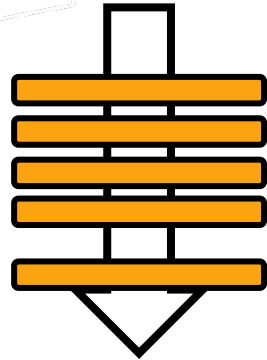


**SIU**

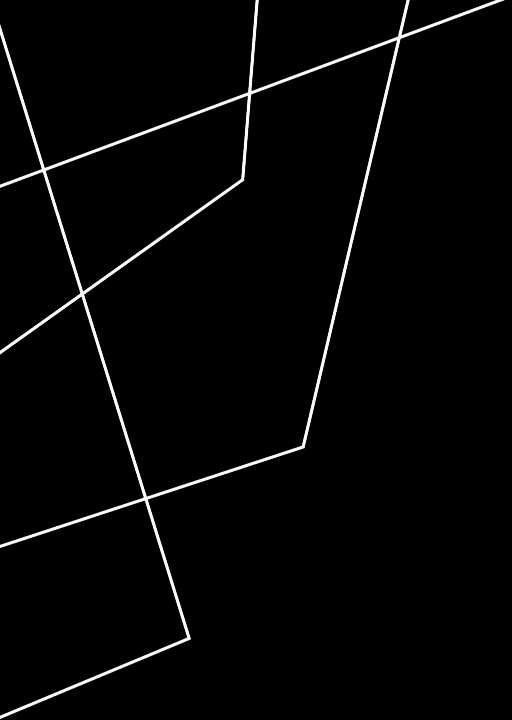


**BZI<sup>®</sup>**

**YOUTH  
ENGINEERING  
ACADEMY**



**FUNDAMENTALS  
OF 3D PRINTING**



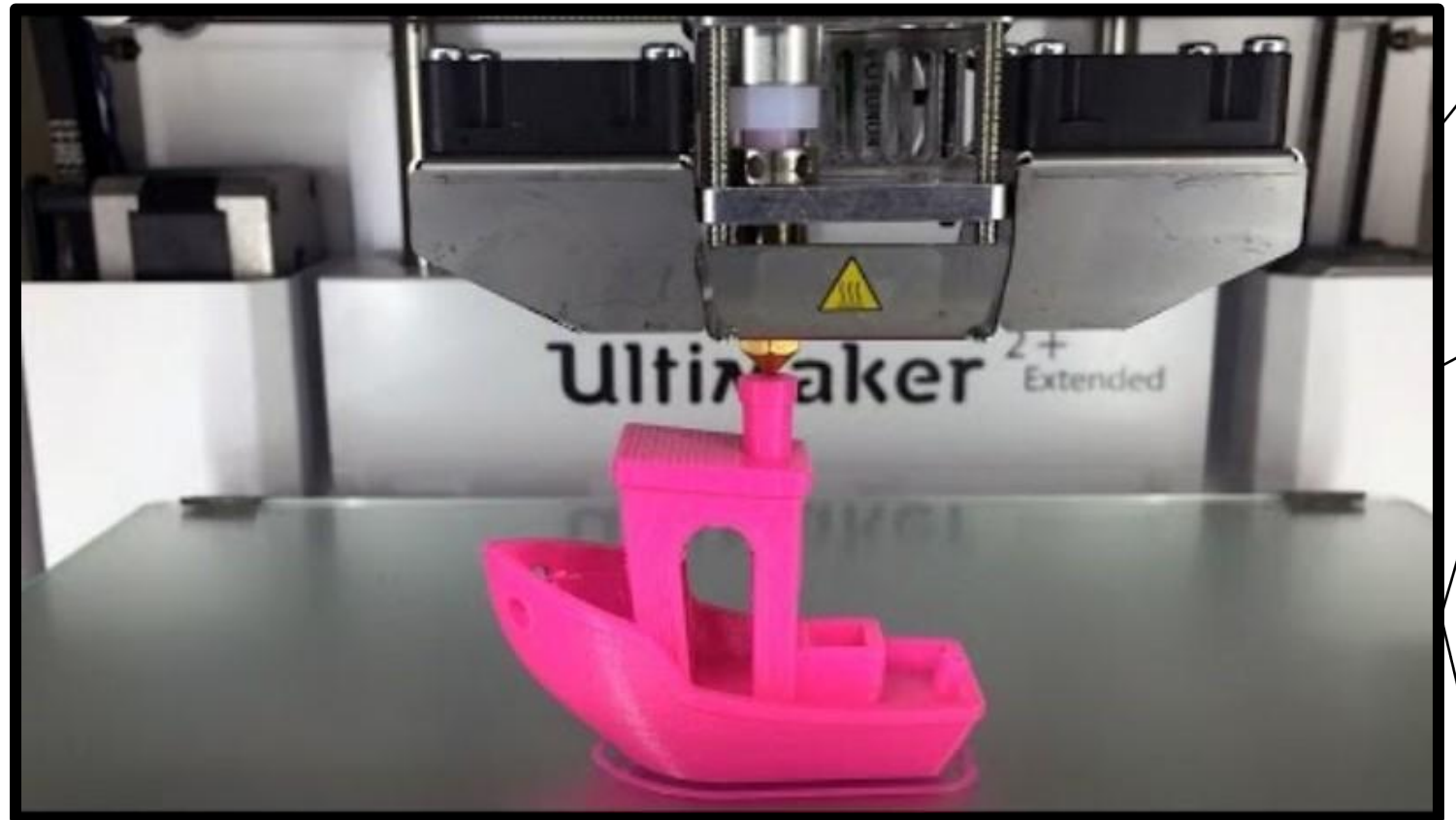
- What is FDM printing?
- Anatomy of a part
- Parts of a printer

# SESSION 1

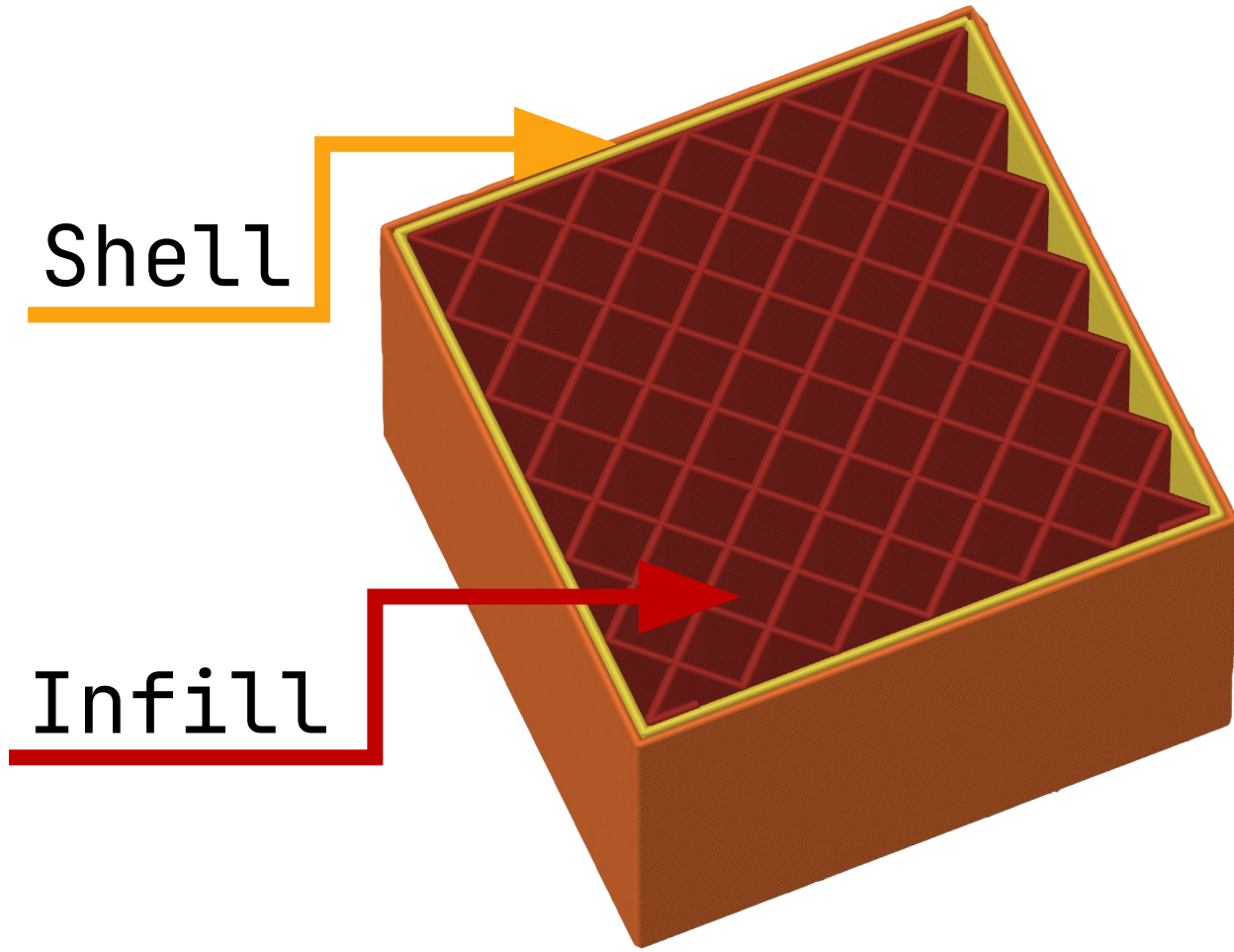
# PRINTING A BENCHY

In case you have never seen a 3D printer at work, here is a Benchy being printed.

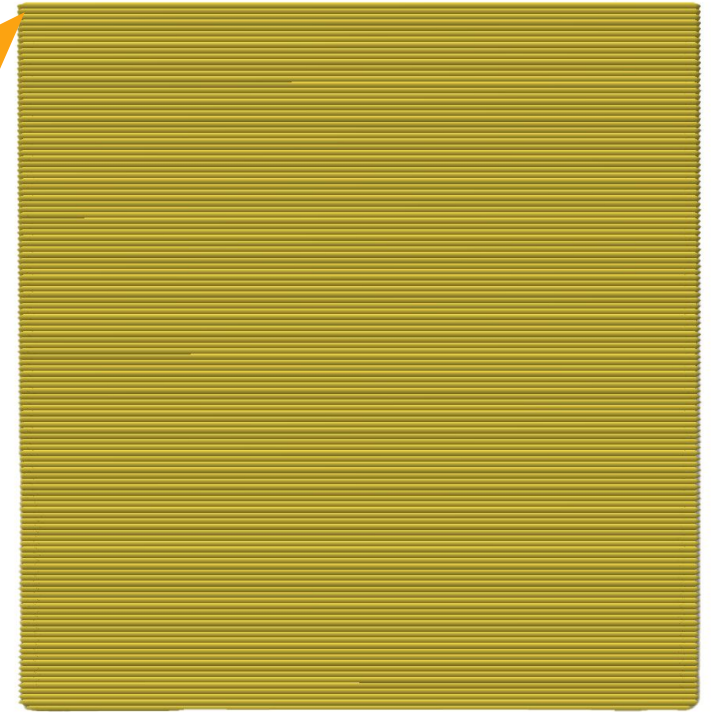
This is one of the most famous prints used to test the functionality of a 3D printer.



# ANATOMY OF A PART



Layers



# INFILL

Infill is adjusted by two primary settings.

Percent: changes how much of the infill area is filled

Pattern: changes what kind of pattern we want to use.

Grid is:

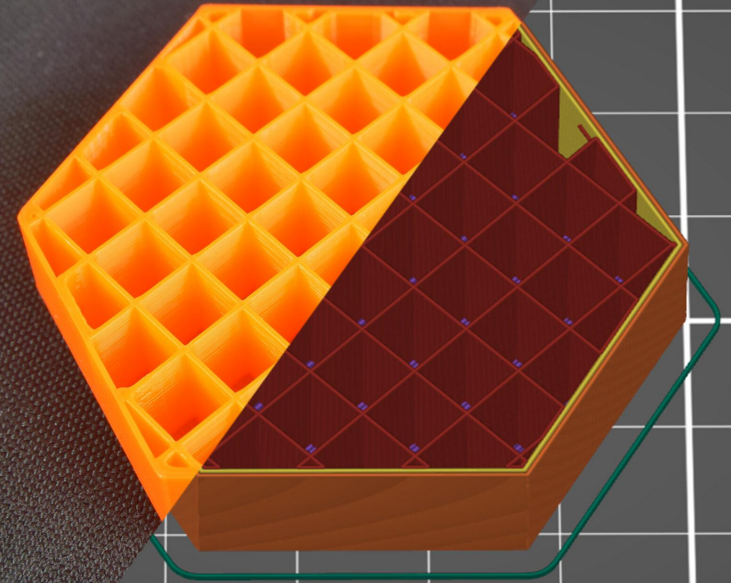
- very fast
- very strong
- risky as it overlaps infill and can cause failures.

Gyroid:

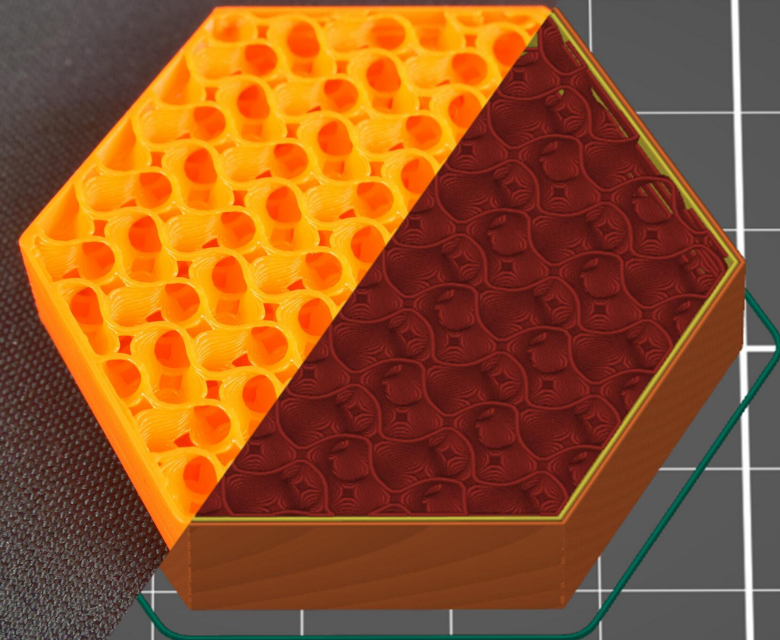
- fairly fast
- very strong
- uses material efficiently

Note: any infill over 30% has greatly diminishing returns on strength. Most prints should be done with 15-20% infill.

GRID infill



GYROID infill



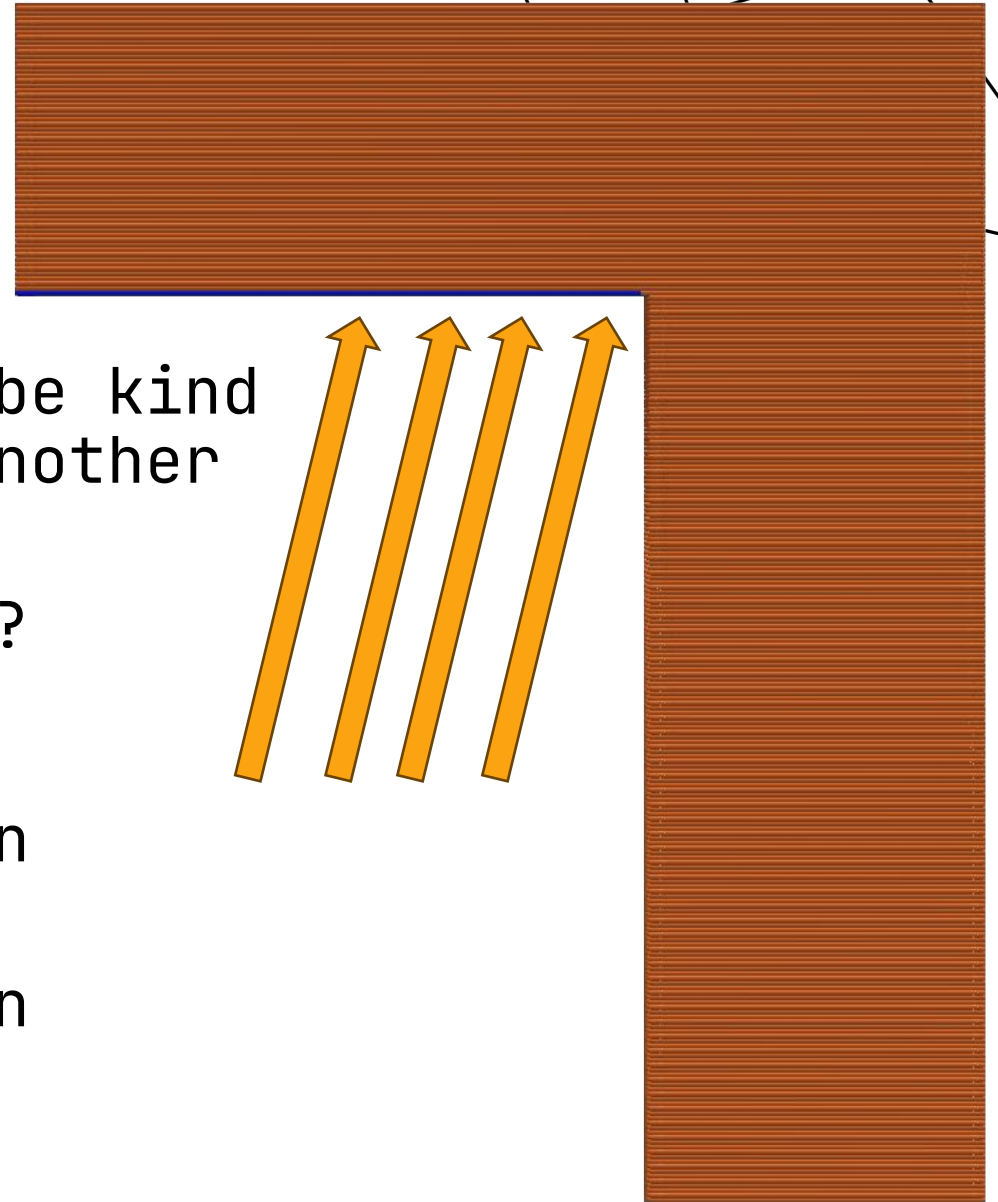
# A SMALL HICCUP

Now, just printing cubes would be kind of boring. So, let's throw up another theoretical shape.

Can we see any problems with it?

Unfortunately, we can't print in midair.

A feature like this is called an **'overhang'**

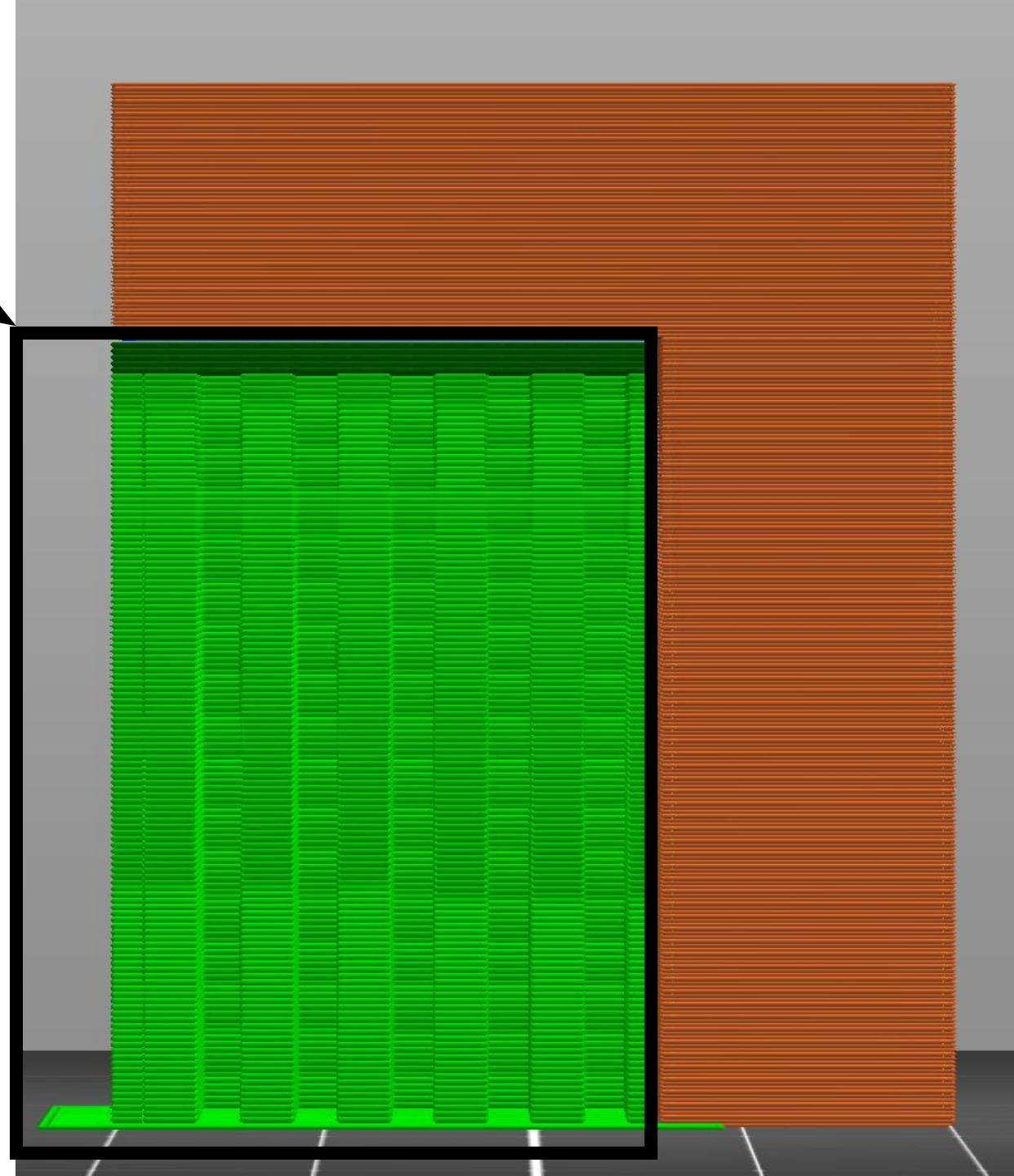
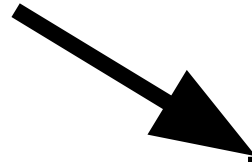


# SUPPORT

To solve this problem, we print 'support' to provide a surface we can print onto.

This is done in the slicer.

Support is meant to be broken off after the print finishes.



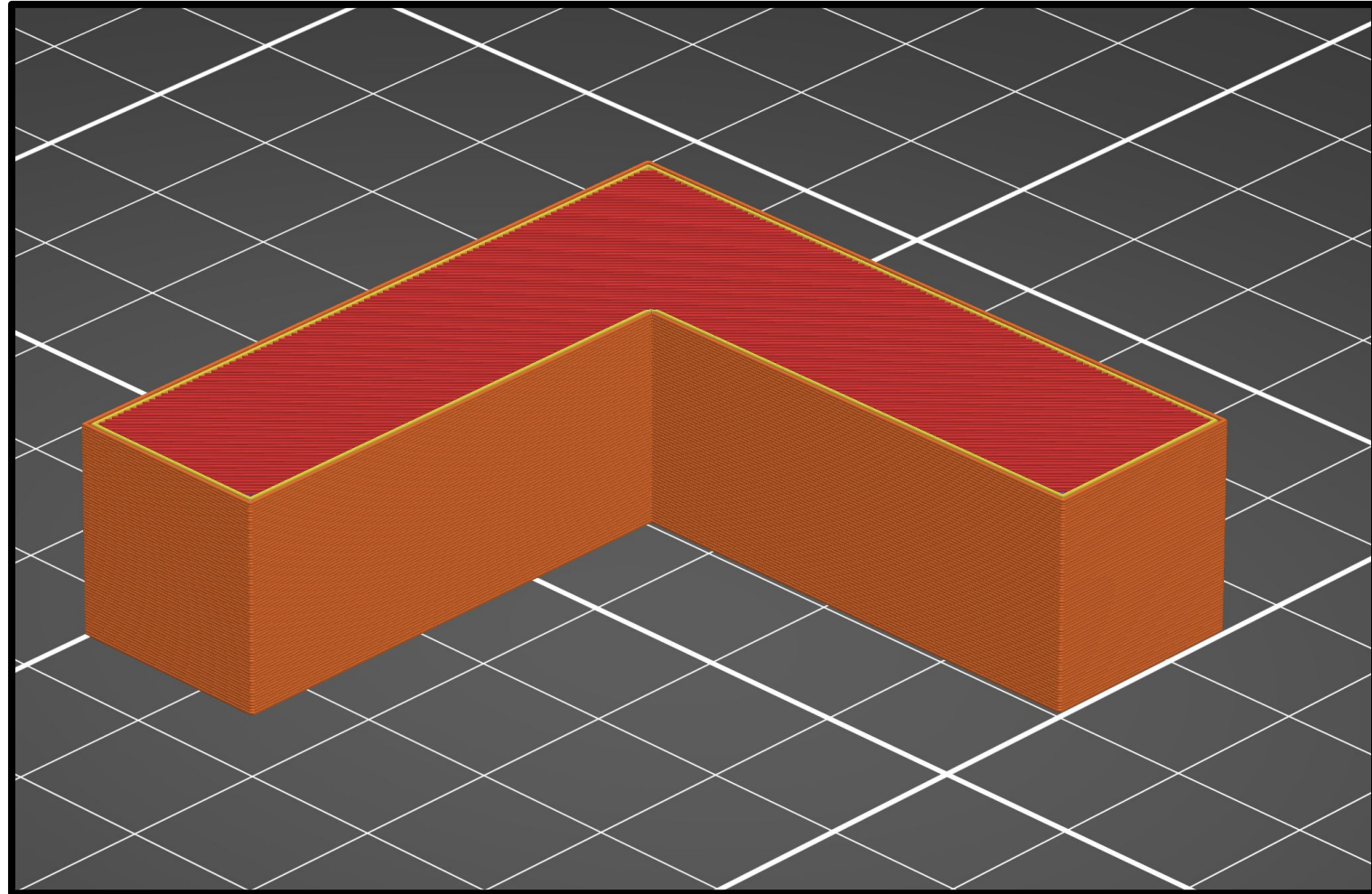
# VIDEO ON SUPPORT STRUCTURES



# ALTERNATIVELY

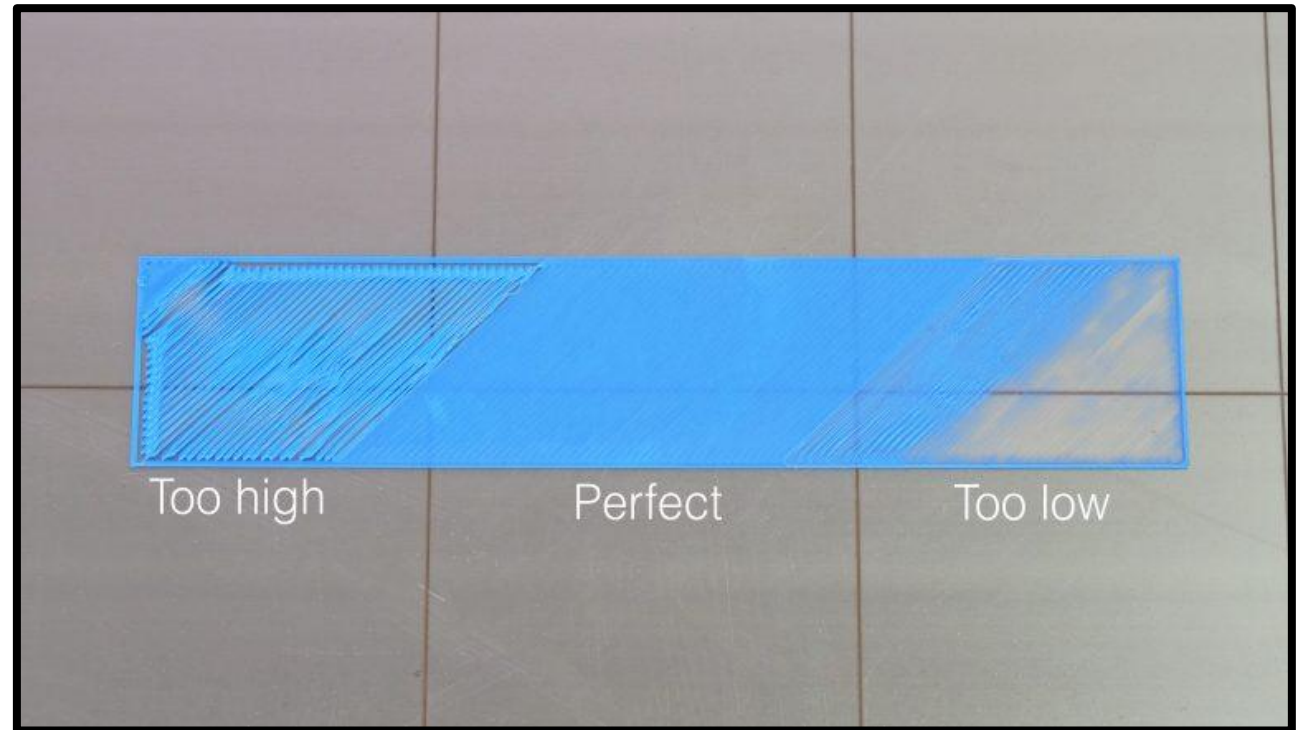
However, we can save a lot of plastic and time by just laying the part on the side.

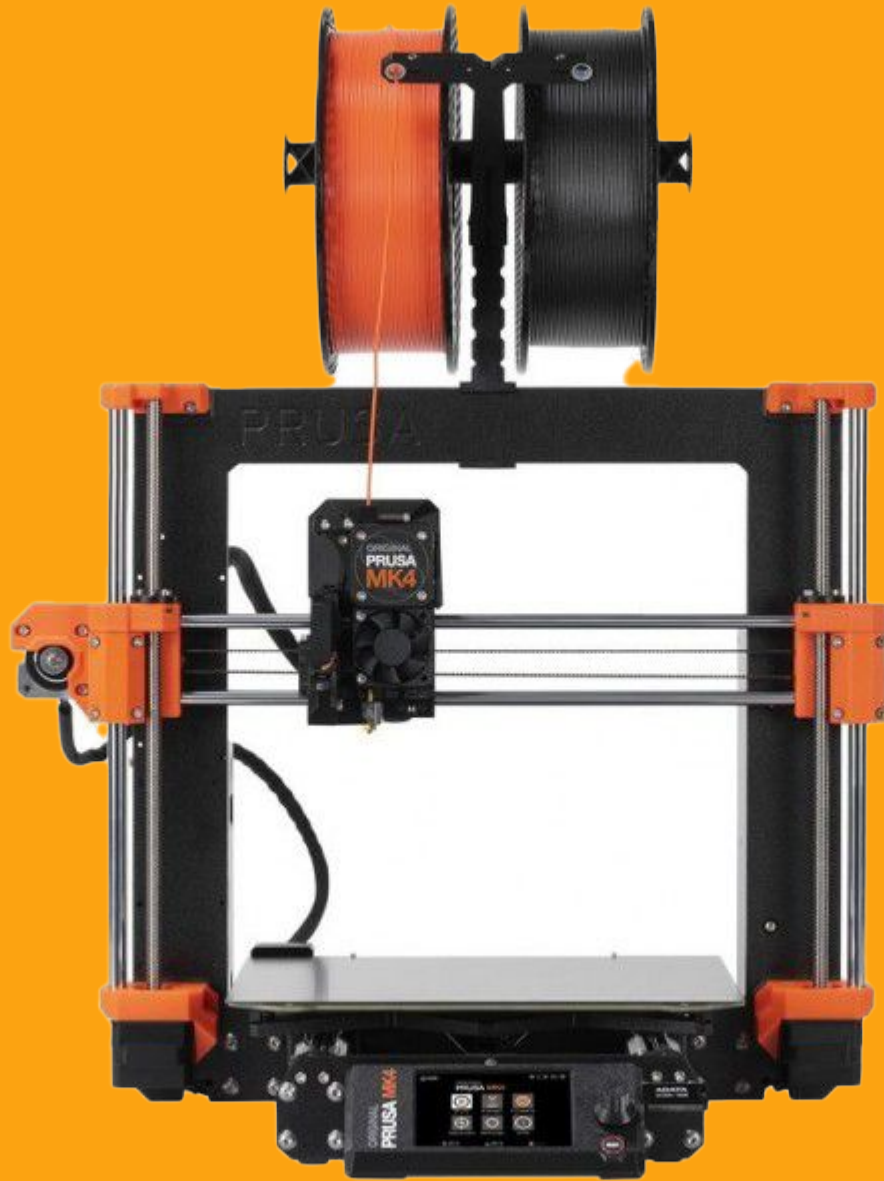
Always try to reduce overhangs whenever possible.



# FIRST LAYER

It's critical, both for appearances and adhesion that our first layer be the perfect height.





# PRINTER

This is the machine that lays down the plastic. Let's go over the most important components of the printer.

# PRINTER FORMATS



DELTA



CORE-XY



CORE-XZ

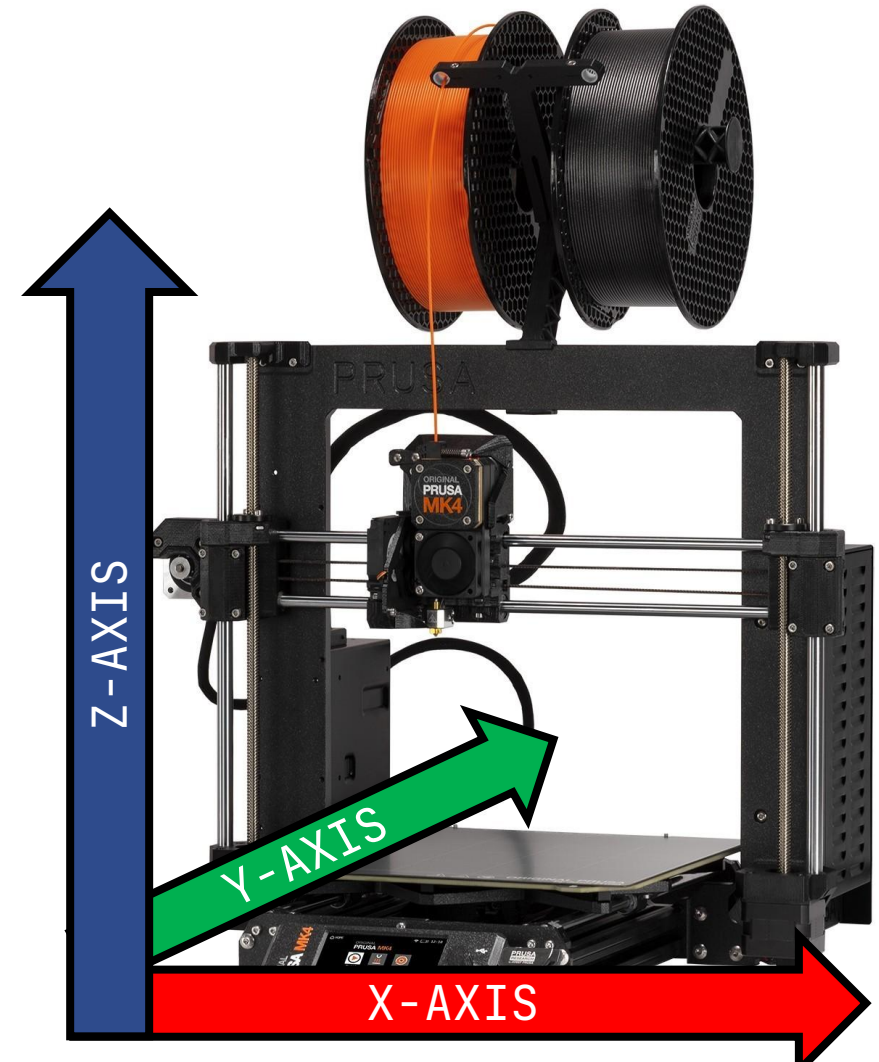
We will be discussing CORE-XZ printers this week.  
However, most things apply to all formats.

## PRINTER AXES

A 3D printer is called that because it prints along **3 Axes**. Just like you can plot points in a 3d graph, we do the same with printers.

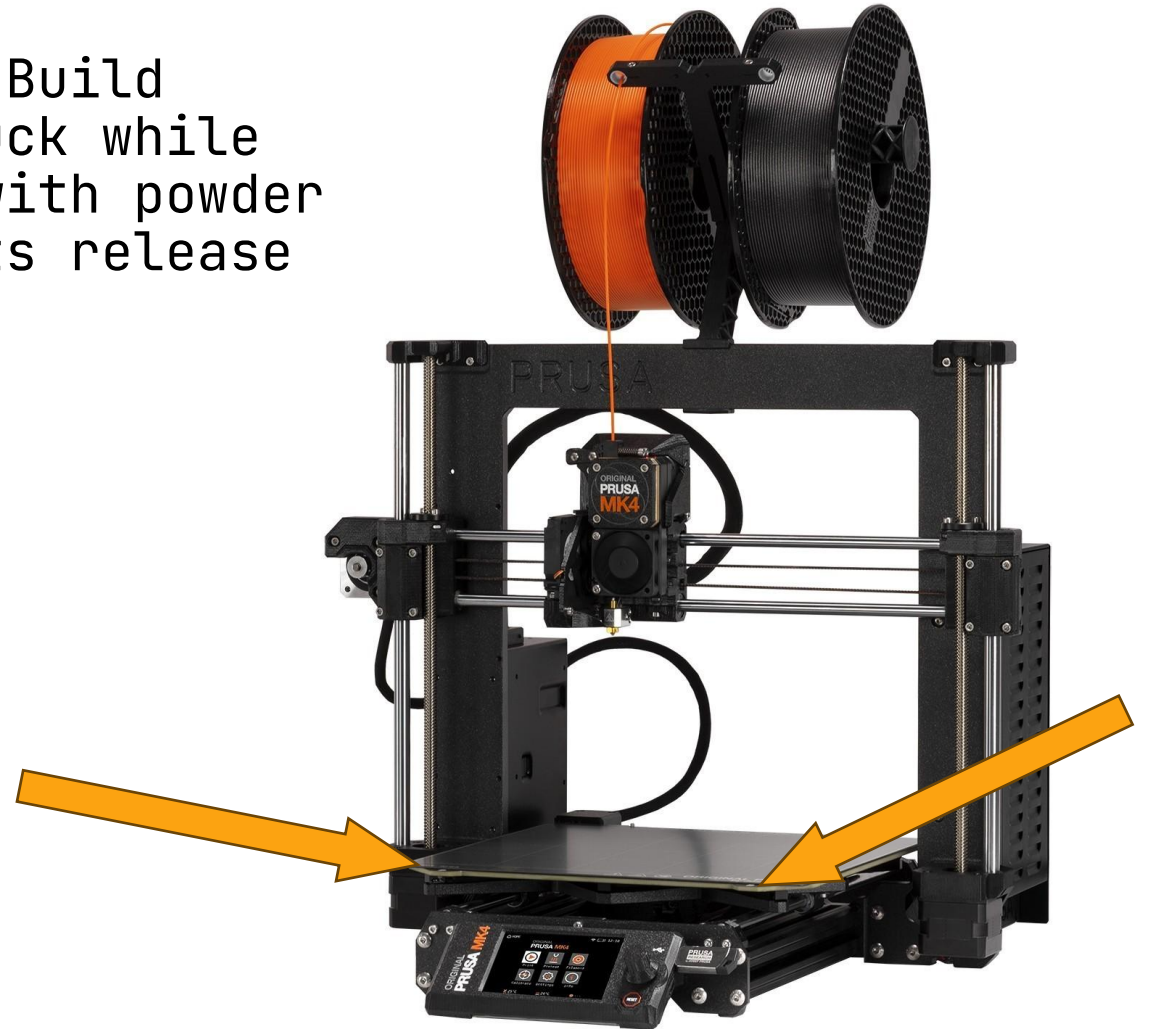
So when we talk about the z-axis, we mean the part of the printer that moves up and down.

We control the printer by feeding it coordinates that we want it to go to.



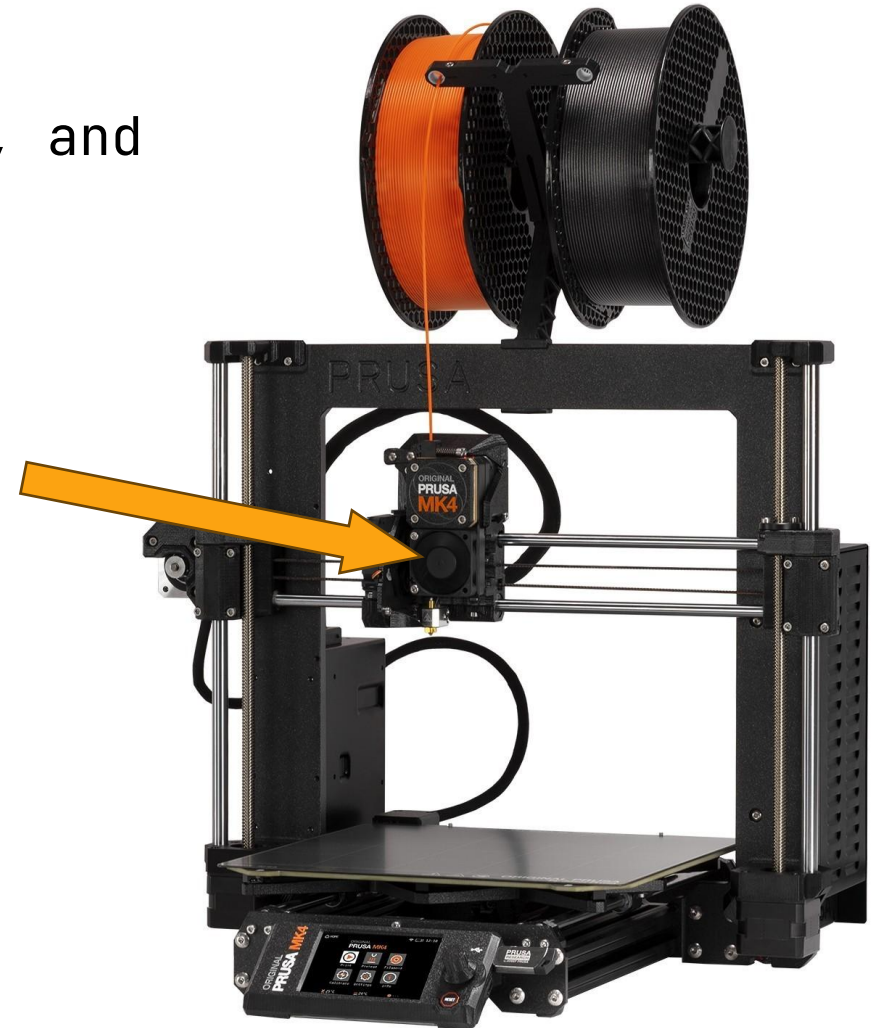
## PRINT BED/BUILD PLATE

The surface we print our parts on. Build plates are heated to keep parts stuck while printing. They are usually topped with powder coating or plastics that help prints release after the print finishes.



## HEAD

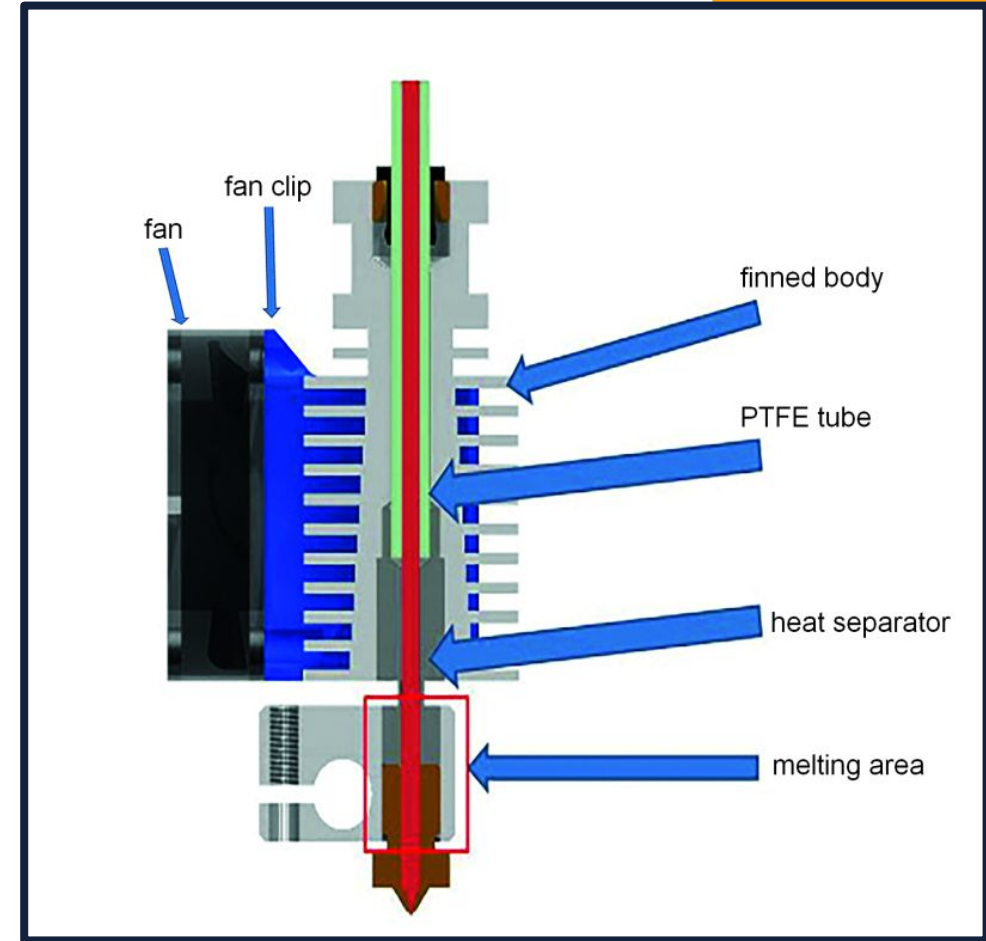
This is the most important part of the printer. It grabs the filament, heats it, and pushes it through the nozzle.



## HEAD (CONTINUED)

There are a lot of parts in the printer's head.

One notable thing is the separation between the hot and "cold" ends. If the filament gets too hot near where we grab it with the extrusion gears, it will get soft and possibly jam.



## HEAD (CONTINUED)

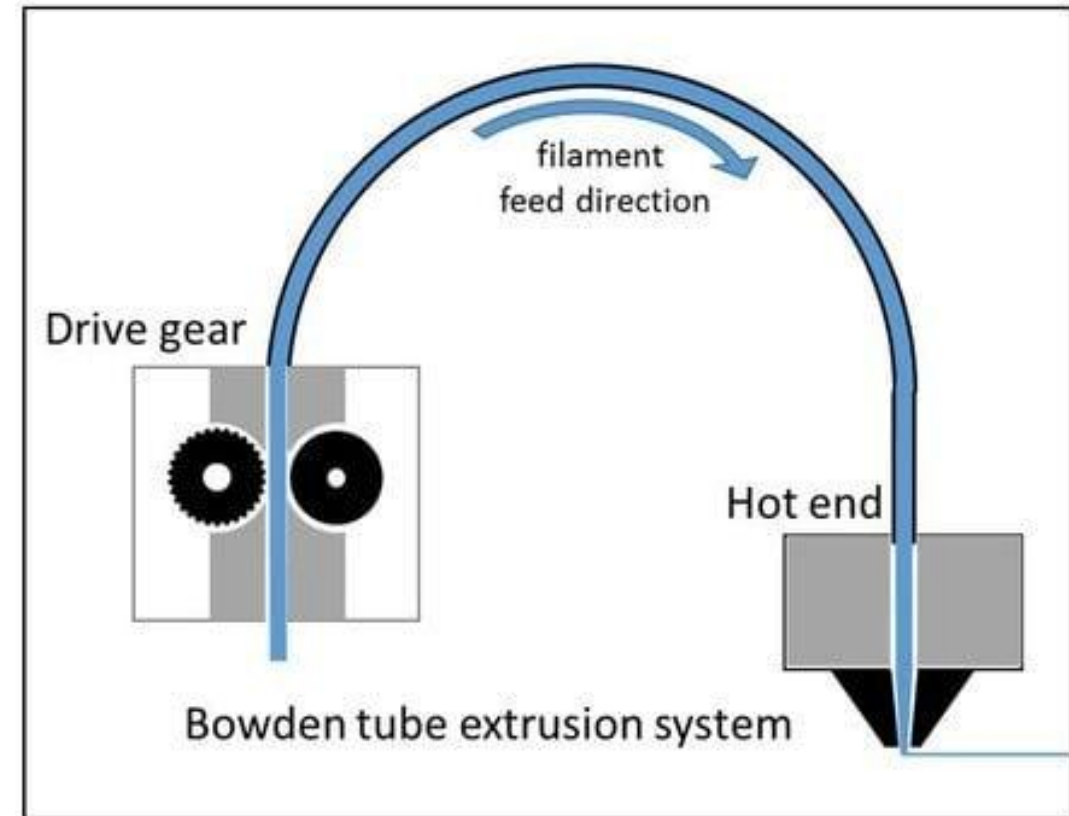
A note about drive types:

Bowden tube extrusion used to be more common. It moves the drive motor away from the head, therefore decreasing its overall weight. However, this invites problems with filament jamming and poor retraction performance due to the slack in the filament.

They 'push' the filament.

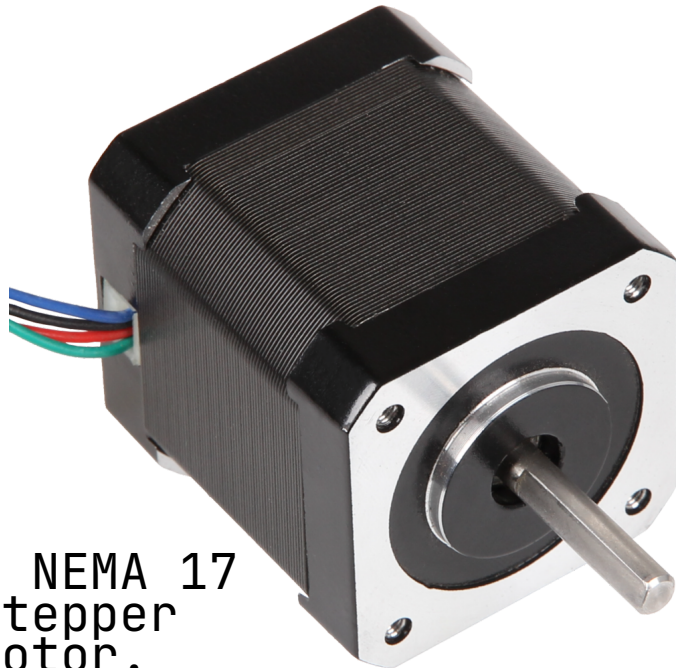
---

Premium and modern printers use direct drive (as shown on the previous slide). They pull the filament.



## MOTORS

There are motors all over a 3d printer. They control the belts that move the print head and the build plate. They also move the gears that grab the filament.

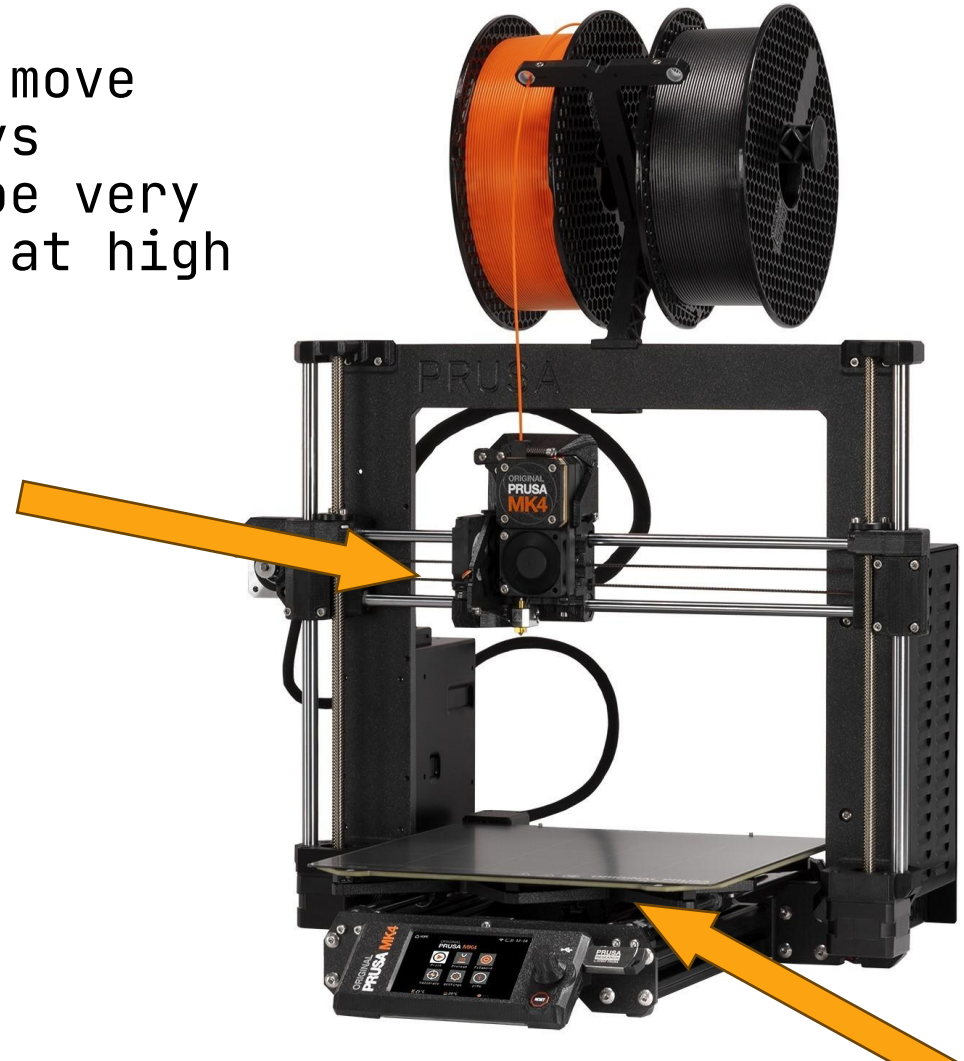


A NEMA 17  
stepper  
motor.



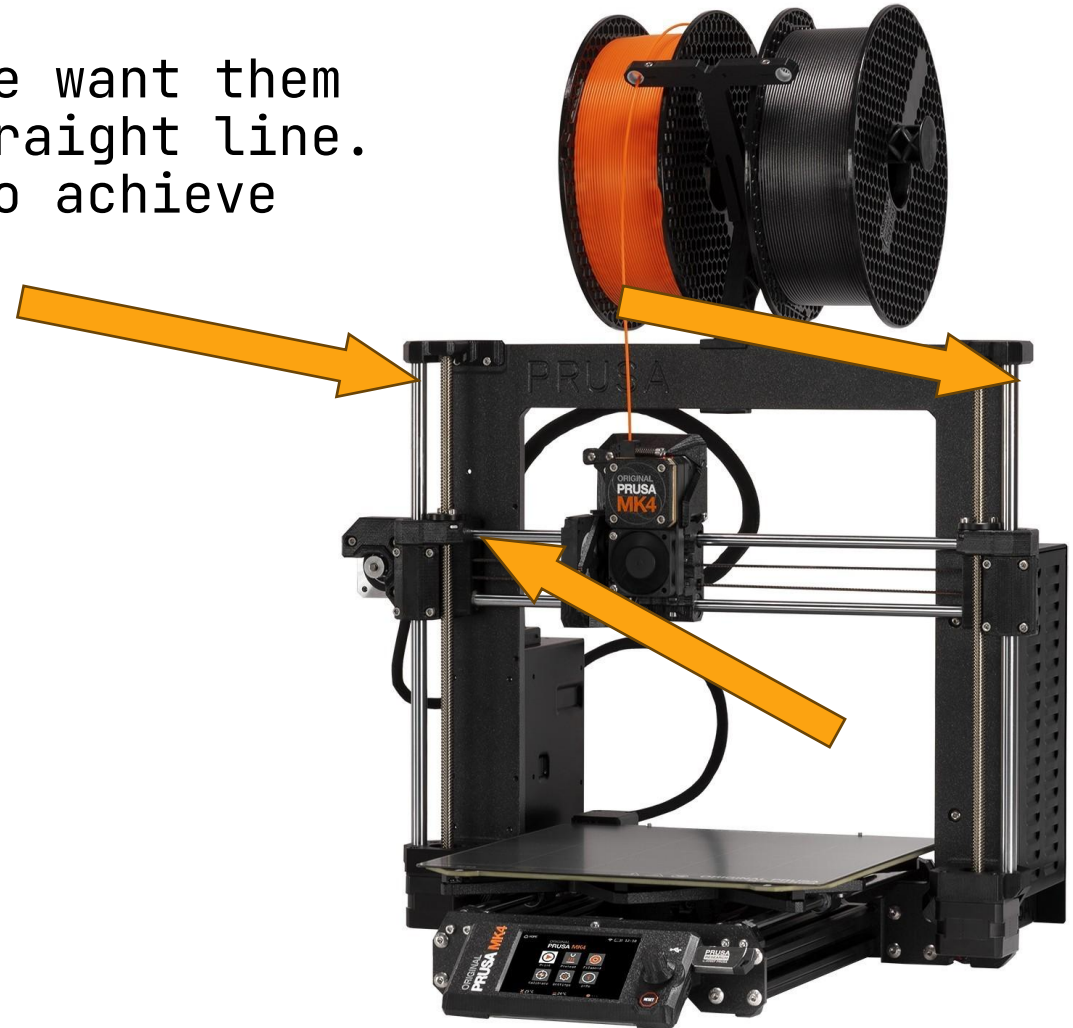
## BELTS AND PULLEYS

For parts of the printer that need to move rapidly, we often use belts and pulleys attached to the motors. They tend to be very lightweight and resistant to slipping at high speeds.



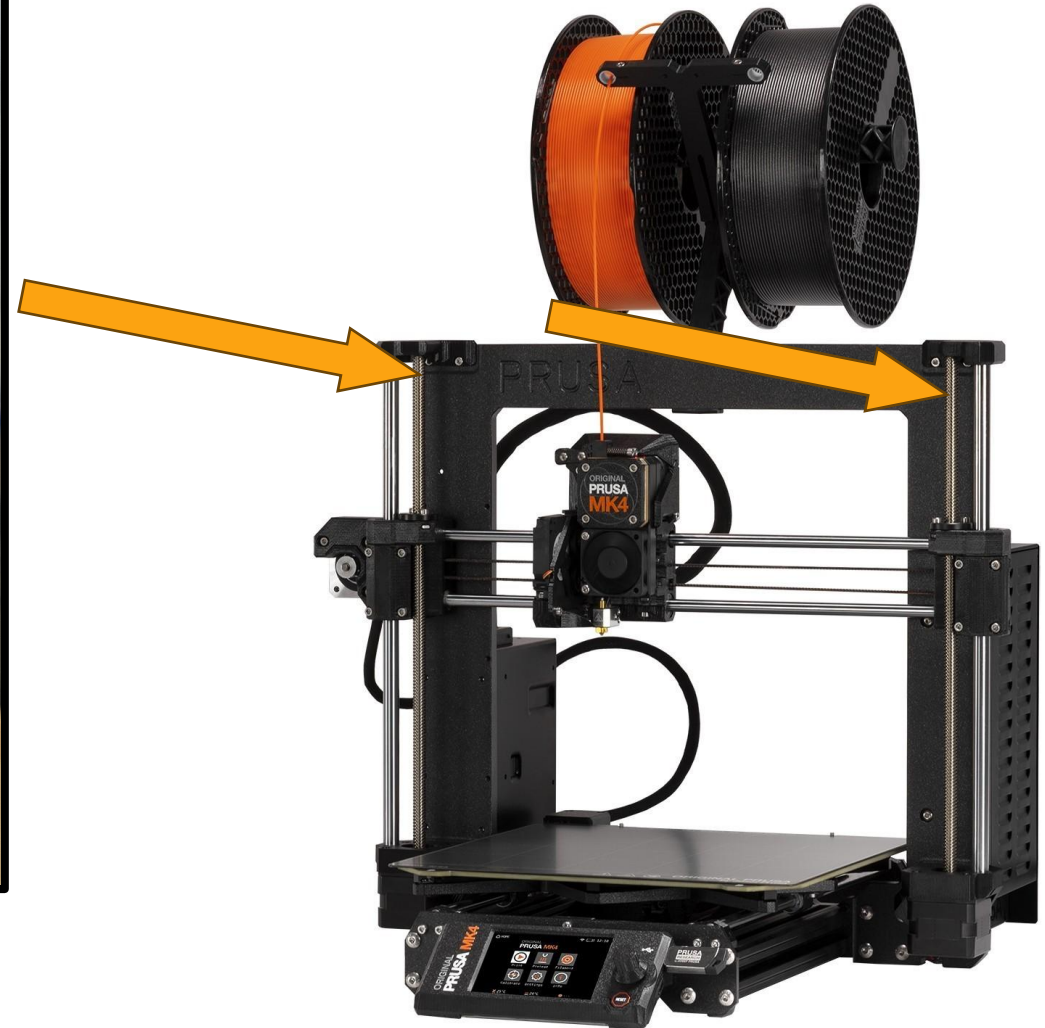
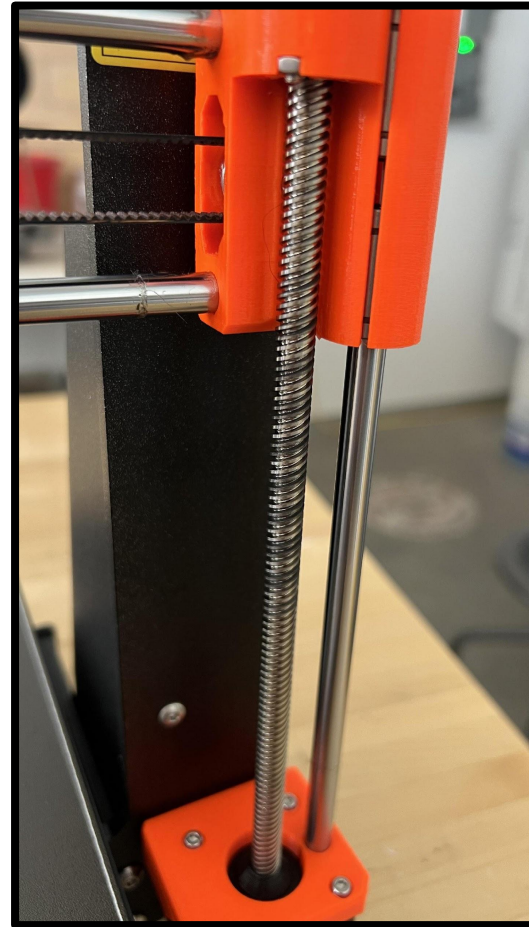
## LINEAR RAILS & BEARINGS

When we move our printer's axes, we want them to be stable and move in a very straight line. We use linear rails and bearings to achieve this.



## LEAD SCREWS

We often use screws to raise and lower the build plate of the printer. If you've ever learned about simple machines, you might remember a screw can give you mechanical advantage. We also want to have very fine control over the height of the bed.



# KAHOOT TIME!

- 'YEA 3D Printing Introduction'



# FILAMENTS/ PLASTICS

This is the plastic the 3d printer melts and lays down. It is a 1.75mm continuous strand rolled up. They generally come in 1 kilogram reels (2.2 pounds).

# WHAT KIND OF PLASTICS CAN WE USE?

FDM 3d printing uses a category of plastics called '**thermoplastics.**' These plastics get soft with heat and can be melted repeatedly without any change in material properties.

Thermo - from Greek therm meaning "heat"

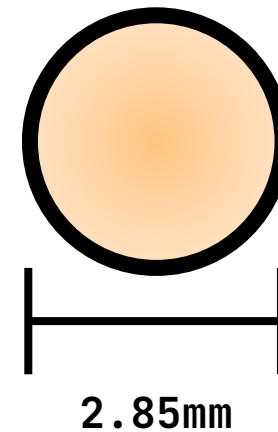
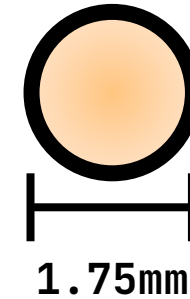
Plastic - from latin plasticus meaning "of molding"

## FILAMENT DIAMETERS

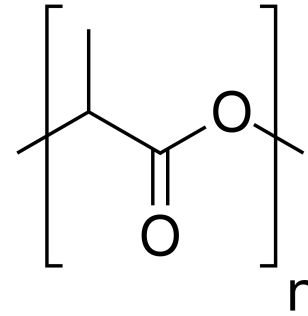
A quick note about diameters of filaments used in FDM printers.

Most printers, **including ours**, utilize 1.75mm filaments. This is thick enough to not break, but thin enough to easily control flow rate.

In the past, 2.85mm filament was more common with Bowden tube systems. In fact, most Ultimaker printers still use it. However, it's worth noting that their most expensive ones have switched to 1.75mm and direct drive extruders.



## PLA - POLYLACTIC ACID

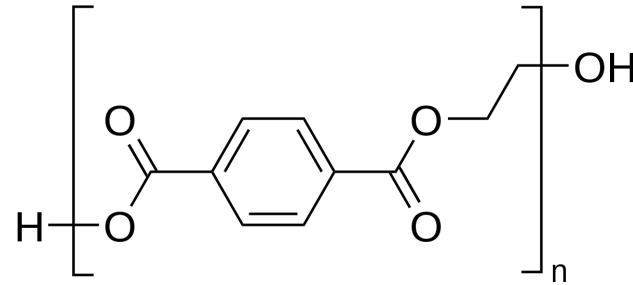


PLA is the most commonly used plastic for FDM printing. This is due to many factors such as:

- Biodegradable
- Made from biomass resources such as corn starch.
- High strength
- Low melting temperature
- *Relatively* safe/inert chemically

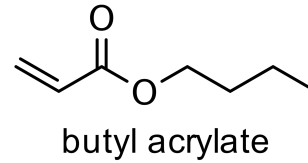
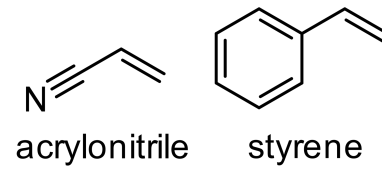
This is what we'll be using in this class.

# PETG - POLYETHYLENE TEREPHTHALATE GLYCOL



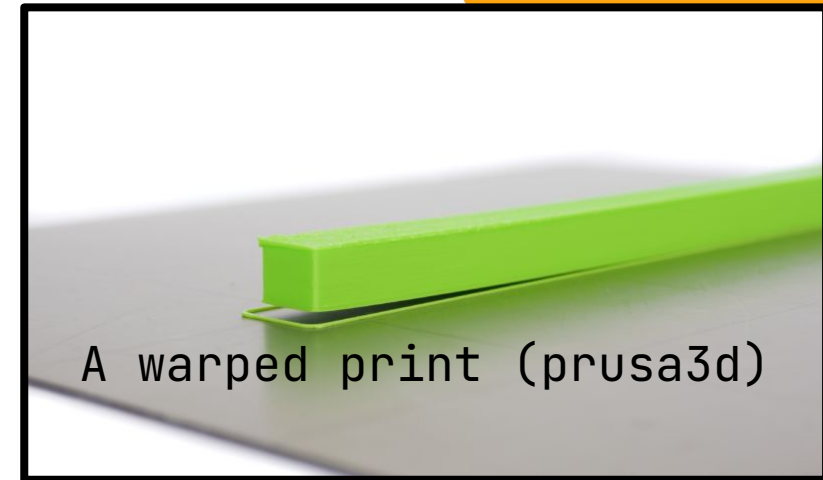
PETG is the second most popular plastic behind PLA for 3d printers. It prints at a higher temperature but is most notable for being less brittle. This makes it very useful for thin prints for parts you need to bend slightly during use.

# ASA - ACRYLONITRILE STYRENE ACRYLATE



ASA is a much more difficult plastic to print. It tends to warp and peel. However, it can be very useful due to:

- Chemical resistance
- UV resistance (doesn't get brittle/weak in the sun)
- High glass transition temperature (temperature at which it is not fully melted, but greatly softened)



# CARBON FIBER ADDITIVES

A quick note on carbon fiber filament.

Carbon fiber is often ground up and added to plastics like PLA, PETG, or nylon. However, it isn't always better. It makes prints more **stiff** but also more **brittle**.

Commonly, we add carbon fiber to PLA or PETG simply to improve looks. Carbon fiber prints have some interesting texture and matte appearance.



**T I N**  
**K E R**  
**C A D**

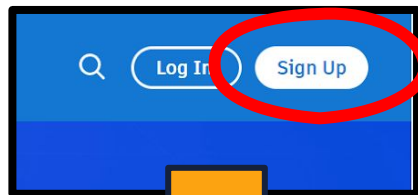
# **TINKERCAD**

This is the software we'll use to make a model by combining basic shapes together.

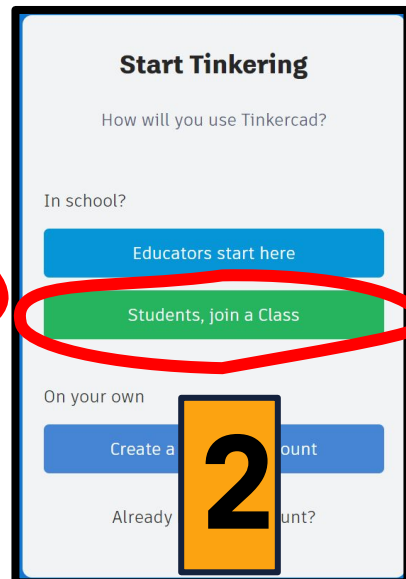
## TINKERCAD SIGNUP

First, we need to make an account. We'll be using a classroom for this.

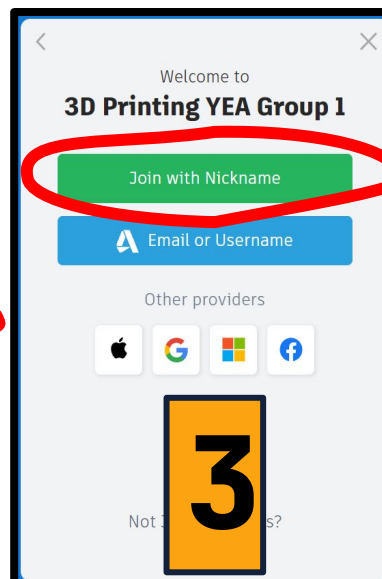
- Go to tinkercad.com or look it up on Google.
- Use the 'Sign Up' Button
- Enter the code given by your teacher
- Enter your nickname given by your teacher.



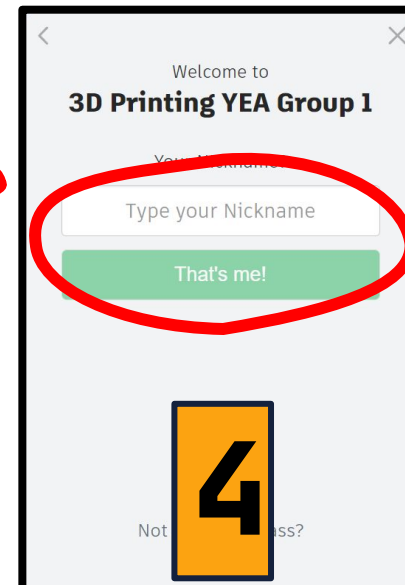
1



2



3



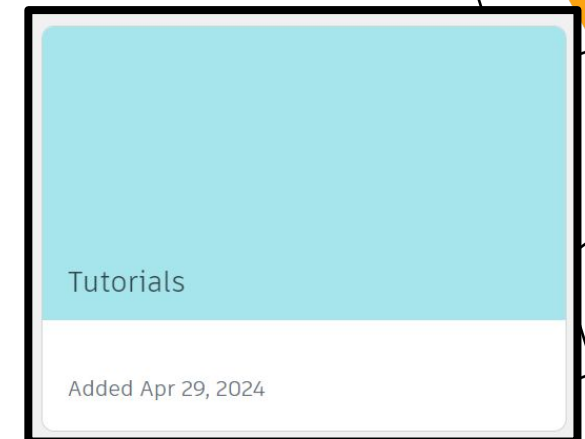
4

## TUTORIALS

To get you familiar with Tinkercad, you are going to complete the basic tutorials.

- Go to the classes tab
- Select the class
- Enter the Tutorials Activity
- Complete the assigned tutorials

**Teachers will go around helping. Please ask for help!**

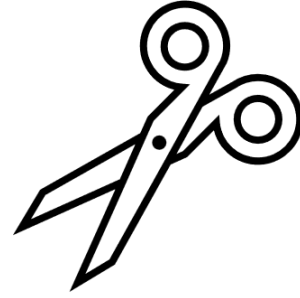




# PRUSA SLICER

This is the software that  
changes our model into  
**instructions** for the 3d  
printer.

## WHAT IS A SLICER?



We call the program a slicer because it takes a mesh object and 'cuts it up' for the printer.

The slicer tells the printer how fast to go, how hot to print, where to move.

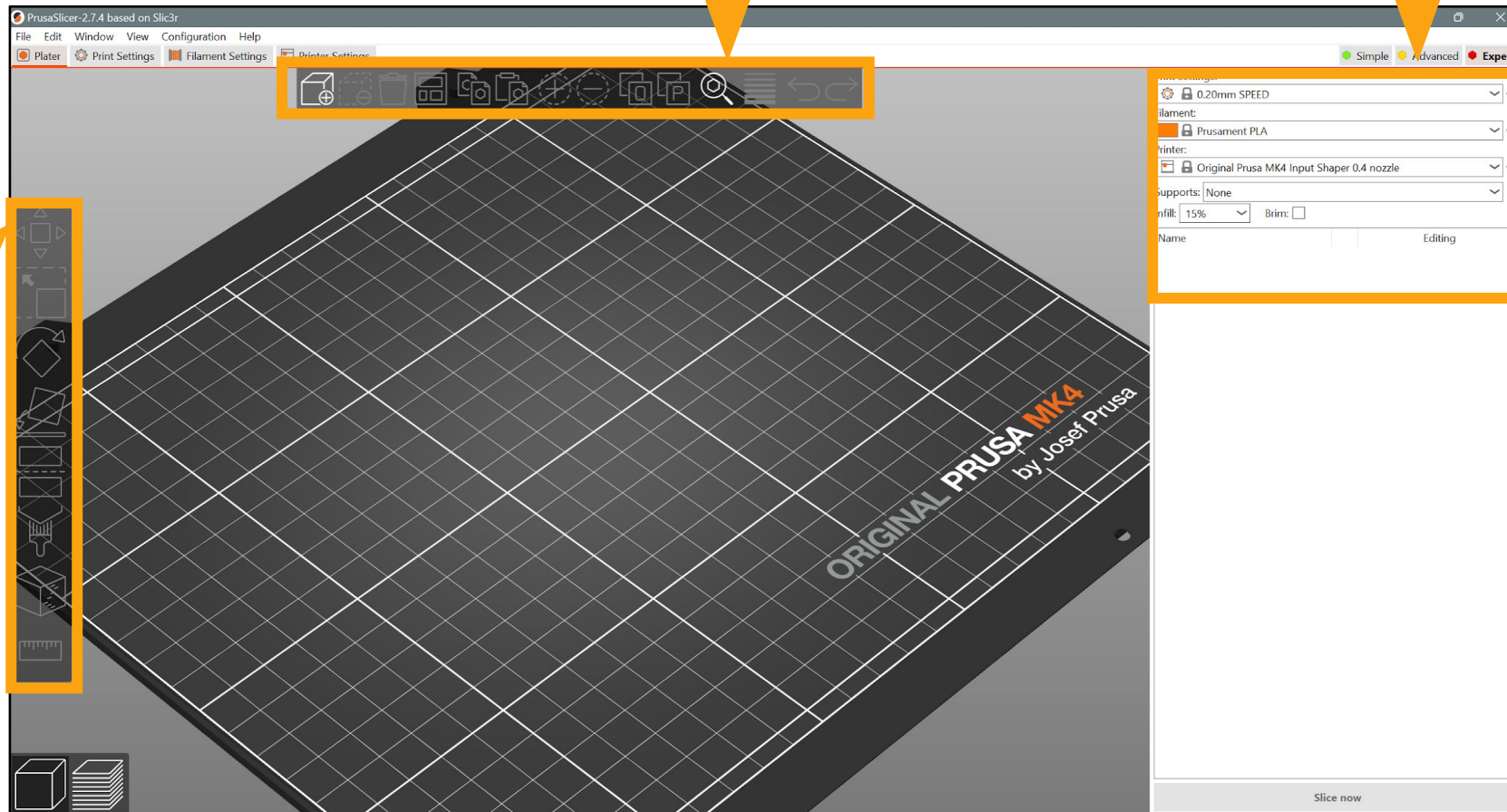
This means we choose detail settings, orient parts, choose materials, all inside this step.

## OVERVIEW

Plate Controls

Slice Settings

Part & Object Controls



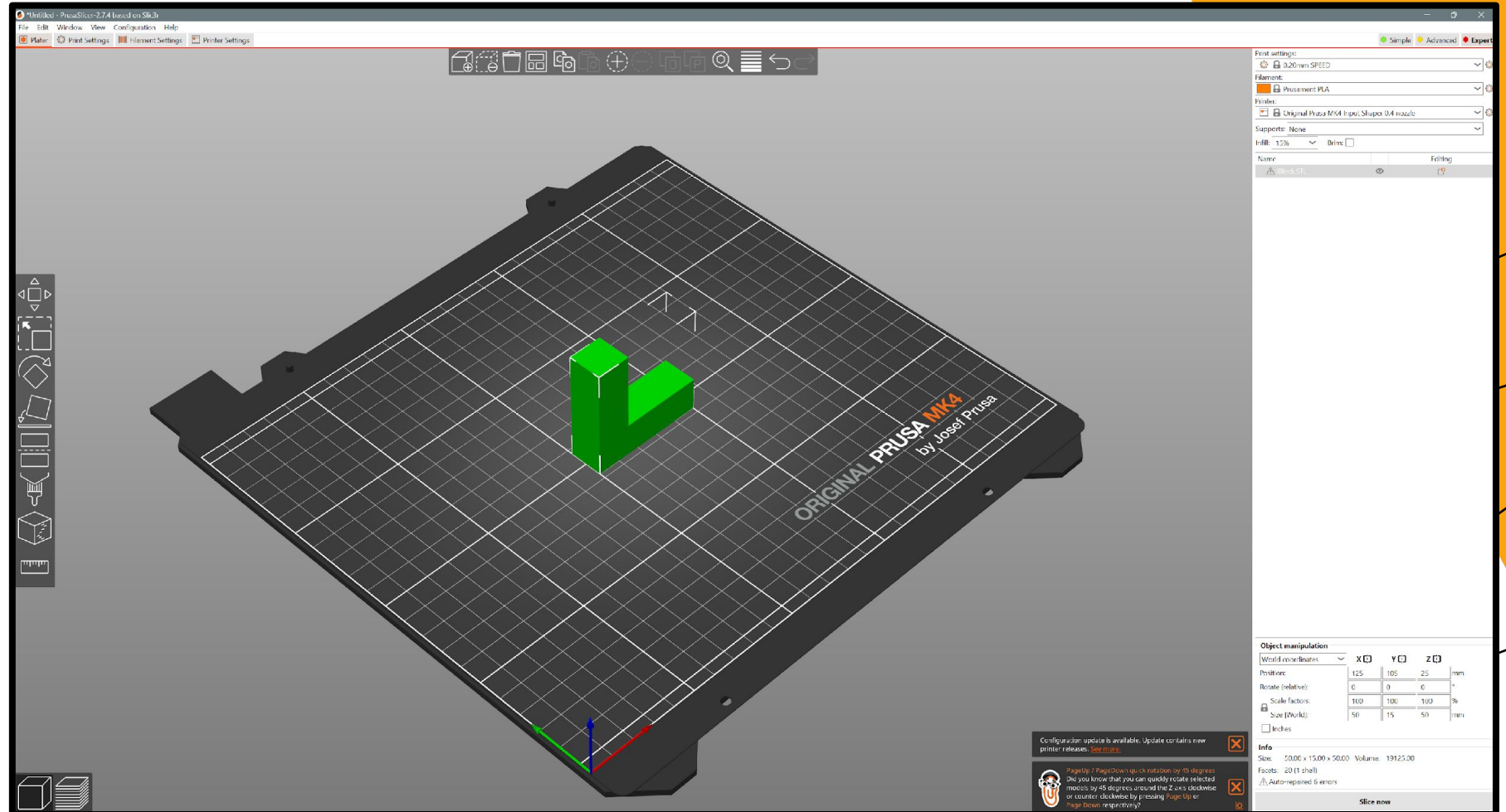
## IMPORT A PART

First, import your part (.stl file).

You can drag and drop it into the window from File Explorer

OR

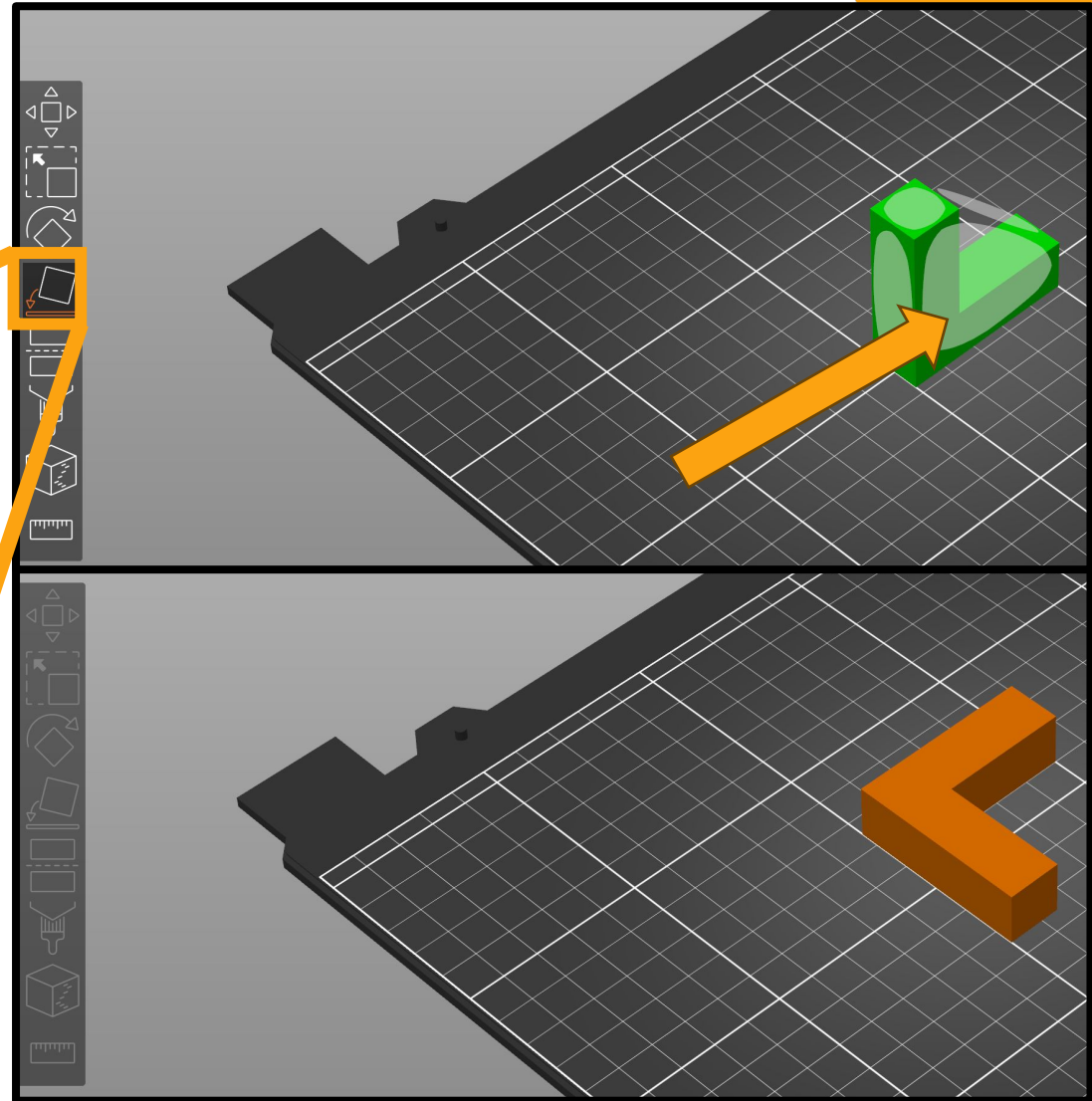
File > Import > Import STL/3MF...



# ORIENT YOUR PART

Remember, we want to avoid overhangs and keep large flat surface in the build plate.

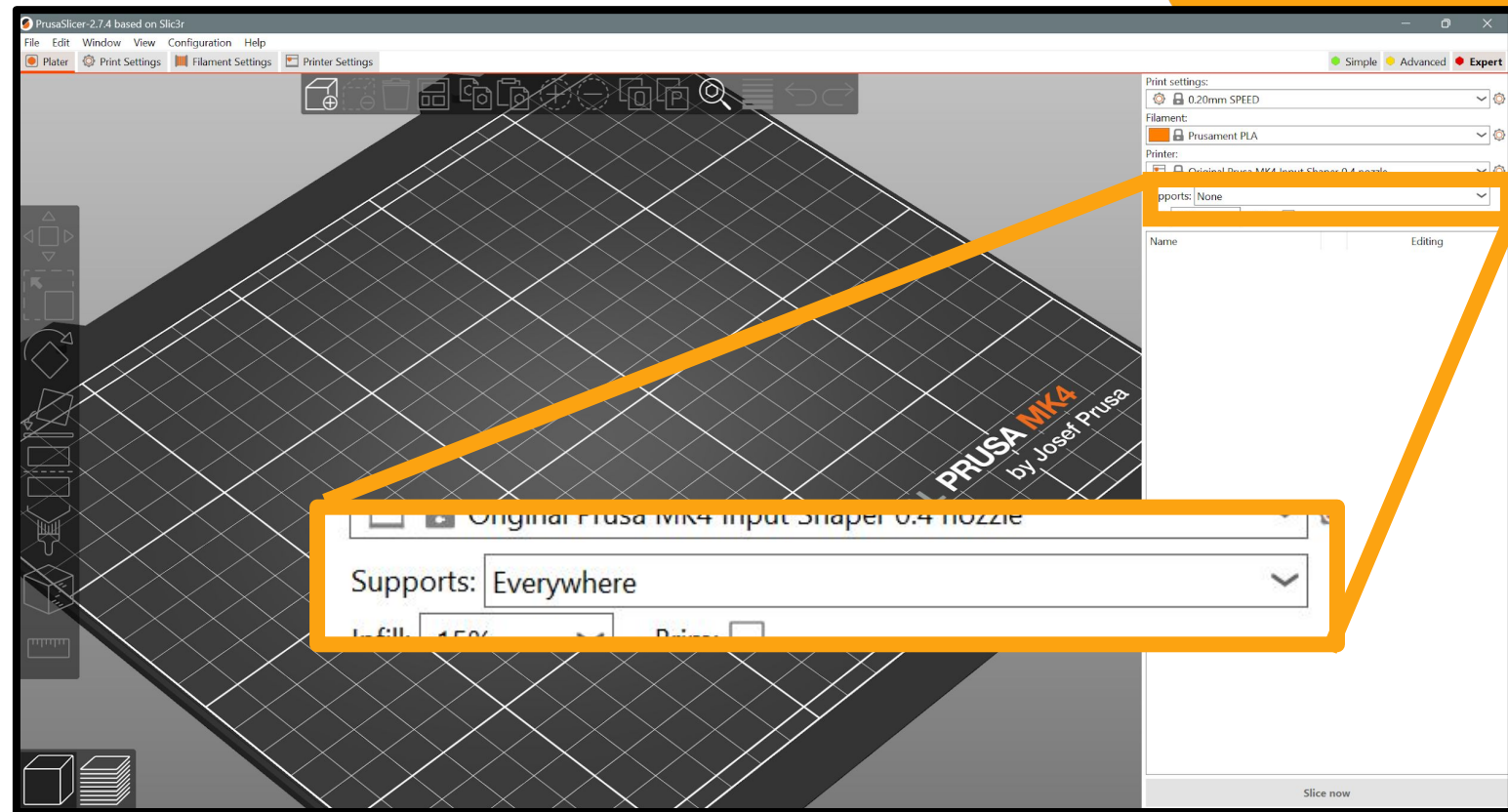
First select your part.  
Then, using the 'lay flat' tool, click a face to align it to the plate.



## ADDING SUPPORT (IF NEEDED)

By turning on this setting, the slicer will detect likely problems and add supports.

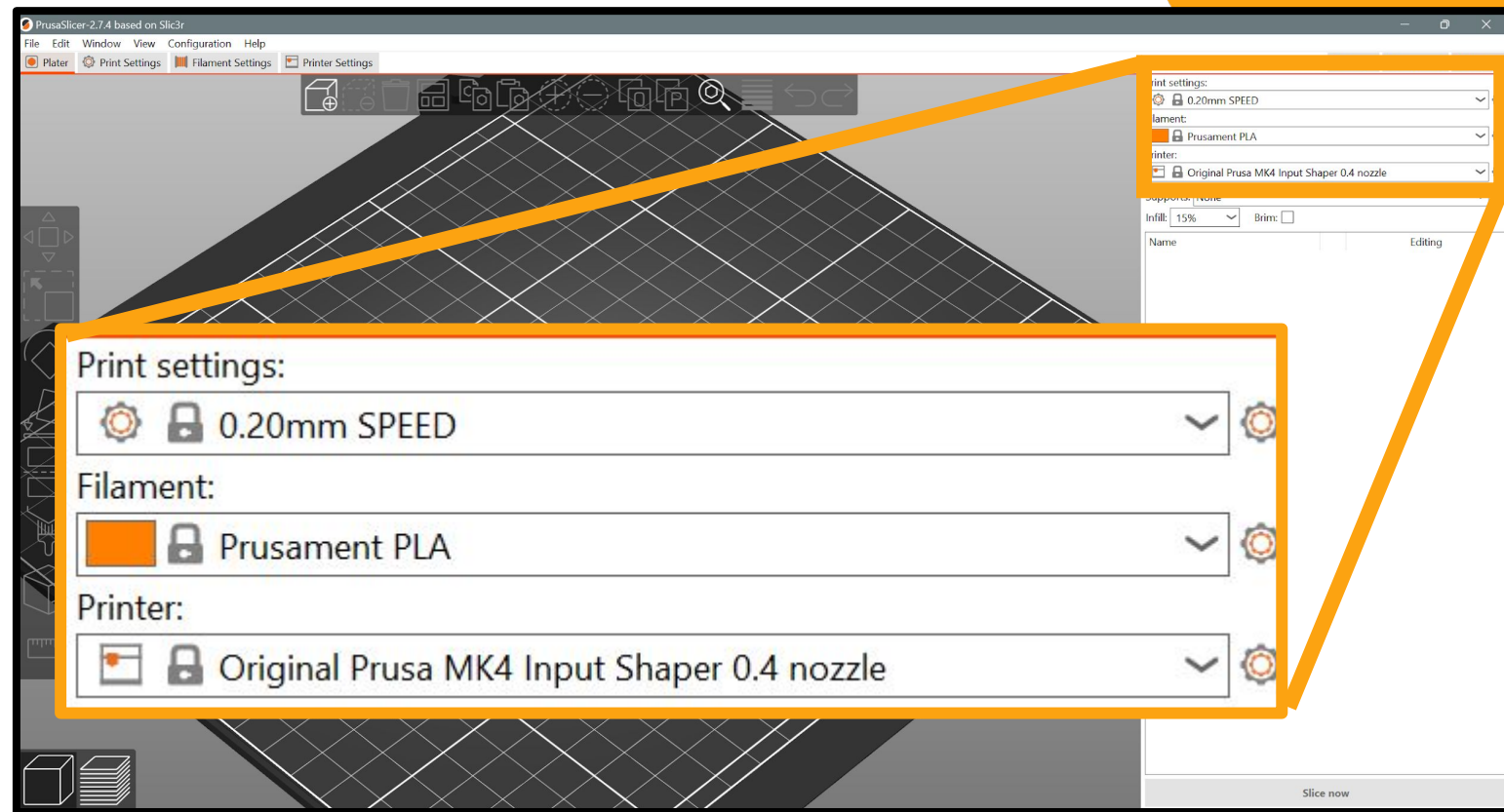
On more complicated parts, we can add them manually if necessary.



## FINAL SETUP

Select the printer, filament, and print settings profile.

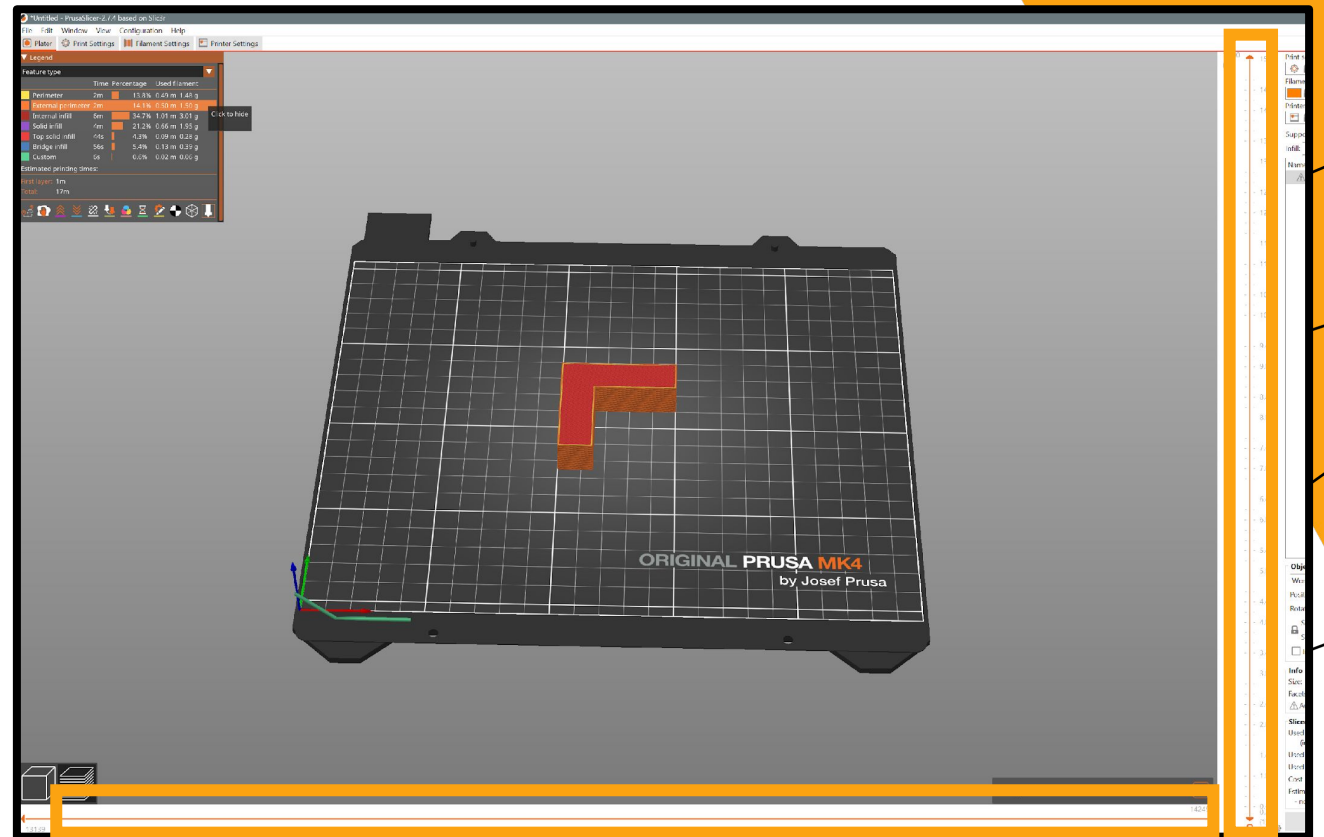
Note the print settings change depending on the printer.



# SLICING

Slice now

Click the 'Slice now' button. Now you can export your slice '.gcode' file that goes straight to the printer. You can look through the layers and preview the printer's movement by using the sliders shown.



## SLICING

Once we've sliced our model, pay attention to some important information it gives us.

It tells us the Used Filament in grams (remember filament rolls are 1 kilogram or 1000 grams).

It also gives us an estimation of how long the print will take with our settings.

### Object manipulation

World coordinates x [ ] y [ ] z [ ]

|                    |     |     |     |    |
|--------------------|-----|-----|-----|----|
| Position:          | 125 | 105 | 7.5 | mm |
| Rotate (relative): | 0   | 0   | 0   | °  |
| Scale factors:     | 100 | 100 | 100 | %  |
| Size [World]:      | 50  | 50  | 15  | mm |

Inches

---

### Info

Size: 50.00 x 15.00 x 50.00 Volume: 19125.00  
 Facets: 20 (1 shell)  
 Auto-repaired 6 errors

---

### Sliced Info

|                                  |               |
|----------------------------------|---------------|
| Used Filament (g)                | 8.66 (201.66) |
| (including spool)                |               |
| Used Filament (m)                | 2.90          |
| Used Filament (mm <sup>3</sup> ) | 6983.99       |
| Cost                             | 0.31          |
| Estimated printing time:         |               |
| - normal mode                    | 17m           |

Export G-code

## SLICING

Once we've sliced our model, pay attention to some important information it gives us.

It tells us the Used Filament in grams (remember filament rolls are 1 kilogram or 1000 grams).

It also gives us an estimation of how long the print will take with our settings.

### Object manipulation

World coordinates x [ ] y [ ] z [ ]

|                    |     |     |     |    |
|--------------------|-----|-----|-----|----|
| Position:          | 125 | 105 | 7.5 | mm |
| Rotate (relative): | 0   | 0   | 0   | °  |
| Scale factors:     | 100 | 100 | 100 | %  |
| Size [World]:      | 50  | 50  | 15  | mm |

Inches

---

### Info

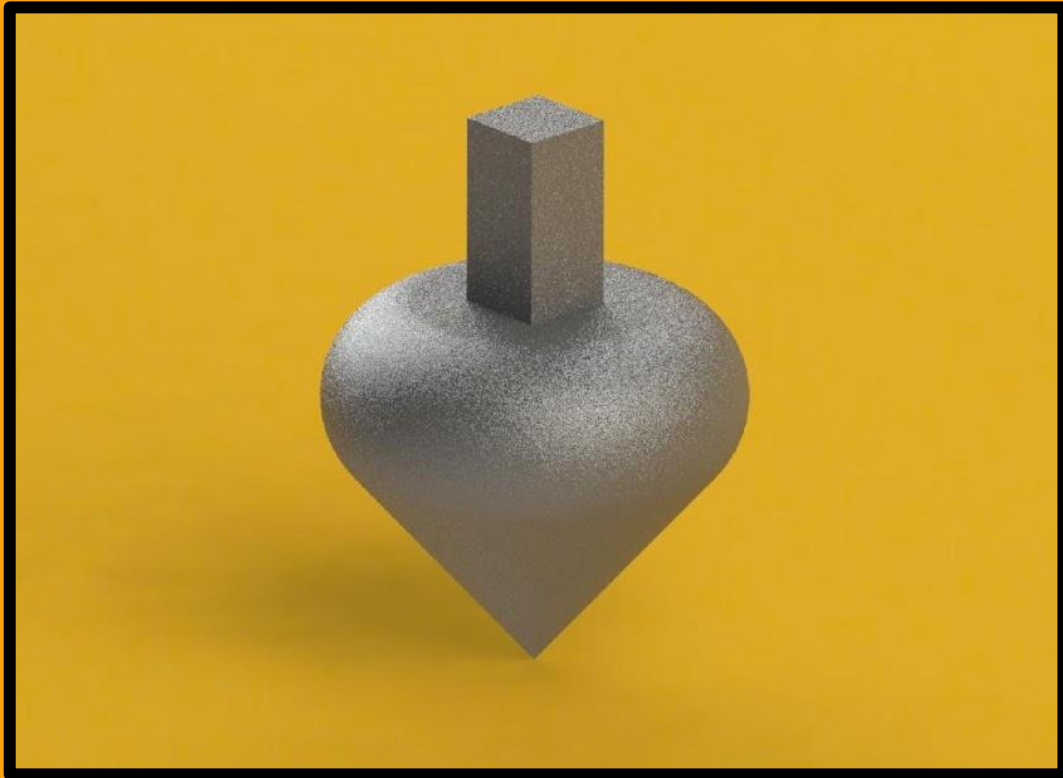
Size: 50.00 x 15.00 x 50.00 Volume: 19125.00  
 Facets: 20 (1 shell)  
 Auto-repaired 6 errors

---

### Sliced Info

|                                  |               |
|----------------------------------|---------------|
| Used Filament (g)                | 8.66 (201.66) |
| (including spool)                |               |
| Used Filament (m)                | 2.90          |
| Used Filament (mm <sup>3</sup> ) | 6983.99       |
| Cost                             | 0.31          |
| Estimated printing time:         |               |
| - normal mode                    | 17m           |

[Export G-code](#)



# FINAL PROJECT

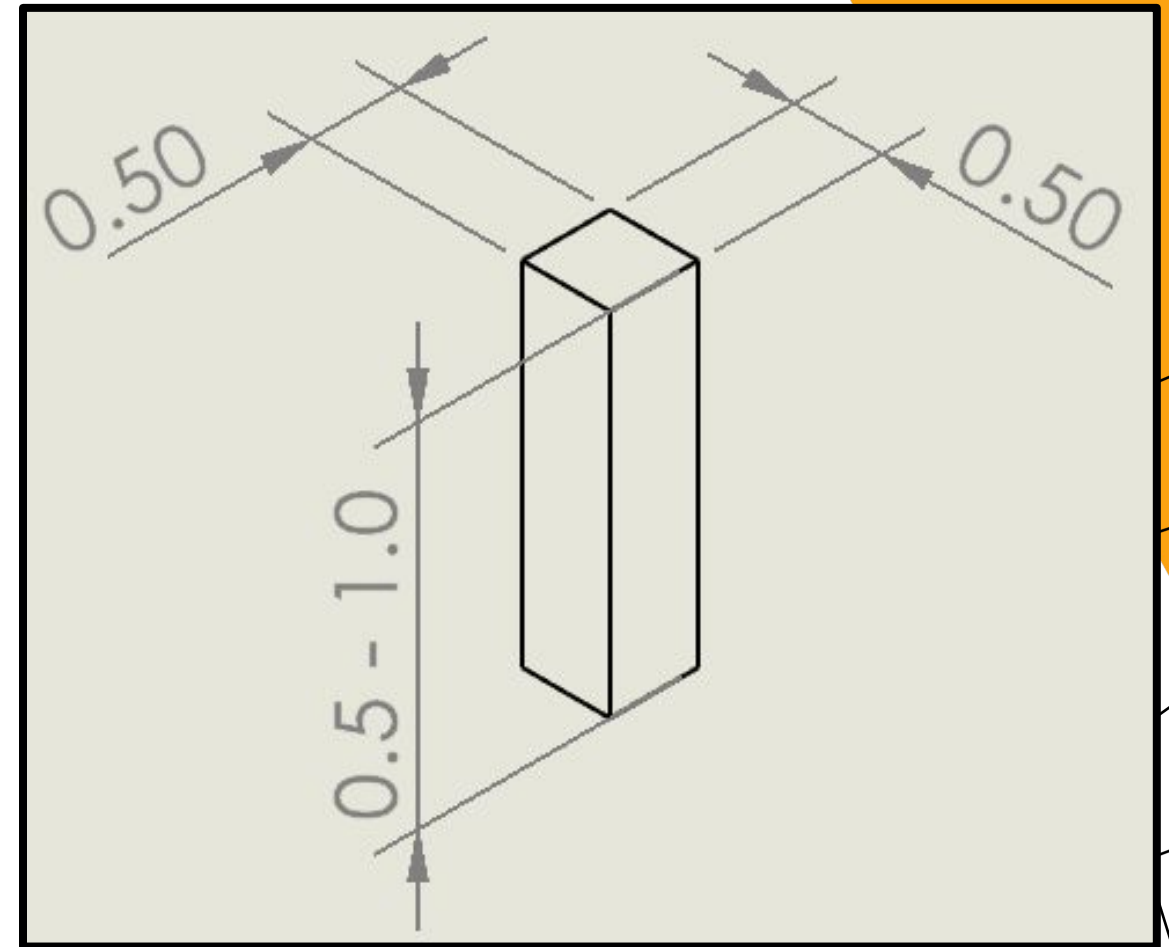
Let's make some spinning tops!

## ADD THE SHAFT

Once we have the body of the top modeled, we'll add the shaft to spin it with.

Because we want everybody's part to fit in the spinning apparatus, we all need to have the same size of shaft.

Make the hole  $0.5'' \times 0.5''$  and anywhere from  $0.5''$  to  $1''$  deep.



# COMPETITION & DESIGN CONSTRAINTS

On Friday, we will be competing to see who makes the top that can spin longest.

Your top needs to follow these rules:

- Fits standard shaft -  
0.5"x0.5"x.5"
- Prints in less than 20 grams of plastic (check in slicer)

# EXPECTATIONS

In order to check print viability and estimations of material used (in PrusaSlicer), you'll be changing and exporting file quite often. This is the intended goal, to get you comfortable with the workflow.