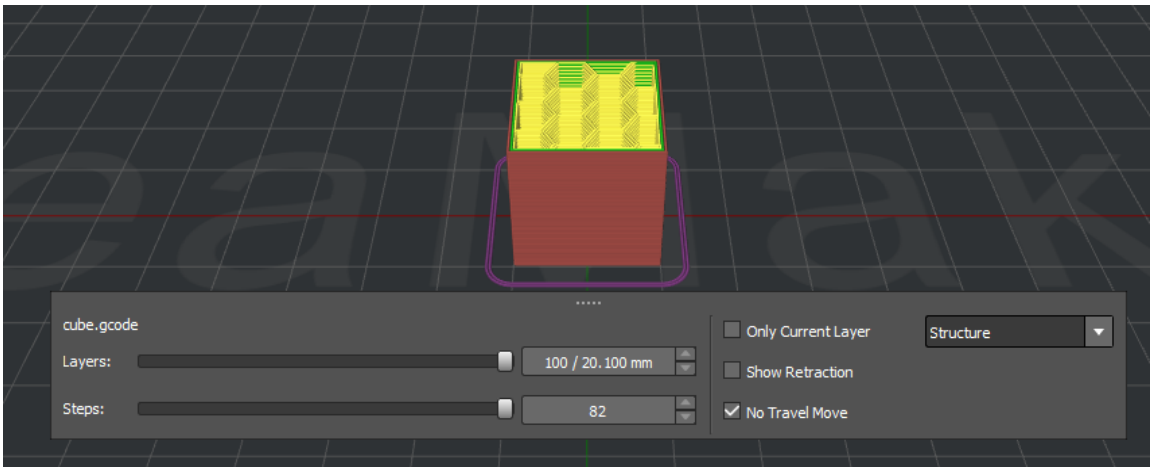


- **Show Retraction:** if this is selected will show the points in the print where the filament is being retracted as the print head moves



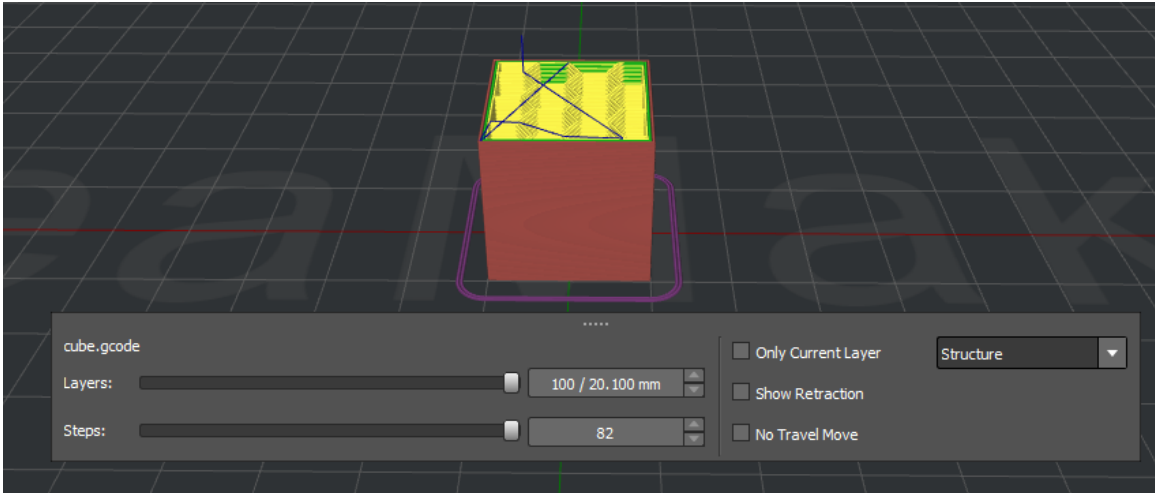
- **No Travel Move:** if this is deselected, you can see the movement of your print head for every step.

No travel mode selected:

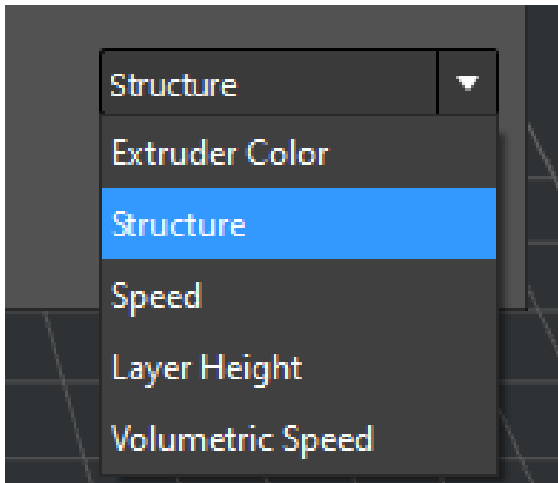




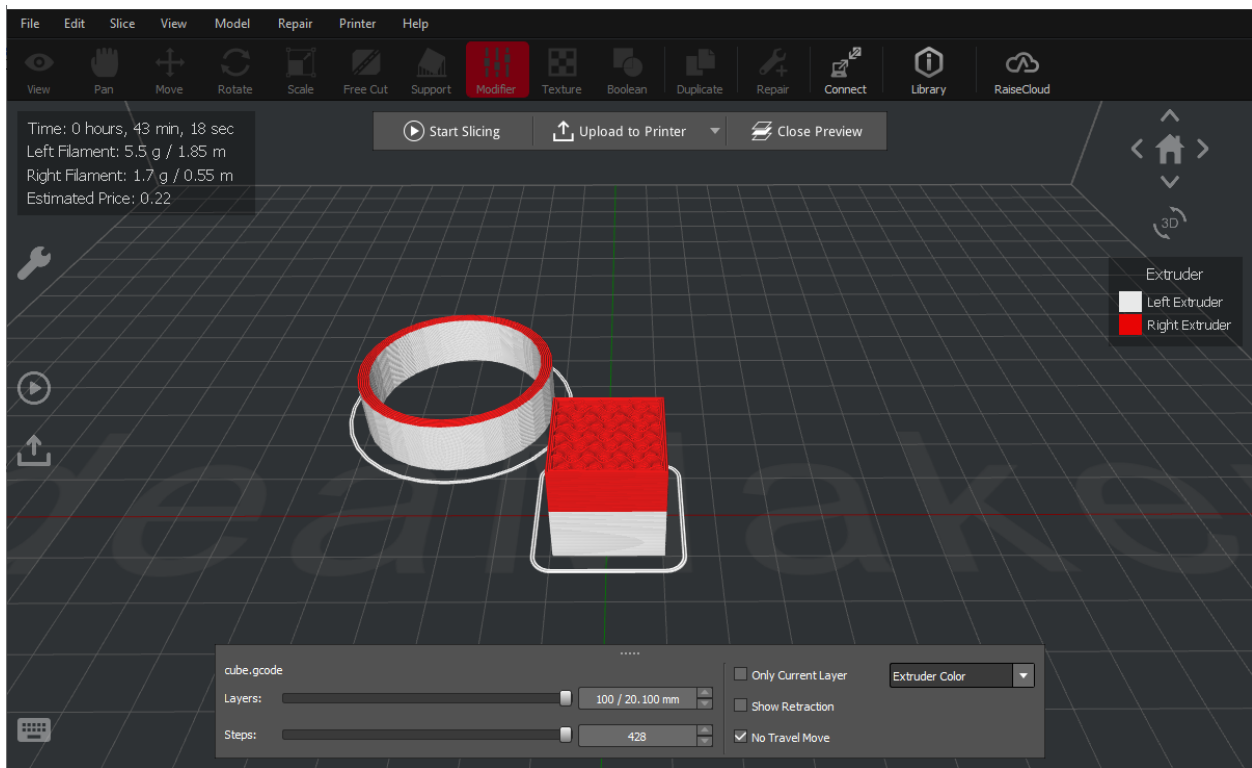
No travel mode deselected:



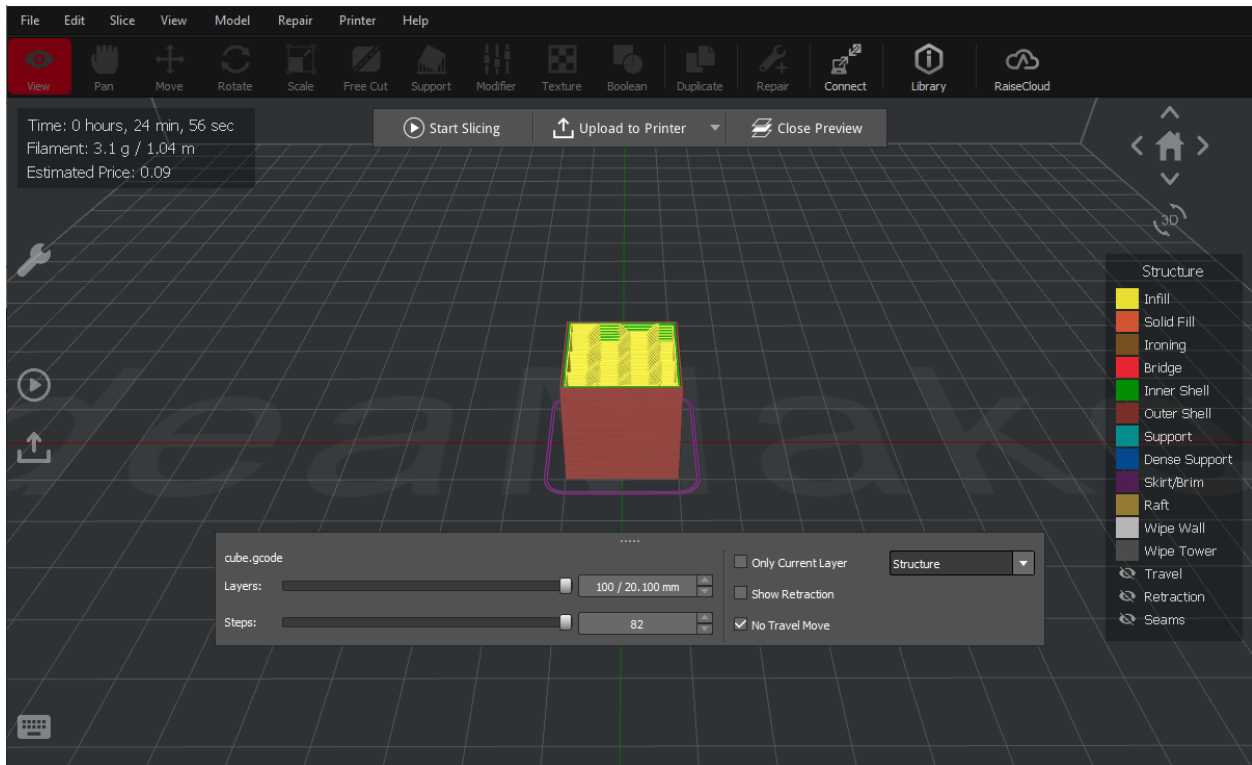
- **View mode:**



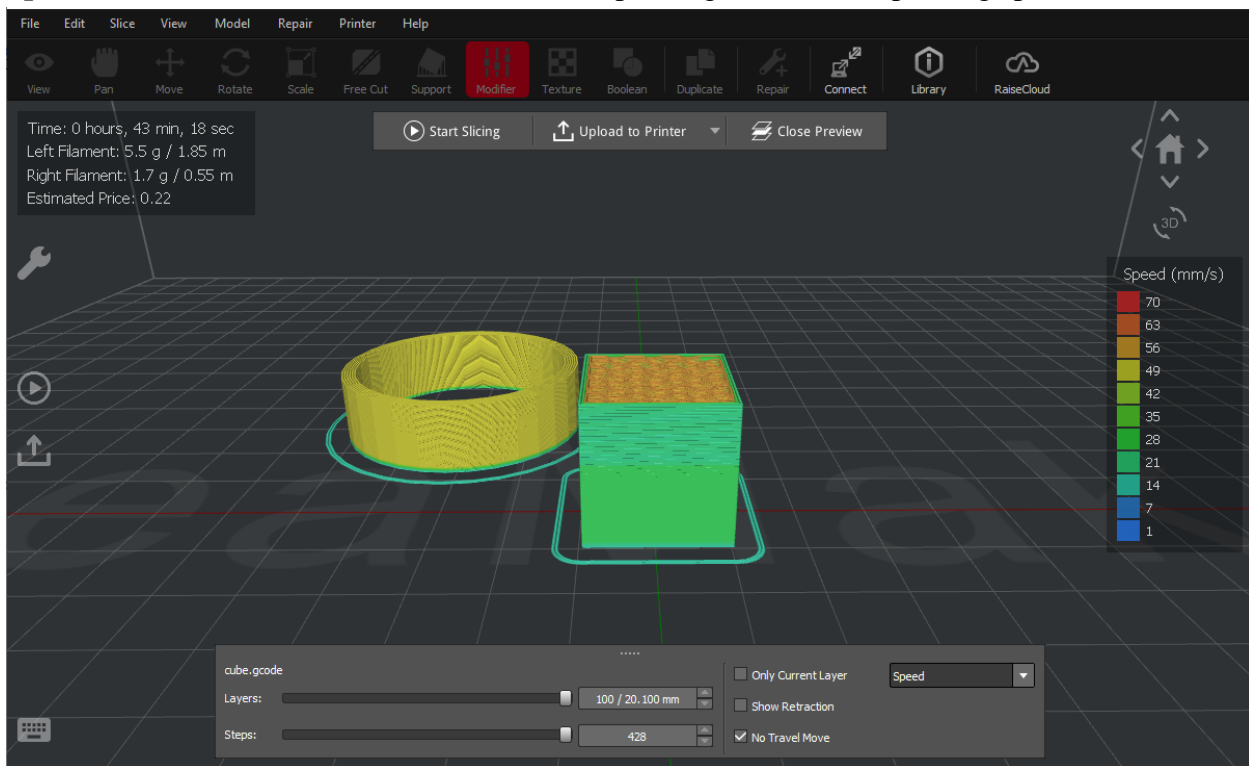
**Extruder Color:** if your print is dual color, ideaMaker will show the model with selected colors.



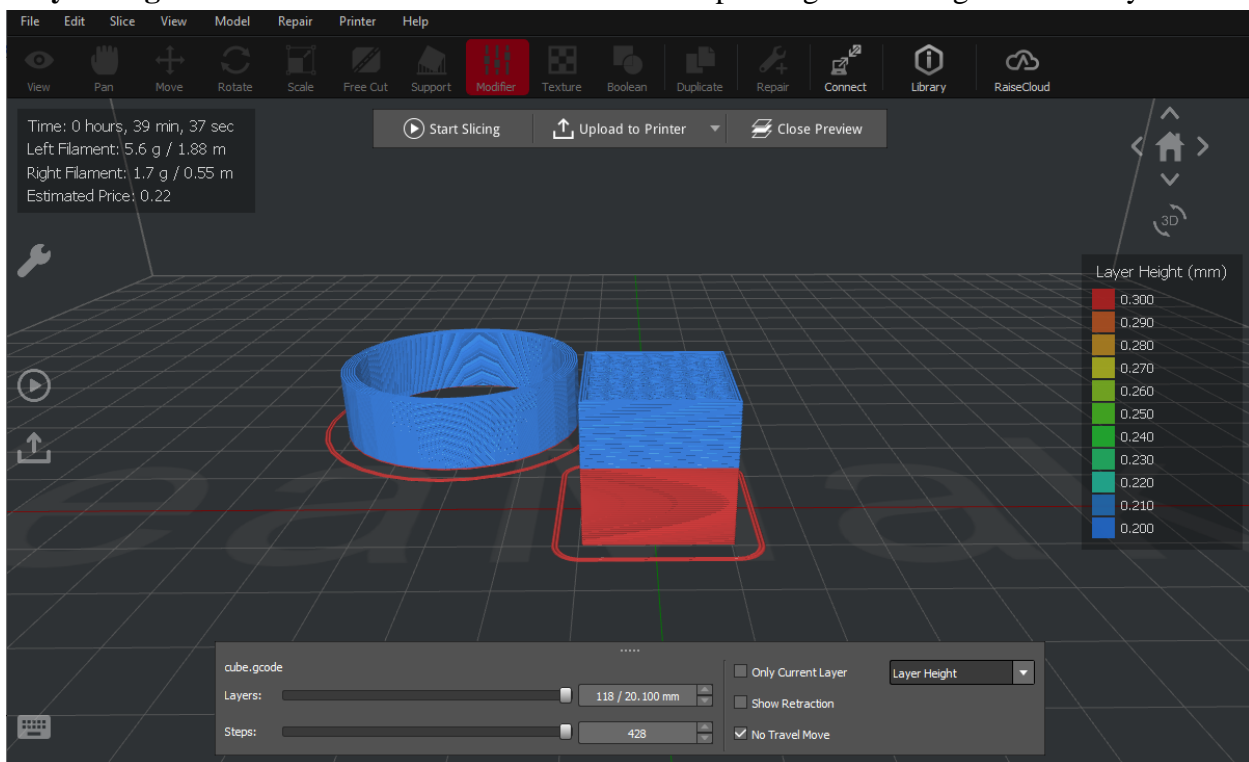
**Structure:** ideaMaker will show the different structures in the model using the color scheme shown on the right hand side



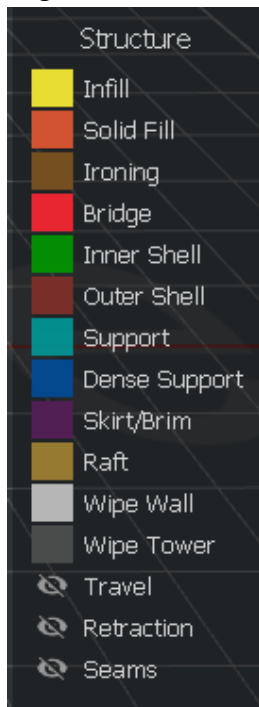
## Speed: ideaMaker will color code the model depending the different printing speeds



## Layer height: ideaMaker will color code the model depending on the height of each layer



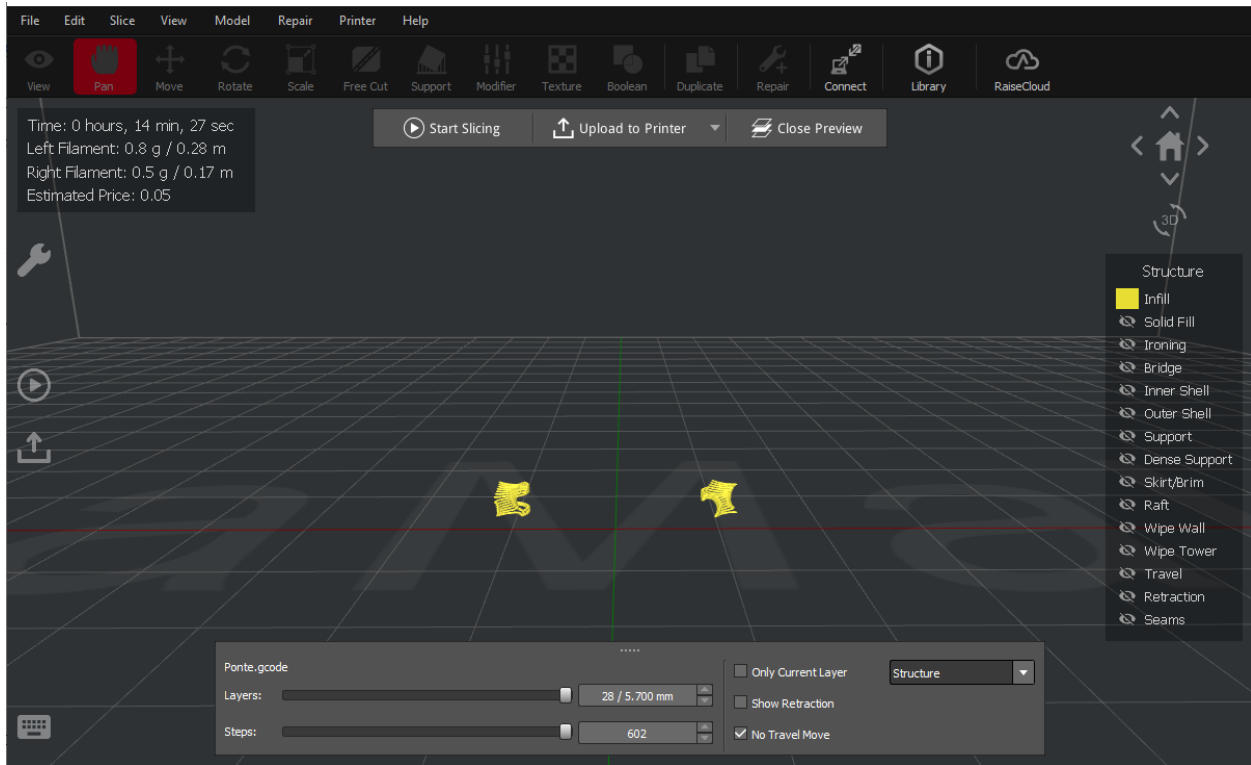
Right hand side of “Structure mode”:



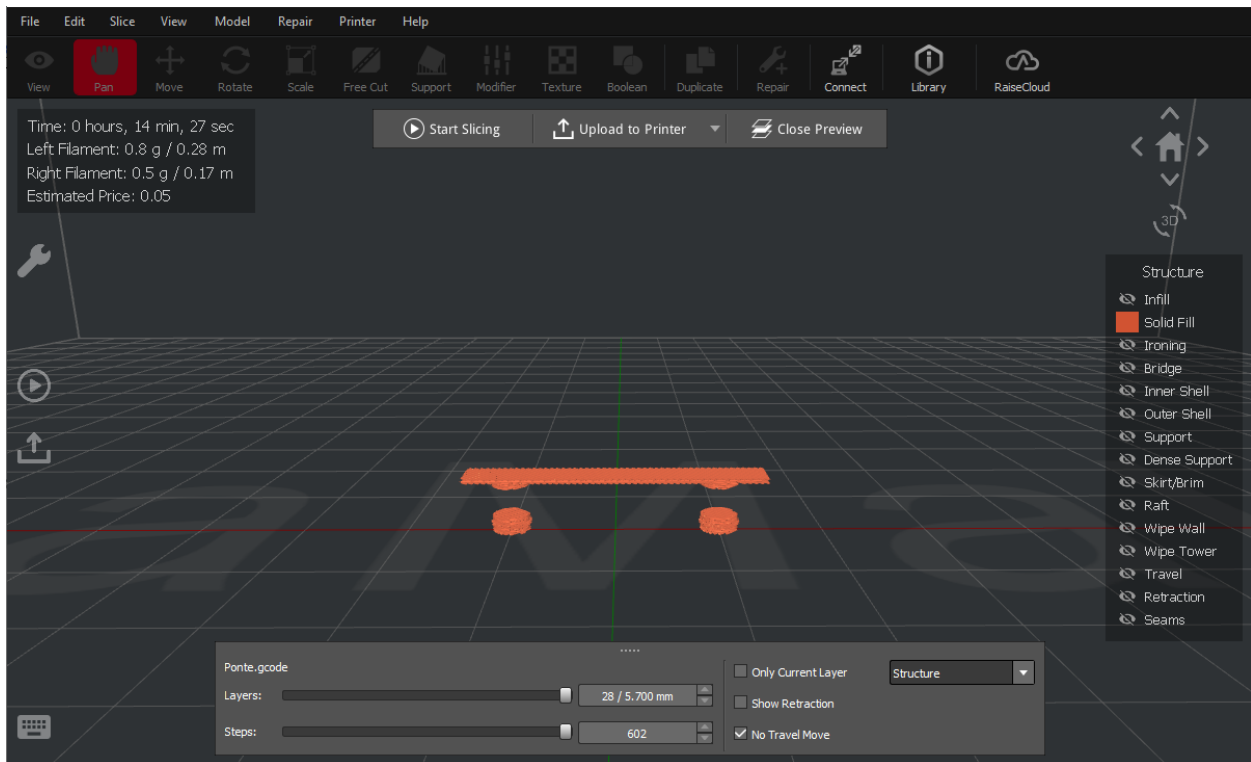
The model being used in example:



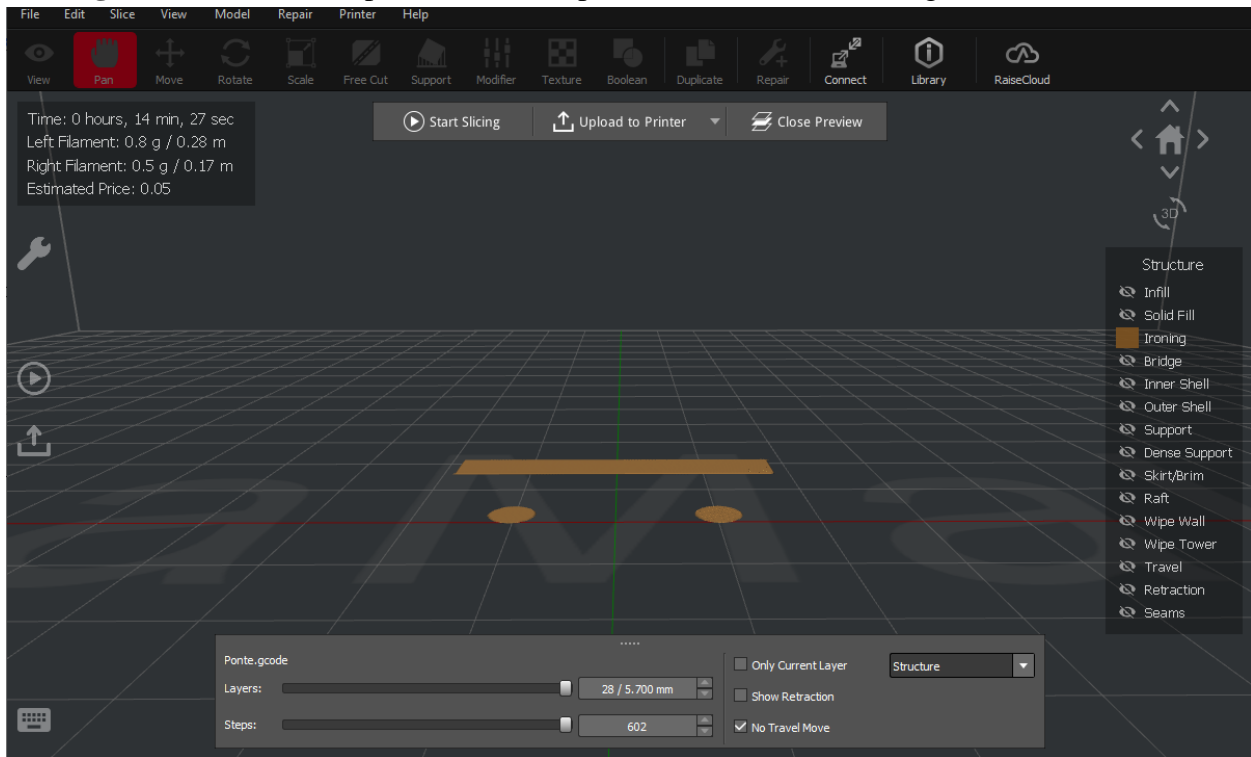
**Infill:** Provides support for top layers, which would otherwise have to bridge over empty spaces



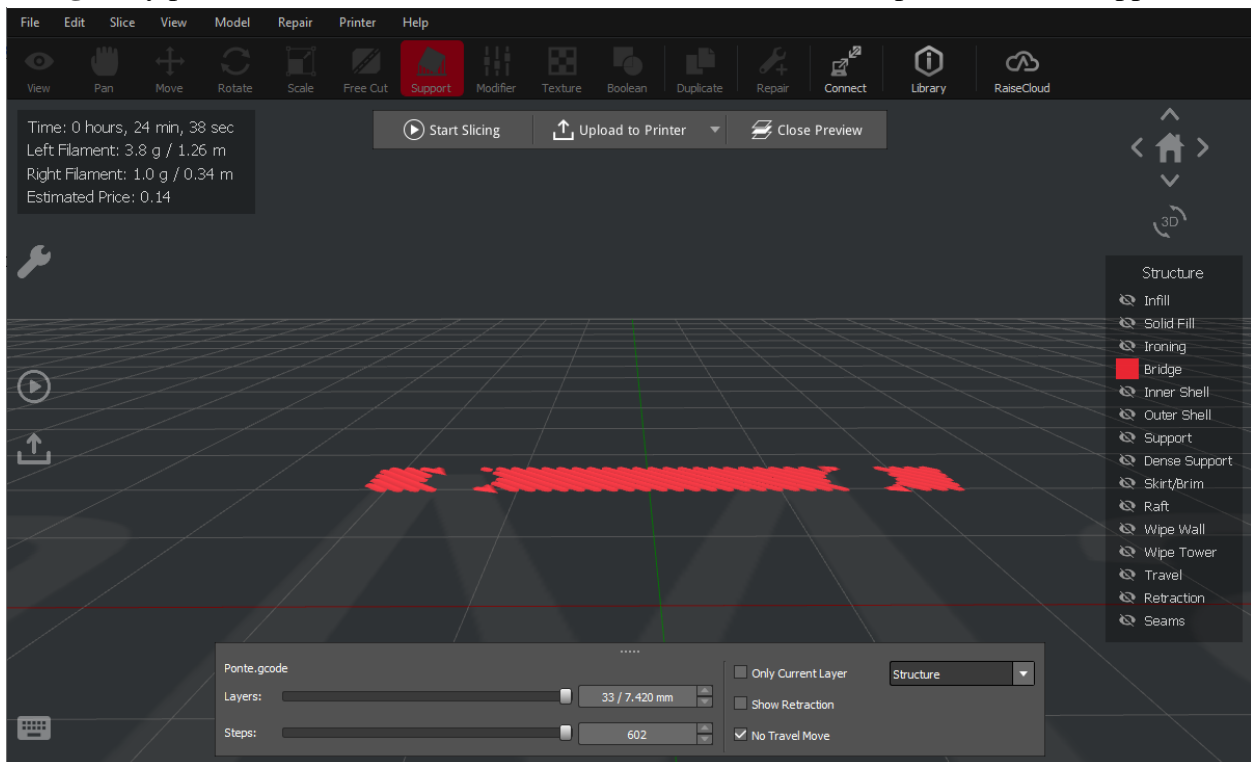
**Solid fill:** any 100% infill



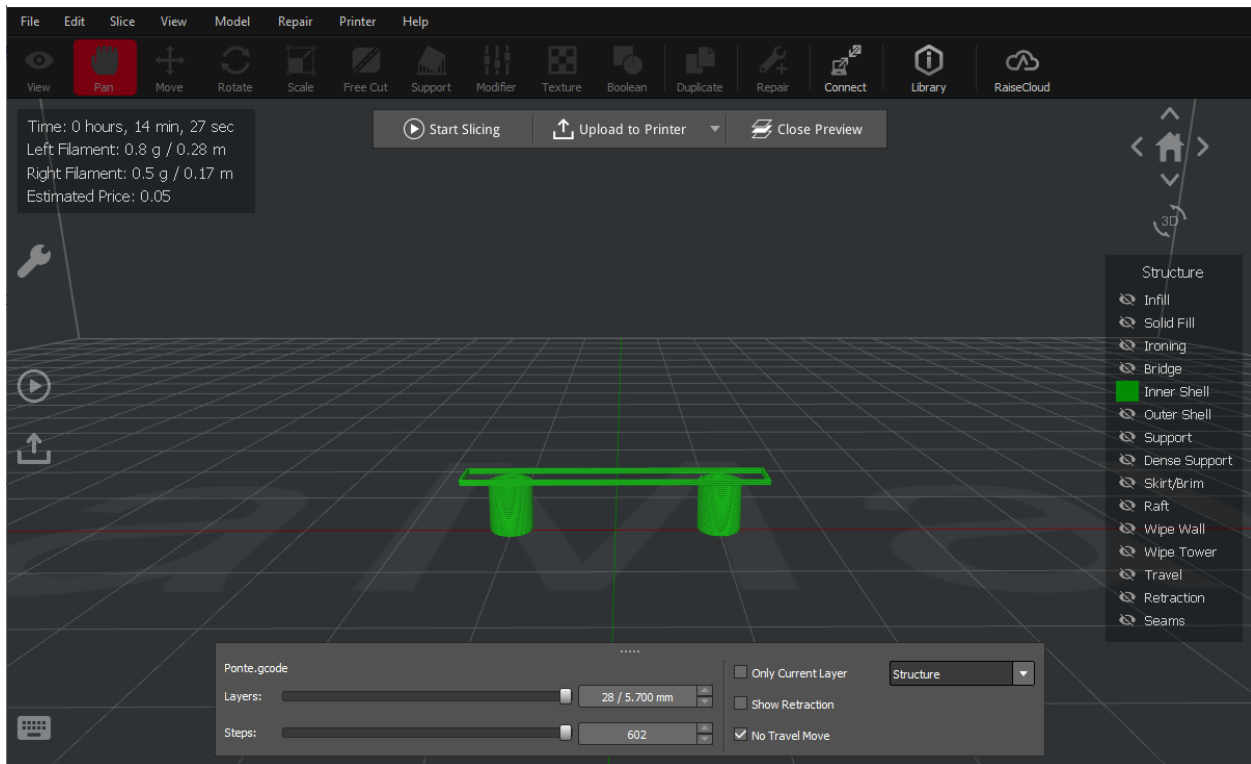
**Ironing:** smooths the flat top surface. This option is in solid infill setting



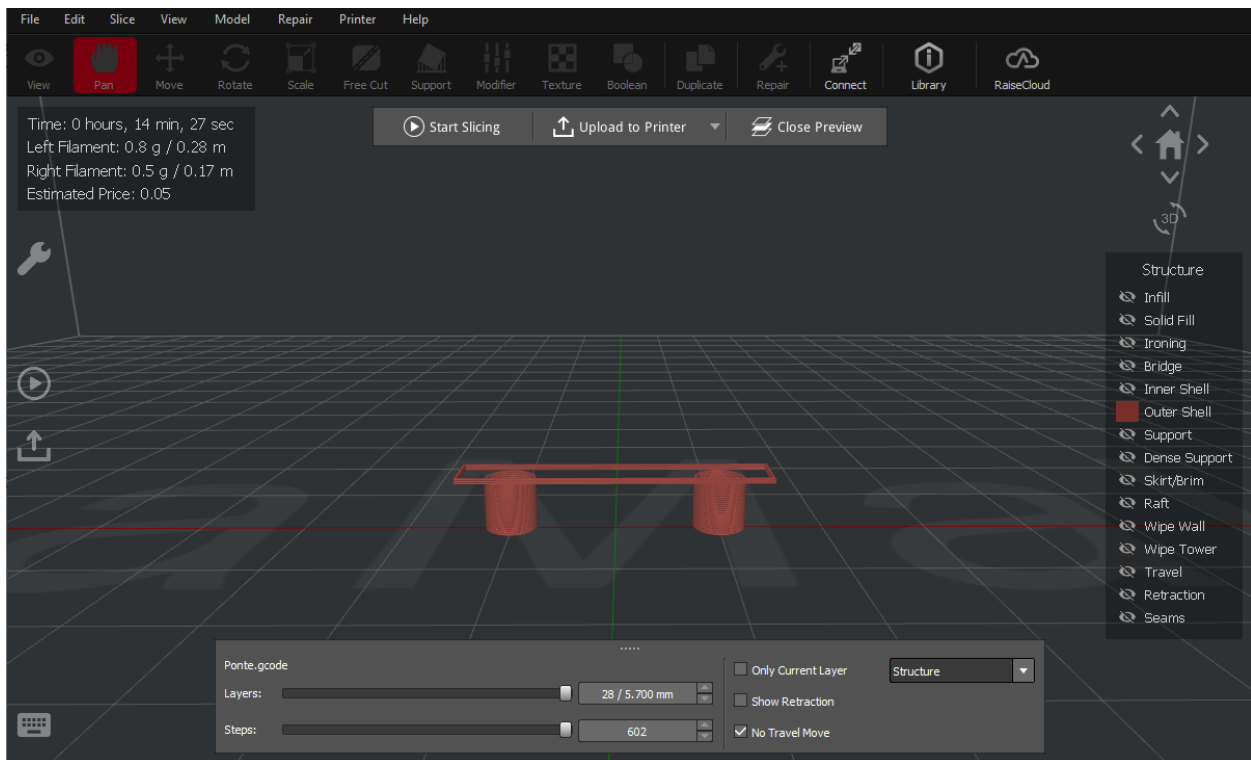
**Bridge:** any part of the model that needs to be extruded between two points without support



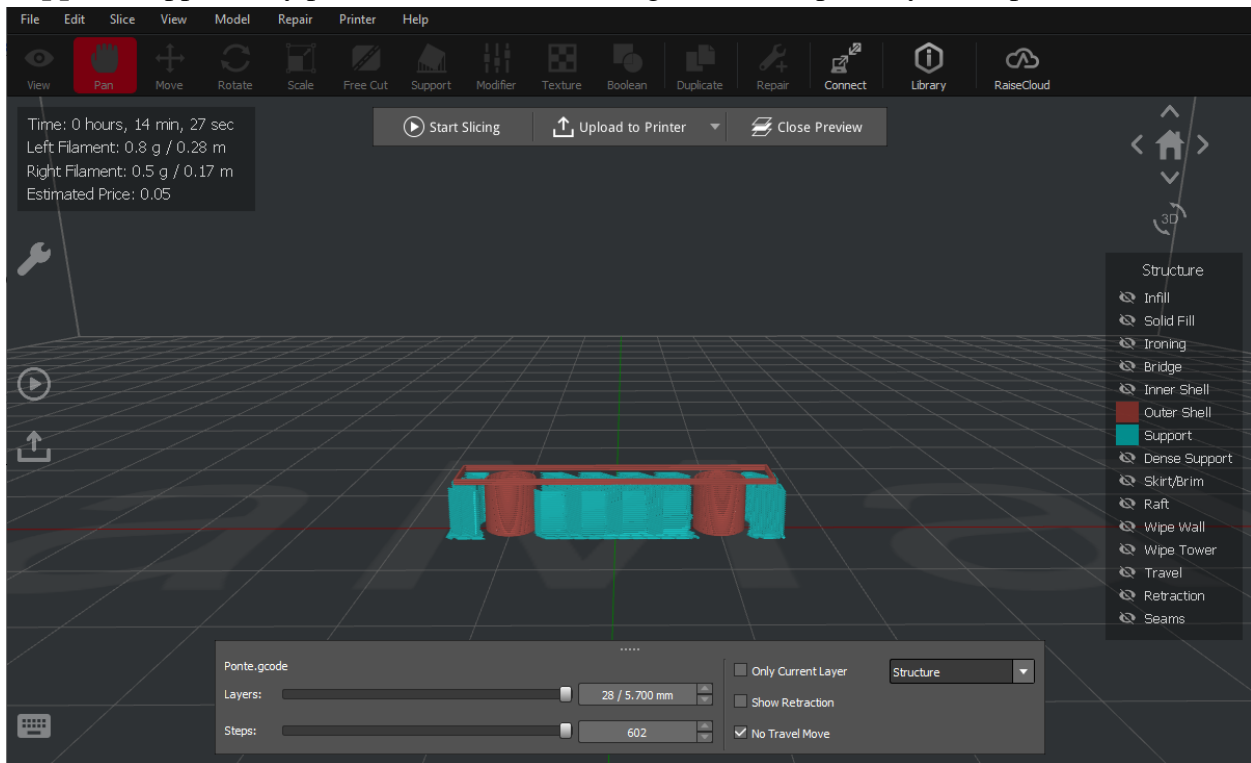
## Inner shell: backup shells behind the outer shell that give outer shell extra support



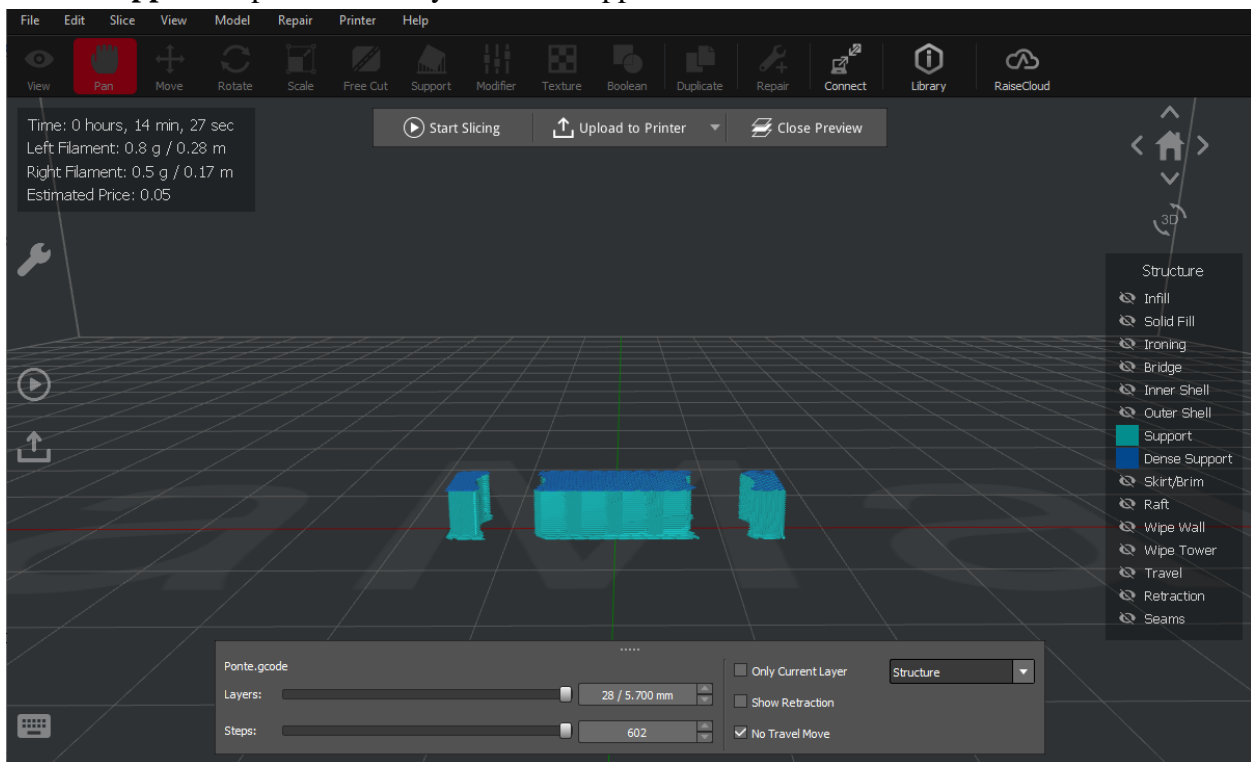
## Outer shell: outer most shell that we see



**Support:** supports any part of model that overhangs and could possibly cause print failure

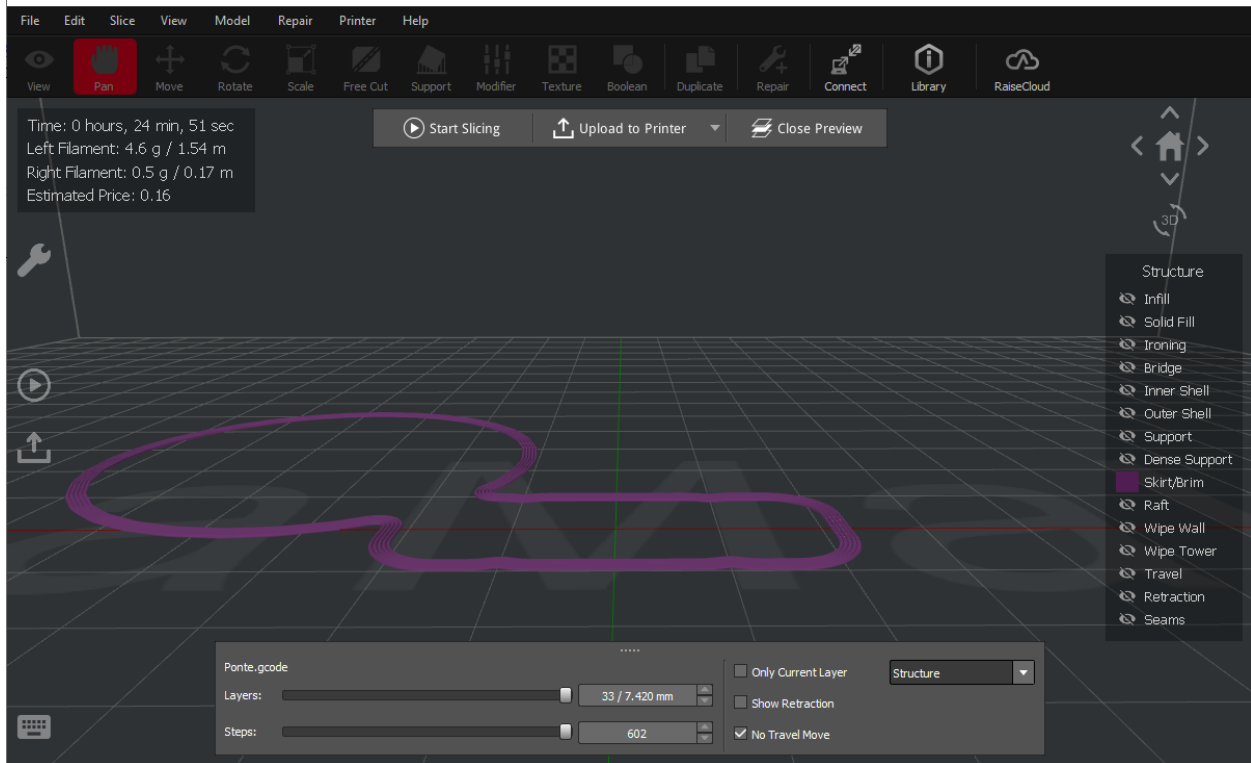


**Dense Support:** topmost solid layers of the support structure that touch the model

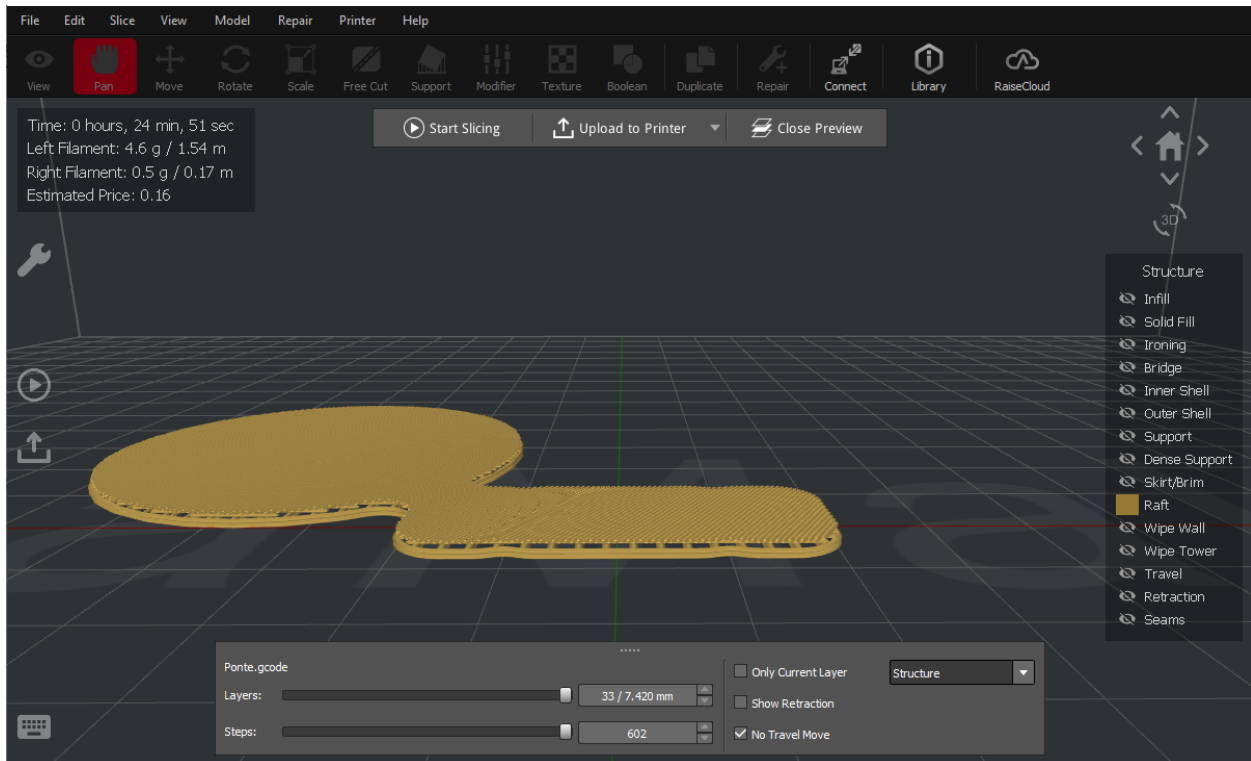




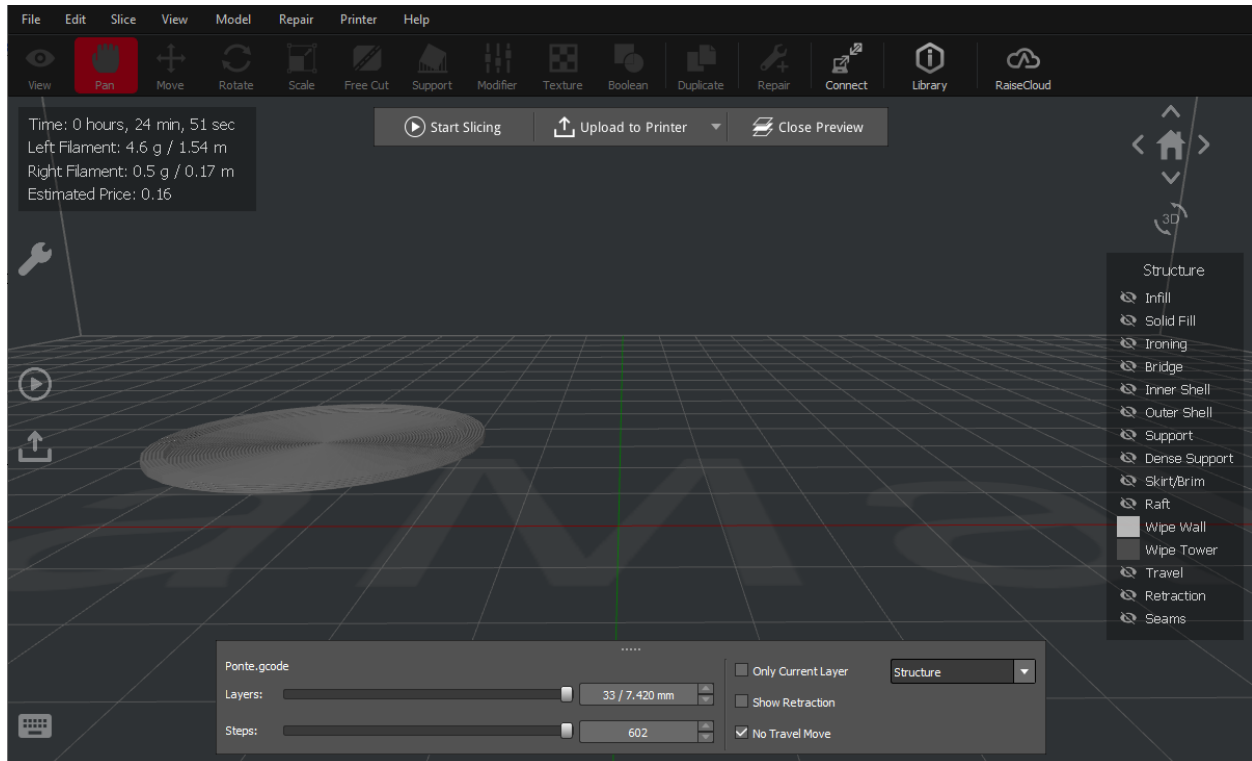
**Skirt/Brim:** are loops of thermoplastic that are added around the model being printed. Skirt ensures that the print nozzle is clean, and brim helps model adhere to the build platform better



**Raft:** provide a stable foundation for the printed parts. Use only if the model has a very small contact area with the build plate and requires significant support structures.



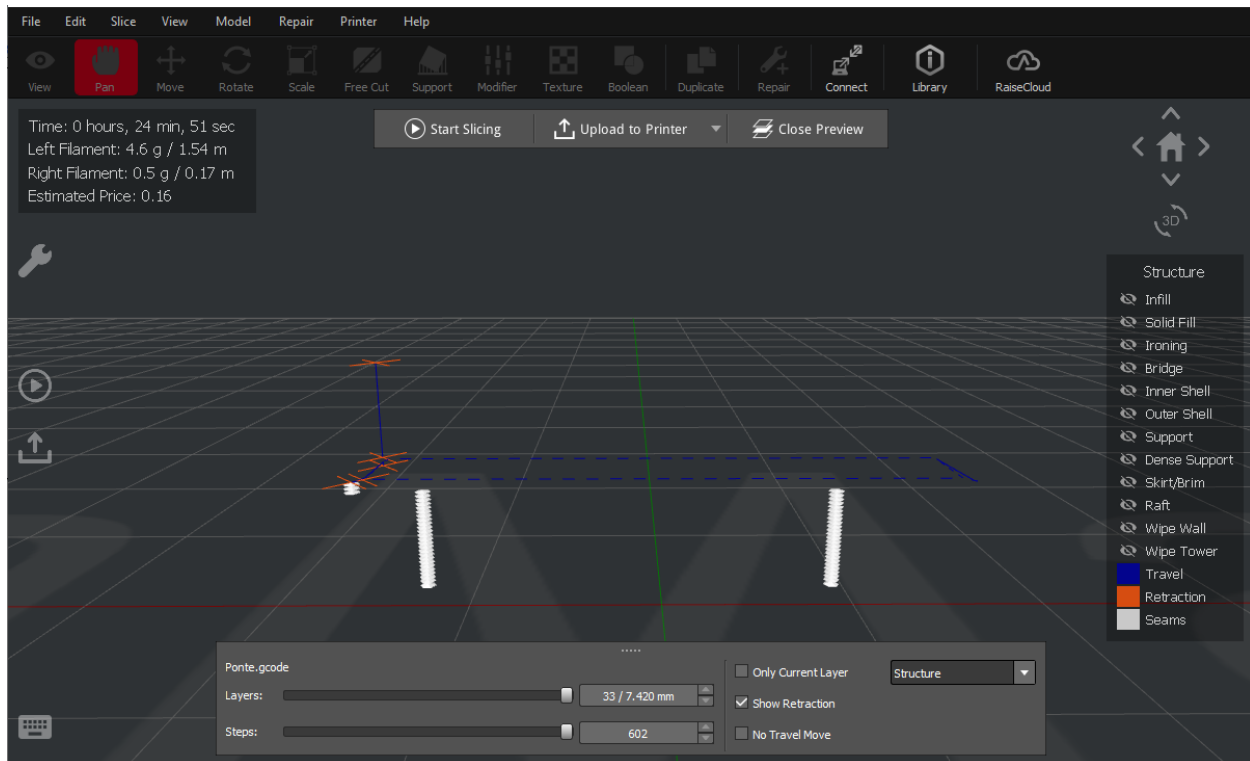
**Wipe wall/tower:** in dual extrusion printing, wipe wall or wipe tower are shell layers printed away from the model to produce cleaner prints after extruder switch



**Travel:** the path that the printer head is traveling as it prints the model

**Retraction:** the points where the extruder is retracting the material within the printhead during movement

**Seams:** the point where the printer's extruder nozzle begins a new layer



## Section 5. Naming Convention for File

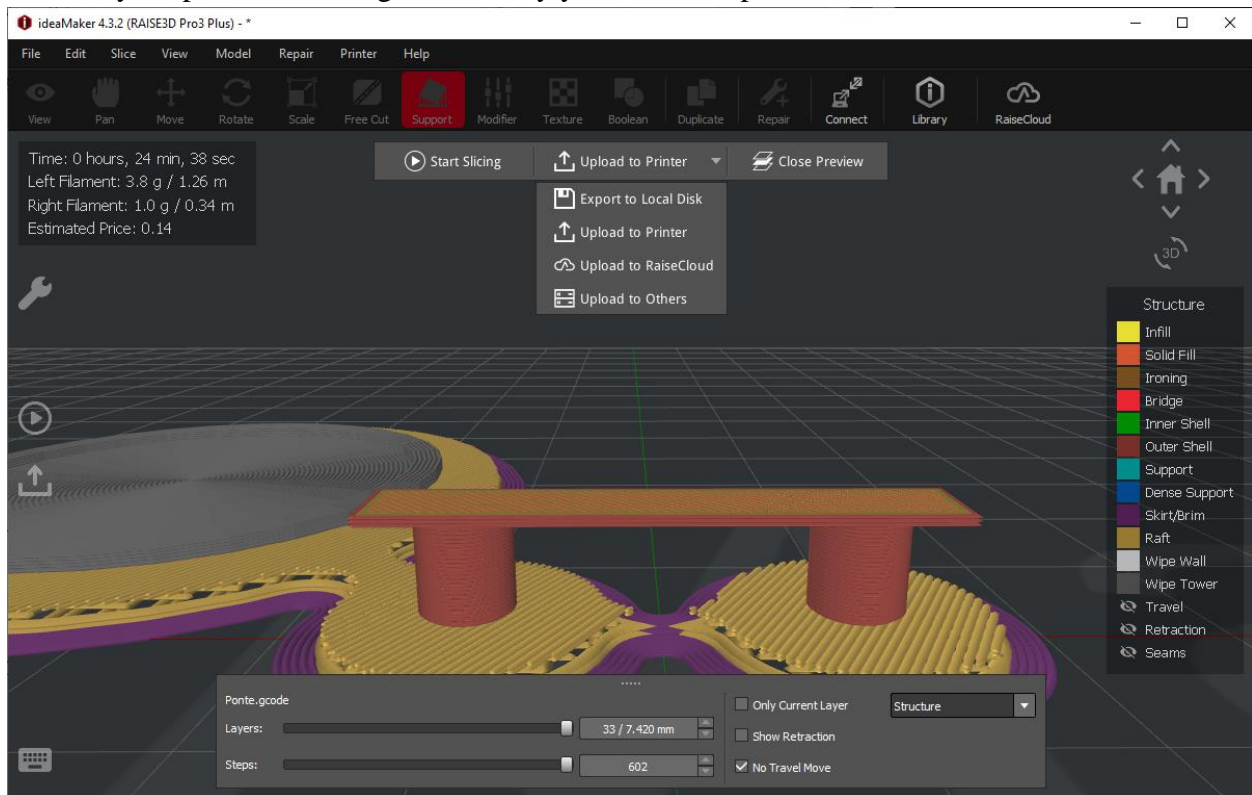
It is important to rename the file to follow the format required by the center:

*LastName\_firstName\_month-day*

Ex. Wasowski\_Mike\_03-02

## Section 6. Printing your Print

You are now ready to send your print to the printer! After you verified and confirmed your print, please ask a technician for help. The technician will need to confirm the print is appropriate per the Maker Center rules and that the file is properly formatted. Depending on printer availability, the technician will help guide you to preparing your print for an available machine and will start the print for you. In case of no printer availability, a technician will place your print into a queue, confirm your preferred settings, and notify you when the print is finished.



## Section 7. Instructions for Pickup

You will be contacted, using your preferred preference through Sine, on when your print is ready to pickup. In case of print failure, you will also be contacted regarding next steps.

---

## **Conclusion**

Congratulations, you printed your first 3D print! Hopefully, this is just the beginning of your 3D printing journey. Use this manual as a guide to look back on or reference anything.

---

# Glossary of Common 3D printing Terms

<https://www.raise3d.com/academy/3d-printing-glossary/>

## 123

**.3mf**- 3D Manufacturing Format is an open-source file-type designed for additive manufacturing. Can include information that cannot be in STL files.

**.amf**- Additive Manufacturing File Format. Similar to STL but can include more information such as colors and materials.

**.ply**- A polygon file format commonly used with 3D scanners.

**.vrmf**- Virtual Reality Modeling Language is an older file format used to represent 3D graphics, replaced by X3D.

**3D BioPrinting**- The utilization of 3d printing technologies to fabricate biomedical objects as similar to the real thing as possible.

**3D File**- An electronic file created in CAD programs to represent a 3D object.

**3D Model**- A model of something made in three dimensions using CAD software.

**3D Modeling**- Making a 3D model in CAD software.

**3D Modeling Program**- A program used to make 3D models.

**3D Positioning System**- A system used to find the position and location of an object in 3D space.

**3D Printer**- A manufacturing machine used to create 3D objects one layer at a time. There are different types of 3D printers, each utilizing different technologies.

**3D Printing**- 3D printing is an additive manufacturing process that uses thin layers of filament to create a physical object from a three-dimensional model. Learn more about 3D printing here

**3D Printing Pen**- A handheld tool that uses FFF technologies to melt thermoplastics, allowing the user to “3D draw” with melted plastic that hardens quickly.

---

**3D Scan-** A 3D model made by a 3D Scanner of a real-world object.

**3DPrinterOS-** 3D printing cloud-based management.

**45° Rule-** The rule that an object with angled features designed greater than 45° will most likely need support(s) when printing. Bridging excluded.

## **A**

**ABS-**Acrylonitrile Butadiene Styrene (ABS) is a thermoplastic printing filament commonly used in 3D Printing. ABS is stronger than PLA and more durable.

**Acetone-** Acetone is a liquid commonly used in 3D Printing for vapor baths. ABS is soluble in acetone, so if you put ABS prints in a container with acetone vapors, it will smooth all of the surfaces.

**Additive Manufacturing-** The process of manufacturing a 3D object by adding material together to create a final object. 3D printing is an additive manufacturing method.

**Adhesion-** An object's ability to stick to a surface. Normally referring to bed Adhesion, this is how well a 3D print adheres to the printing surface on the print bed.

**Alias-** A product design software by Autodesk.

**Amorphous-** Without a clearly defined shape or form. Normally referring to Amorphous Metals that can be 3D printed.

**Anisotropic-** A material is anisotropic if it has different strengths in different directions. Wood is a common example. FDM 3d prints are anisotropic.

**ASA-**Acrylonitrile Styrene Acrylate. ASA is a strong thermoplastic that can be 3D printed. It is similar to ABS but has great weather resistance. This makes it great for any outdoor application for 3D printing.

**Atomic Method-** A technique used to unclog a clogged print nozzle of an FFFDM 3d printer.

**Autodesk Dreamcatcher-** A 3D design software used to generate and optimize designs.

---

## B

**Bed-** The bed of the printer is the print surface that a part is printed on.

**Bed Leveling-** Adjusting the bed of a 3D printer to make it as level as possible. This is an important step of 3D printing as it helps ensure a successful print.

**Belt-** The belts on a 3D printer takes the rotation of a stepper motor and converts it into linear motion for the print head. Belts are usually teathed and tight to ensure a precise motion transfer.

**Binder Jetting-** Binder Jetting is an additive manufacturing method where powdered substances are joined by a liquid binding agent. Binder Jetting can be done in lots of materials and colors.

**Blender-** A free 3D graphics software that can be used to create 3D models. It is open-source and is also used to make animations, visual effects, and motion graphics.

**Blue Painters Tape-** Used in FDM printing on the bed of a printer to improve adhesion. Often used in FDM 3D printing to improve bed adhesion. A cheap solution to help a part stick to the print bed.

**Bottom/Top Thickness-** A slicer program setting that is used to determine how much material will be laid down before the infill printing starts and how much material will be laid down after the infill printing is finished. See also Slicer.

**Bowden Extruder-** A method of conveying thermoplastic filament used by some FDM-type 3d printers. On a printer with a Bowden extruder, the cold end is separated from the hot end and attached somewhere on the printer frame. See also Cold End and Hot End.

**Bowden Tube-** The part on some FDM-type 3d printers with a Bowden extruder setup. The Bowden tube is used to guide thermoplastic filament from the feeder assembly in the cold end to the hot end where it is heated and extruded.

**Bridge-** A 3d modeling term to describe a horizontal overhang placed between two vertical supports.

**Brim-** A brim is a layer or layers of extruded thermoplastic that is used to stabilize small parts or islands on a printed object. A brim helps these areas to adhere to the print bed. Unlike a raft, a brim is connected only to the perimeter of an island, not to the bottom.

**Brittleness-** A property of materials where it breaks without significant deformation. Chalk and ceramics are examples of brittle materials.



---

**Build Plate-** The area where a 3D print is printed upon.

**Build Resolution-** Typically refers to the layer height that a 3D print is printed at. Similar to the resolution on a television or computer monitor but in 3D the lower the build layer height the higher the part resolution.

**Build Surface-** The surface on which a printed object is produced. Often various types of build surfaces will be placed onto or attached to the printer bed to improve adhesion.

**Build Time-** The total time it takes for a 3D printer to complete a 3D print.

**Build Volume-** The maximum size of an object that a 3d printer can produce, measured in length times width times height.

**BuildTak-** Used in FDM printing on the bed of a printer to improve adhesion. More information on the use of Buildtak can be found [here](#).

## C

**CAD-** Short for Computer Assisted Design. CAD is the use of computer software to produce a digital design in either two or three-dimensional formats that can then be used to print a physical object. CAD was originally developed for use in architecture and engineering. However, there are now several user-friendly applications on the market that are either free or available at a low cost.

**Cartesian Coordinates-** A system of coordinates along three axes representing length, width, and height and expressed as x, y, and z. Cartesian coordinates are used by 3d printers to move through three dimensions while printing an object.

**Casting-** The process of pouring a liquid material (typically metal) into a hollow cavity to produce a solid part of a specific shape.

**Chamfer-** A 3d modeling term that describes the symmetrical, sloping surface at an edge or corner that is used to avoid violating the 45° rule.

**CNC Machining-** Computer numerically controlled machining – a subtractive method of manufacturing that involves a computerized machine removing material over a predetermined path to produce a final part.

---

**Cold End**– The part on an FDM-type 3d printer. The cold end grabs and pulls thermoplastic filament from the spool it is stored on and moves it into the hot end. A typical cold end consists of either a hobbed gear or a knurled wheel that is attached to a feeder motor. As the shaft of the motor spins, it rotates the hobbed gear or knurled wheel which grabs the filament and moves it toward the hot end.

**Cold Method**– See Atomic Method.

**Copolymer**- A type of plastic used in FDM printing. A copolymer is a material that is made up of several substances, each of which exists in long molecular chains. For example, ABS is a copolymer and consists of strands of acrylonitrile, butadiene, and styrene molecules all bound together.

**Cracking**– A 3d printing defect. Cracking occurs when one layer of print material bonds inadequately with another layer. When this happens, as the object cools, a split or crack occurs between the two inadequately bonded layers. See also Splitting.

**Creep**– The tendency for materials to move or deform over time when subjected to a continuous load. Resins and polymers often experience this phenomenon.

**Crystalline**- Any solid in which the atoms and molecules are organized in a lattice pattern. Metals are crystalline solids. The opposite of amorphous.

**Cupping**- This occurs in the SLA process when a hollow section of a print sucks up resin during the peeling process (similar to an upside-down empty cup entering the water). This suction effect can cause a part with thin walls to fracture.

**Curing**- The process of hardening a 3d printing material to its final form. Commonly used term in SLA printing where light is used to harden liquid photopolymer resin. See also Hardening and SLA.

## D

**Desiccant**- A hygroscopic substance used as a drying agent. Desiccants are often employed in FDM printing where many printing materials are hygroscopic. See also hydrolysis.

**Desktop 3D Printer**– The main characteristic of a desktop 3D printer is its build volume. Build volume is an industrial term that refers to the size of a part that a 3D printer can produce. Desktop 3D printers generally have a smaller build volume and produce smaller 3D printed parts. Due to its smaller build volume, a desktop 3D printer is generally more affordable.

---

**Direct Drive Extruder**– A method of conveying thermoplastic filament used by some FDM-type 3d printers. On a printer with a direct drive extruder, the cold end is placed on top of the hot end. See also Cold End and Hot End.

**DLP**– Short for Digital Light Processing. A form of 3d printing where a light source is used to cure photopolymer resin to produce a printed object.

**Dual Extrusion**– A FDM-type 3d printer with two extruders. Each extruder can print with a different filament material. Useful for building soluble support structures and producing multicolored objects.

**Ductility**- A material is said to be ductile if it can be deformed without losing toughness. A wire is an example of a ductile material. The opposite of brittle.

## E

**Elongation**– Pulling or stretching a material. An important term in plastics to understand how a material will deform under load

**Enclosure**- A part on a 3d printer that protects the user from moving parts and high-temperature objects. Is also used to increase or stabilize the ambient air temperature around the print to stop warping or cracking of the print, caused by cooling too fast.

**End Part**- A component that is intended to be used directly in a functional capacity.

**End Stops**- A part on a 3d printer. End stops are switches mounted on each of a printer's axes. The switch is tripped when a particular axis moves to its end. End stops enable a 3d printer to find its starting point when beginning to print.

**Extrude**- The process of forcing out a thin layer of melted thermoplastic onto a build surface to build up a printed object.

**Extruder**– See Bowden Extruder and Direct Drive Extruder.

**Extruder Motor**- A motor in the cold end that uses a hobbed gear or knurled wheel to move thermoplastic filament from a storage spool to the hot end for extrusion.

## F

**Fan**- See Heat Sink Fan and Layer Cooling Fan.

---

**FDM-** Short for Fused Deposition Modeling. A 3d printing process where melted thermoplastic is deposited in successive layers to produce a finished object using a digital model.

**Feeder**– See Cold End.

**FFF-** Short for Fused Filament Fabrication. An alternative name for FDM.

**FFM**– Short for Fused Filament Manufacturing. An alternative name for FDM.

**Filament**– The printing material used by FDM-type 3d printers. The filament is usually a thermoplastic that is fed by a cold end to the hot end as a solid. In the hot end, it is heated to printing temperature and extruded out through the print nozzle. Filament comes in different diameters and usually sold in spools. There is a wide variety of filament materials available, as well as a wide variety of quality. In general, a high-quality filament will produce better results than a less expensive filament that may be of poorer quality.

**Filament Drive Gear-** A part on an FDM-type 3d printer. The filament drive gear grabs the printing filament and moves it off of the storage spool and to the hot end of the printer for extrusion.

**Fill Density**– A slicer program setting that is a measure of how much material will be printed inside the outer shell of the object in question. Infill density is used to conserve filament while printing and speed up printing times. More information on slicer program settings can be found [here](#). See also Slicer.

**Fixture-** A frame used to hold components or parts in a fixed position used in the assembly or manufacturing process. It can also be known as “jigs and fixtures”. To learn more about jigs and fixtures [click here](#)

**Flexural Strength-** The stress (in MPa) at failure in bending.

**Frame-** A part of a 3d printer. The frame is the chassis or outer case of a 3d printer. The frame is usually made of acrylic plastic, aluminum, or stainless steel. A solid frame reduces printer vibration which increases printer accuracy and results in more precise end objects.

## **G**

**G-Code**– A programming language that controls the actions of a 3d printer – things like motion, speed, rotation, and depth. Commonly, this code is generated by a slicer program. See also Slicer.

---

**Glass Transition Temperature-** The temperature region where a material transition from hard, glassy material to a soft, rubbery material.

**Glue Stick**– Used in FDM printing on the bed of a printer to improve adhesion. More information on the use of a glue stick can be found [here](#).

## H

**Hairspray-** Used in FDM printing on the bed of a printer to improve adhesion. Not recommended due to mess and inconsistencies.

**Hardening**– See Curing.

**Heat Creep-** Heat creep is a problem that occurs in FDM-type 3d printers when higher temperatures extend back and upwards from the hot end. This causes the “melt area” to extend farther back as well, softening and melting the print material well before the nozzle end of the extruder. The softened thermoplastic increases the amount of pressure needed for extrusion. Eventually, the extruder motor can’t keep up and the nozzle gets clogged.

**Heat Sink Fan-** A part of an FDM-type 3d printer. A heat sink fan helps to dissipate the heat from the heat sink in the hot end.

**Heated Build Chamber-** A part of an FDM-type 3d printer. An enclosed compartment around the build plate that eliminates drafts and temperature variations to reduce or prevent material warping.

**Heated Print Bed**– A part on an FDM-type 3d printer. A heated print bed keeps the build surface warm, promoting greater adhesion and decreasing incidents of warping.

**Heated Build Chamber-** A part of an FDM-type 3d printer. An enclosed compartment around the build plate that eliminates drafts and temperature variations to reduce or prevent material warping.

**Heated Print Bed-** A part on an FDM-type 3d printer. A heated print bed keeps the build surface warm, promoting greater adhesion and decreasing incidents of warping.

**HIPS-** Short for high impact polystyrene. High impact polystyrene is a 3d printing filament that is strong, durable, non-toxic, and recyclable. It combines the hardness of polystyrene with the elasticity of rubber to produce a high impact thermoplastic that is tough and strong without being brittle. In 3D printing, HIPS makes an excellent soluble support material. HIPS is soluble in Limonene, an easily obtainable solvent that is derived from the skin of lemons.

---

**Hobbed Gear-** See Filament Drive Gear.

**Hollow-** A 3D print that is not solid and also does not contain any infill. Hollow models are much faster and cheaper to print but have very low strength.

**Hot End-** A part on an FDM-type 3d printer. The hot end heats the thermoplastic printing filament to the melting temperature and extrudes the melted material onto the build surface. A typical hot end consists of a heating block that produces the heat necessary to melt the print filament, a thermistor that controls the temperature of the heating block, and a print nozzle through which the melted filament is extruded. A heat sink is also typically used to radiate excess heat away from the print end.

**Hydrolysis-** The chemical breakdown of a hygroscopic material due to exposure to water.

**Hygroscopic-** The ability of a material to absorb water. Many thermoplastic printing materials exhibit a hygroscopic tendency to one extent or another and need to be insulated from exposure to atmospheric moisture.

## I

**ideaMaker-** A 3D slicing software from Raise3D that can prepare STL, OBJ, and 3MF files within 2 clicks. ideaMaker has defined settings that provide limitless customizations for all users.

**Industrial 3D Printer-** An industrial 3D printer is designed to be a high-quality machine with mechanical as well as hardware and software features in place to ensure a high success rate and high print quality.

**Infill-** See Fill Density.

**Injection Molding-** The process of injecting plastic in a melted liquid form into a die. The plastic fills the empty cavities of the die and cools until it has solidified. The solid plastic part is then ejected from the die and the process is repeated.

**Islands-** Occur in SLA printing and refer to cross-sectional areas of a model that are not connected.

**Isotropic** – A material that has the same physical properties in all directions. Glass and metal are common examples of isotropic materials. The opposite of anisotropic.

---

## J

**Jig-** A frame used to hold components or parts in a fixed position used in the assembly or manufacturing process. It can also be known as “jigs and fixtures”. To learn more about how to create 3D printed jigs and fixtures, [click here](#)

## K

**Kapton Tape-** Used in FDM printing on the bed of a printer to improve adhesion. More information on the use of Kapton Tape can be found [here](#).

**Knurled Wheel-** See Filament Drive Gear.

## L

**Layer-** In 3d printing, a layer is any one of the individual thin sections of print material that make up a printed object. Before printing, a slicer program takes the STL file generated by the CAD software and slices the digital object into multiple horizontal sections or layers.

**Layer Cooling Fan-** A part of an FDM-type 3d printer. A layer cooling fan cools off the printing material as soon as it is deposited on the build surface.

**Layer Height-** A slicer program setting. Layer height is the setting that establishes the height of each layer of filament in your print. In some sense, layer height in 3d printing is akin to resolution in photography or videography. When you choose a thicker layer height, your object will have less fine detail and the layers will be more visible. When you choose a thinner layer height, a higher level of detail is possible, and your layers will tend to blend into one another. However, keep in mind that the thinner you make the layer height the more time it will take to print the object in question since there will be more layers to print. See also Slicer.

## M

**Melting Point-** The temperature a solid melt or turns into a liquid.

**MEM-** Short for Melted and Extruded Modeling. Another name for FDM printing.

**Metal Powder-** The material used for metal printing.

**Metal Printing-** The process of 3D printing in metal. Objects are created from thin layers of powdered material by selectively sintering or melting it using a high-power laser. There is a large range of metal printing technologies.

---

**Micron**– A measurement of distance regularly used to describe 3D printing layer height. 1000th of a millimeter. A human hair is approximately 17 microns thick.

**Motherboard**- A part on an FDM-type 3d printer. The motherboard is the brain of an FDM-type 3d printer. It takes the commands given by the GCode and turns them into physical movements. The motherboard contains all of the circuitry needed to operate the printer's motors and sensors.

## N

**Nozzle**- The part of a 3D printer where the build material is extruded from.

**Nozzle Diameter**– The diameter of the material that is extruded out of the nozzle. This plays an important role in FDM where shells and walls should be a multiple of nozzle diameter.

**Nylon**- Nylon is a thermoplastic printing filament used in FDM-type 3d printers. It offers excellent strength and durability while, at the same time, it is exceedingly versatile. It can be printed very thin to allow for flexibility and not lose its strength and ability to stand up to wear and tear. It also has a low friction coefficient with a correspondingly high melting temperature. This makes it an excellent choice for prototypes and moving parts of all kinds. Nylon has a printing temperature of 255C to 275C.

**Nylon Powder**- A common build material used in the SLS printing process.

## O

**OBJ**– Short for Object File. A 3d file format used by CAD programs as an alternative to STL files when information about color or material is important.

**Offset**- In 3D printing offset refers to layers that are not printed directly in line with one another and are instead shifted to the side. This is often a printer calibration issue and will impact the quality of a print.

**Overhang**- Any part of a 3d model that lacks support below it. Parts that protrude at angles greater than 45° are generally considered overhangs. See also Support Materials and Support Structures.

## P

**Painters Tape**- See Blue Painter's Tape.



---

**PC**– Short for Polycarbonate. See Polycarbonate.

**PEI**– Short for polyetherimide. Used in FDM printing on the bed of a printer to improve adhesion. More information on the use of PEI can be found [here](#).

**Perimeter**- A slicer program setting. Perimeter refers to the thickness of the walls or shell of a printed object. The greater the number of perimeters, the thicker the shell of the object will be.

**PETG**- Short for Polyethylene Terephthalateglycol. PETG is a thermoplastic printing filament used in FDM-type 3d printers. An object printed with PETG will be very strong but, at the same time, it will have a bit of flex to it. You may be able to bend it, but it will be very hard to break it. PETG is transparent and has a printing temperature of around 220C-235C. It has no odor when printing and produces a result that has a marvelous finish. Also, PETG is a great material to print with because it shrinks very little when cooling, so objects printed with PETG will experience very little warping.

**Photopolymer**- A material used in 3d printing that hardens when exposed to certain types of light. Photopolymers are used in Digital Light Processing (DLP) and Stereolithography (SLA).

**Pillowing**– A 3d printing defect. Pillowing occurs on the top surface of an object. It looks like there are gaps in the surface layer, along with little bumps or pillows. In general, pillowing is caused by a top layer that is too thin and/or improper cooling of that layer. Under certain circumstances, insufficient infill can also contribute to the problem.

**PLA**- Short for Polylactic Acid. PLA, or Polylactic Acid, is a biodegradable, environmentally friendly thermoplastic that is manufactured out of natural substances, usually corn or sugarcane. You've probably already encountered PLA in your home since it is used to make everything from garbage bags to disposable cutlery and plates. PLA prints at relatively lower temperatures than other printing materials (180C – 210C). Even though it is biodegradable, it remains a strong and durable material, albeit brittle, capable of being used in a wide variety of projects. PLA is available in a wide variety of colors and is not readily soluble.

**PMMA**– Short for Polymethyl Methacrylate. PMMA is a thermoplastic printing filament used in FDM-type 3d printers. PMMA is known commercially as acrylic and is marketed under various brand names, such as Plexiglas, Lucite, and Perspex. Widely used as an alternative to glass in applications where more strength and durability are needed, PMMA has significantly higher impact strength than glass.

**Polycarbonate**- Polycarbonate is a thermoplastic printing filament used in FDM-type 3d printers. It is an extremely strong, lightweight, and transparent thermoplastic. Marketed under the

---

trade name Lexan, it is used to make products as varied as CDs and DVDs, bulletproof glass, riot gear, sunglass lenses, scuba masks, electronic display screens, phone and computer cases, and much more.

**Polyjet-** Similar to inkjet printing, but instead of jetting drops of ink onto paper, jets droplets of liquid photopolymer (in layers) onto a build tray and cures them instantly using UV light. The results are fully cured objects that can be handled and used immediately.

**Polymer-** A type of plastic used in FDM printing. A polymer is a material that is made up of multiple long molecular chains of a single substance. For example, PVC or polyvinyl chloride consists of a bunch of vinyl chloride molecules.

**Post Processing-** Any act of improving the appearance or material properties of a 3d print after it has been printed. This covers a large range of processes in 3D printing that vary by technology (support removal, UV curing, heat treating, sanding, tumbling, polishing, painting, etc).

**Power Supply-** A part on an FDM-type 3d printer. The power supply takes the 240V AC electricity from the wall and converts it to low voltage DC power for your printer to use.

**Print Bed-** See Bed.

**Print Head-** A part on an FDM-type 3d printer. The print nozzle is attached to the bottom of the hot end and is where the melted thermoplastic printing material is extruded. In general, a smaller diameter nozzle will produce finer details in the finished object, albeit at a slower print speed and a greater risk of clogging.

**Print Resolution-** An indication of printing quality. Horizontal resolution refers to the movements made by the print head along the x and y axes. The smaller the movements, the higher level of printing detail the printer produces.

**Print Speed-** A slicer program setting. Print speed is how fast the print head travels while extruding filament. Therefore, optimal speed depends on the object you are printing and the filament material that you are using to fabricate the object. In general, simple objects with less detail can be printed faster without complication.

**Printing Temperature-** The optimal temperature for a thermoplastic printing material to be at for effective extrusion. The printing temperature differs from material to material.

**Printing Volume-** See Build Volume.

---

**Prototype-** An early part or model of a design built before production to test form, function, aesthetics, and interaction usually at a low cost. Prototypes are typically items to learn from to improve a design.

**PVA-** a water-soluble filament which works pretty well with PLA.

To learn more about filaments [click here](#)

To learn more about how to 3D print using PVA [click here](#)

## Q

## R

**Raft-** A raft is a layer or layers of extruded thermoplastic that is used to stabilize a printed object. A raft helps an object to adhere to the print bed. Unlike a brim, a raft is connected to the perimeter and bottom of an object.

**Rapid Prototyping-** The process of creating physical prototypes directly from digital data.

**RepRap-** Short for Replication Rapid Prototyper. A project started in Britain in 2005 to produce a 3d printer capable of printing another 3d printer. Also the brand of the printers produced through the project.

**Resin-** A solid or highly viscous substance that is typically converted into a polymer. SLA uses resin exposed to UV light (a laser) to build a part layer by layer.

**Retraction-** A slicer program setting. This setting is used to pull the filament slightly back into the print head during times when the head is traveling from one print point on an object to another.

## S

**Shell-** The outer wall of a designed object.

**Shell Thickness-** A slicer program setting. Shell thickness refers to the number of layers that the outer wall will have before infill printing will begin. The higher the setting is for shell thickness, the thicker the outer walls of your object will be.

**Sintering-** The process of fusing particles to form a solid mass of material using heat or pressure without melting it.

---

**Skirt-** A line that is initially printed around the print (but not connected to the print) to clean the nozzle head.

**SLA-** Short for Stereolithography, 3d printing technology. SLA focuses a UV laser onto a tank of photopolymer resin. The light cures or hardens the top layer of the resin, building the object from the top down.

**Slice-** A horizontal layer of a digital object produced by a slicer program. Each slice contains coordinates for printing locations on the build surface, as well as for instructions as to layer height, shell thickness, and more.

**Slicer-** A slicer is a piece of 3d printing software that takes a digitized 3d model and converts it into printing instructions that your printer can then use to turn the model into a physical object. In essence, the slicer takes the CAD model and “cuts” it into layers.

**SLS-** Short for Selective Laser Sintering, 3d printing technology. SLS uses powdered polymer material to build a 3d object through the use of a laser. The laser sinters or binds the powder together one layer at a time from the top down.

**Soluble Materials-** Any thermoplastic printing material that is soluble, or dissolvable, when immersed in another substance. PVA and HIPS are both popular soluble printing materials.

**Solvent Method-** A method of unclogging the clogged print nozzle of an FDM-type 3d printer. More information on the solvent method can be seen [here](#).

**Splitting-** See Cracking.

**Stepping Motor-** Unlike regular DC motors, which rotate continuously when given power, stepper motors rotate in increments. This gives them precise control over their position. Most FDM-type 3d printers use NEMA 17 type motors with 200 increments (steps) per revolution.

**STL-** A 3d file format used by CAD programs.

**Strain-** Measure of the deformation of the material relative to its original shape measured in mm/mm (or a dimensionless ratio).

**Stress-** The internal forces that particles of a material exert on each other measured in Pascals.

---

**Stringing**- A 3d printing defect. Stringing is usually caused by the print nozzle oozing print material as it moves from one place to another. The oozed material cools and hardens into thin “strings” – hence the name.

**Subtractive Manufacturing**- The opposite of additive manufacturing. The process of creating a three-dimensional object from a 3d model by removing materials, usually one layer at a time. Machining is an example of subtractive manufacturing.

**Support Materials**- Printing materials used to support overhangs on a designed object. Support materials are usually soluble to facilitate easy removal after printing.

**Support Structures**- A layer or layers of extruded thermoplastic that is used to support overhangs on a designed object. Support structures are usually removed after printing is completed.

**Surface Finish**- In 3D printing, this refers to the roughness of the surface of a 3D printed part. Generally qualitative.

## T

**Tank (resin)**- The area where resin sits before being cured in the SLA process.

**Temperature Differential**- The difference in temperature between 2 points. In 3D printing reducing the temperature differential between 2 nearby points reduces the likelihood of warping or deformation.

**Tensile Strength (ultimate)**- The stress (usually in MPa) at which a material will fracture or break when subjected to a tensile load.

**Tensile Strength (yield)**- The stress (usually in MPa) at which a material will shift from elastic deformation (returning to its original shape) to plastic deformation (permanent deformation) when subjected to a tensile load.

**Thermistor**- Also known as a thermally sensitive resistor. A part on an FDM-type 3d printer. A thermistor is an element with an electrical resistance that changes in response to temperature. Used to regulate the temperature of the heat block in the hot end of a printer.

**Thermoplastic**– A substance, usually a plastic, that can melt and harden at precise temperatures.

**TPU**- Short for Thermoplastic Polyurethane. TPU is a thermoplastic printing filament used in FDM-type 3d printers. It is an extremely flexible and durable extrusion printing material. Its

---

flexibility and elasticity make it an excellent choice for belts, springs, and phone cases. TPU is also very resistant to abrasion, as well as grease, oil, and a wide variety of solvents. This makes it an excellent choice for industrial applications as well. TPU has a printing temperature of 210C to 230C.

## U

**Under Extrusion-** a problem experienced by FDM-type 3d printers. Under extrusion occurs when your printer is unable to supply the correct amount of material needed to correctly print a layer. You can learn more about under extrusion and how to correct it [here](#).

**User Interface-** part on some FDM-type 3d printers. Some FDM-type printers have an LCD screen so they can be controlled directly without hooking them up to a computer.

**UV Light-** For 3D printing, this refers to the type of light that is used to cure (harden) photopolymers in SLA and Polyjet 3D printing.

## V

## W

**Wall Thickness-** Generally associated with minimum wall thickness – the thinnest dimension a wall can be printed at such that it can support the model. Varies by technology. Different from shell thickness.

**Warping-** A 3d printing defect. Warping occurs when an object is cooling after printing. Cooling causes contraction and this contraction causes stress along the object's lateral surfaces. The quicker the cooling occurs, the greater the stress on the object.

**Water Method-** A method of applying Kapton Tape to the build plate of an FDM-type 3d printer to improve adhesion. You can find out more about the Water Method [here](#).

## X

**X-Axis-** A part of the Cartesian coordinate system used by FDM-type 3d printers to move through three dimensions while printing an object. The x-axis represents left to right horizontal movement.

---

## Y

**Y-Axis**– A part of the Cartesian coordinate system used by FDM-type 3d printers to move through three dimensions while printing an object. The y-axis represents front to back horizontal movement.

## Z

**Z-Axis**- A part of the Cartesian coordinate system used by FDM-type 3d printers to move through three dimensions while printing an object. The z-axis represents top to bottom vertical movement.