SUU

BZI

YOUTH ENGINEERING ACADEMY

FUNDAMENTALS OF 3D PRINTING

SESSION 1

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

00:00

ADDITIVE MANUFACTURING

3D printing is a subsection of additive manufacturing. Meaning, we build up our part from nothing rather than removing material from a larger block. There are many types of 3D printing such as:

- SLA
- SLS
- FDM
- & more...

We'll cover a few now.

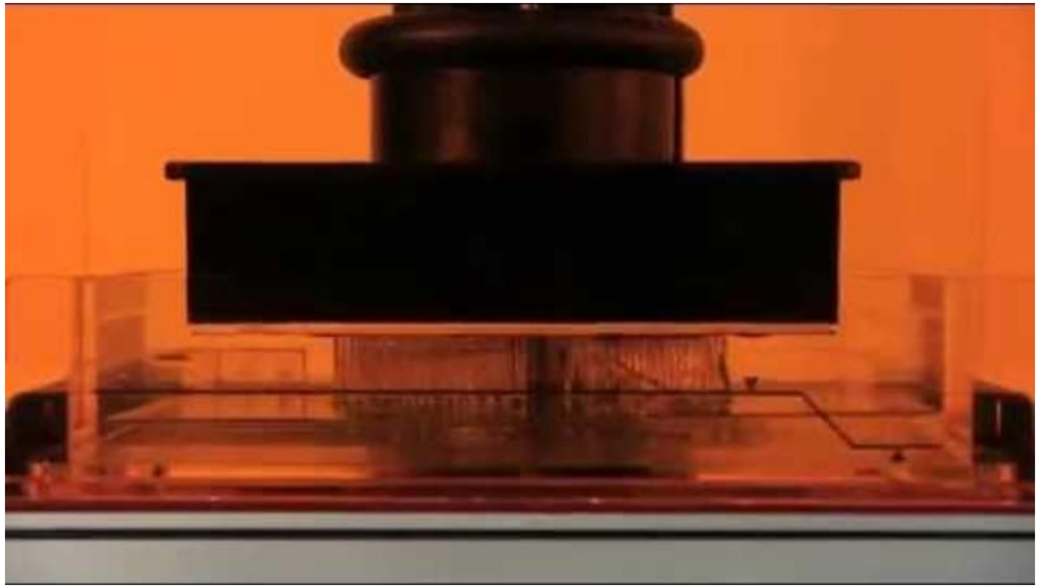
This is opposed to subtractive manufacturing like milling.

Xometry

00:02

STEREOLITHOGRAPHY (SLA / RESIN PRINTING)

Stereolithography uses a UV light or laser to ‘cure’ or harden a liquid resin. This process commonly allows for us to make extremely detailed prints.



SELECTIVE LASER SINTERING (SLS)

Selective Laser Sintering uses a laser to melt a very fine powder into a plastic before laying a very thin layer of powder on top of the melted plastic. This is repeated and the part is built up. It is very useful because parts are very tough and the printers can be very large (several feet of print area across).

Start @ 2:52

00:06

FUSED DEPOSITION MODELING

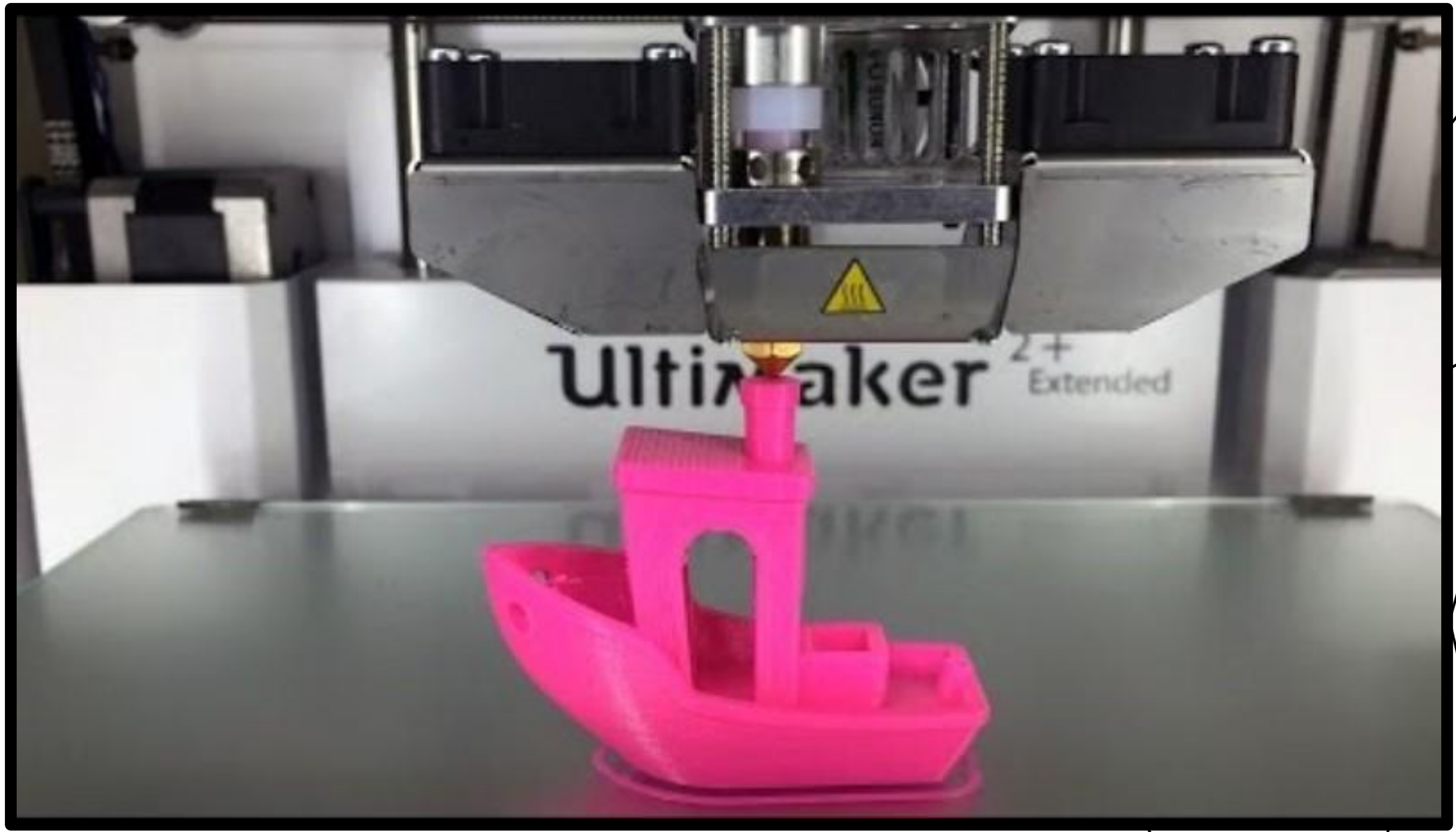
Finally, we get to FDM printing. This is what we’ll be learning about this week. FDM printers have the advantage of being the cheapest and most compact form of 3D printing. FDM printers use a very hot nozzle (210+ degrees C or 410+ Fahrenheit) to melt plastic and squeeze it down in layers, building our part from the bottom up.

00:07

PRINTING A BENCHY

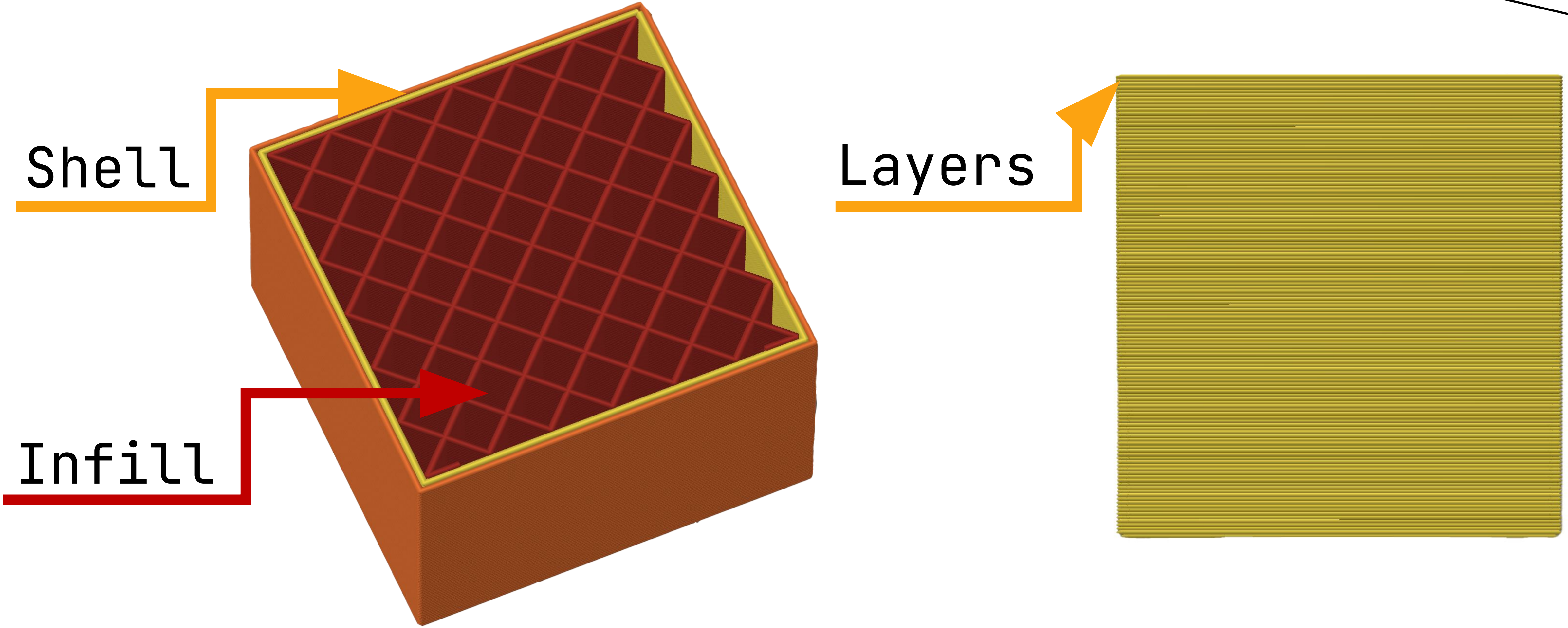
In case you have never seen an FDM 3D printer at work, here is a Benchy being printed.

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



00:10

ANATOMY OF A PART



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.

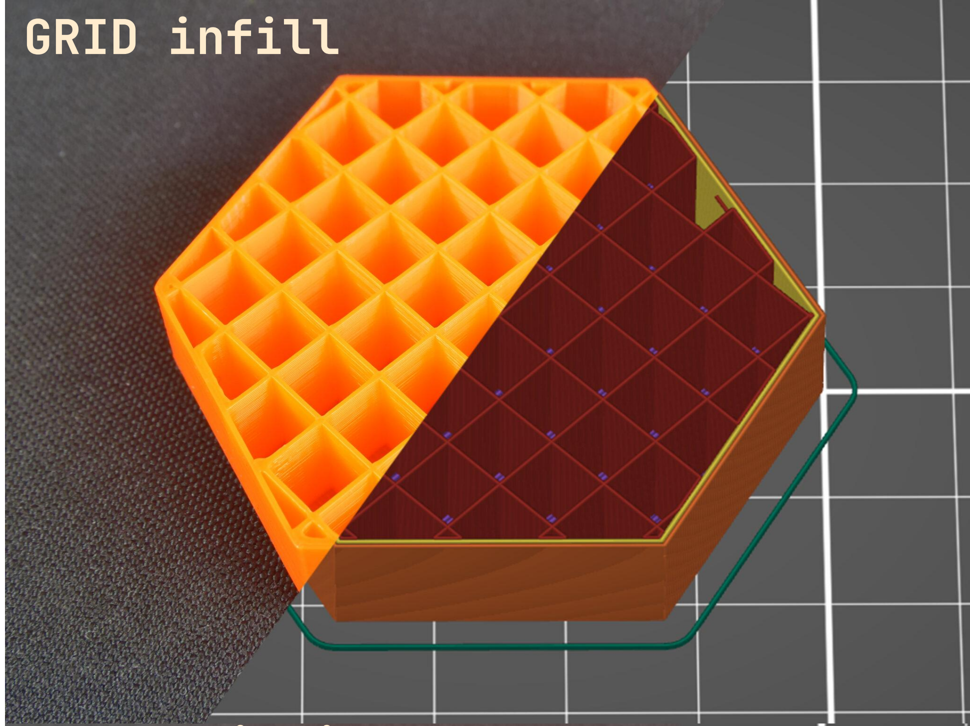
Gyroid:

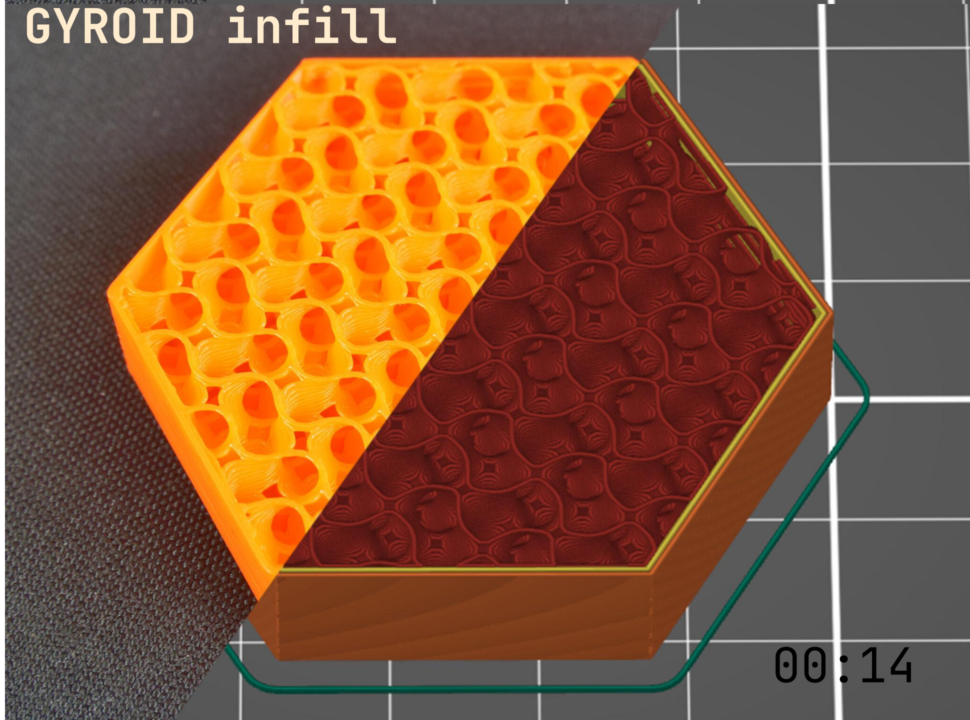
-fairly fast

-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.

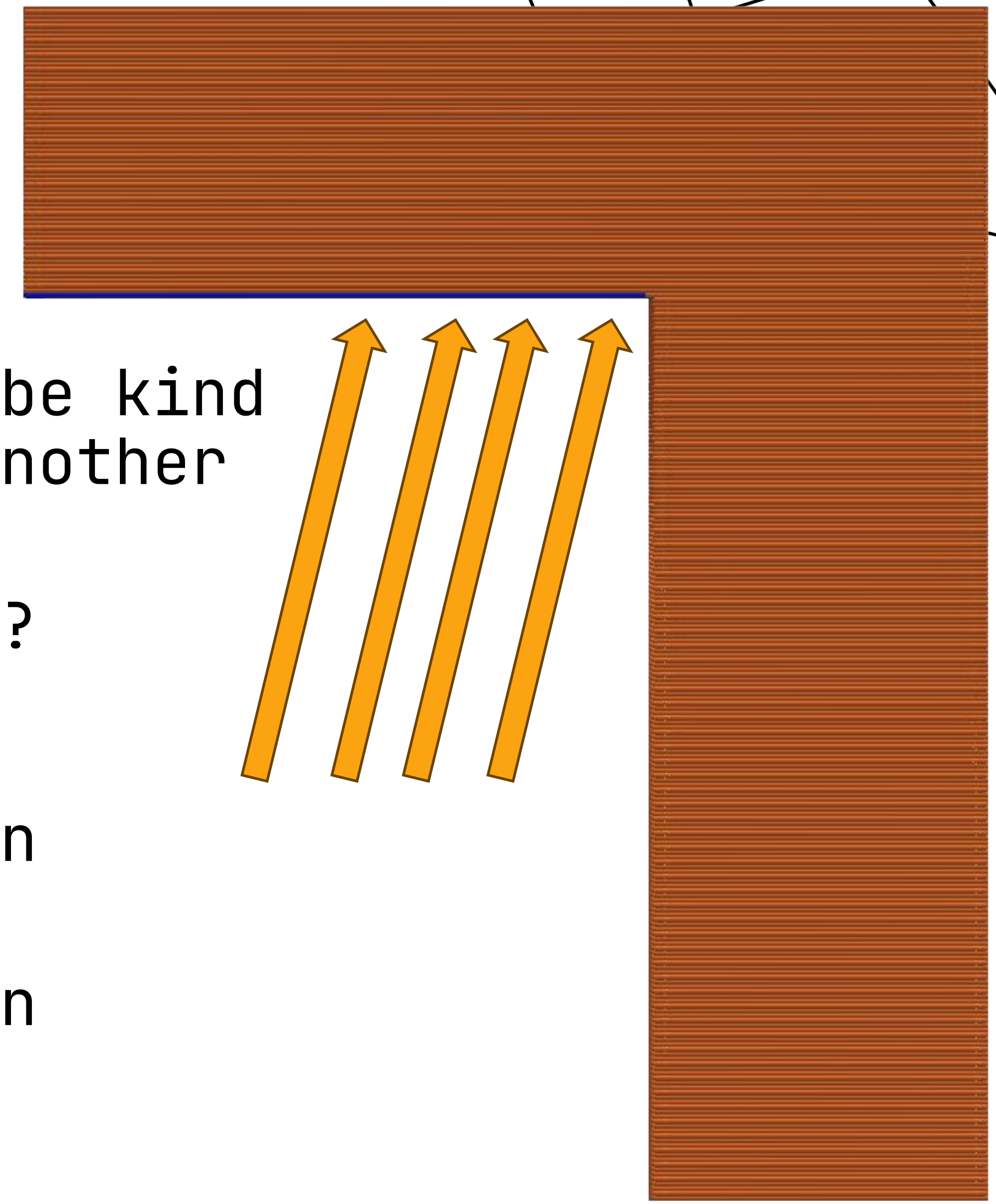




A SMALL HICCUP

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’

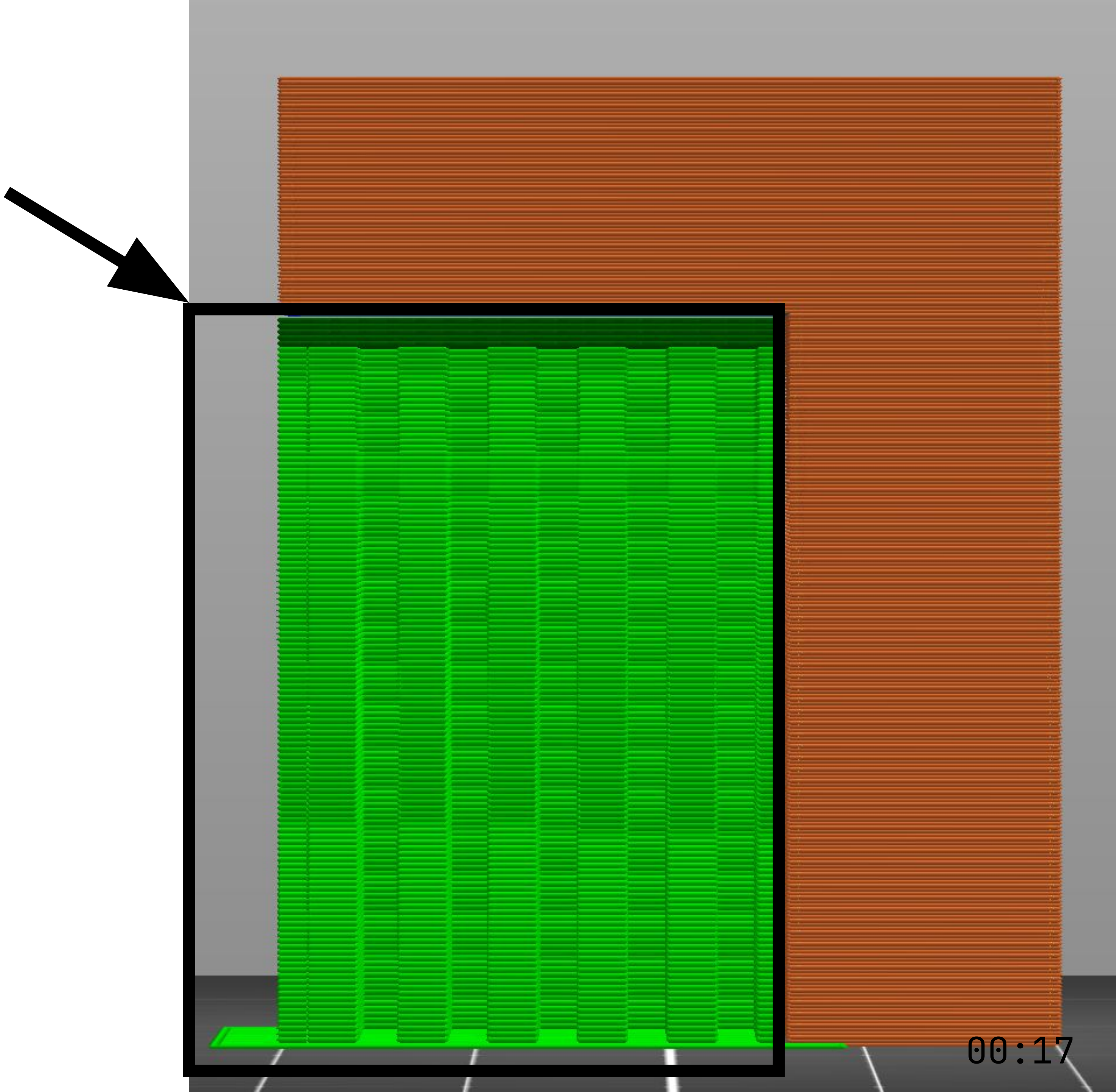
00:15

SUPPORT

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

This is done in the printer software.

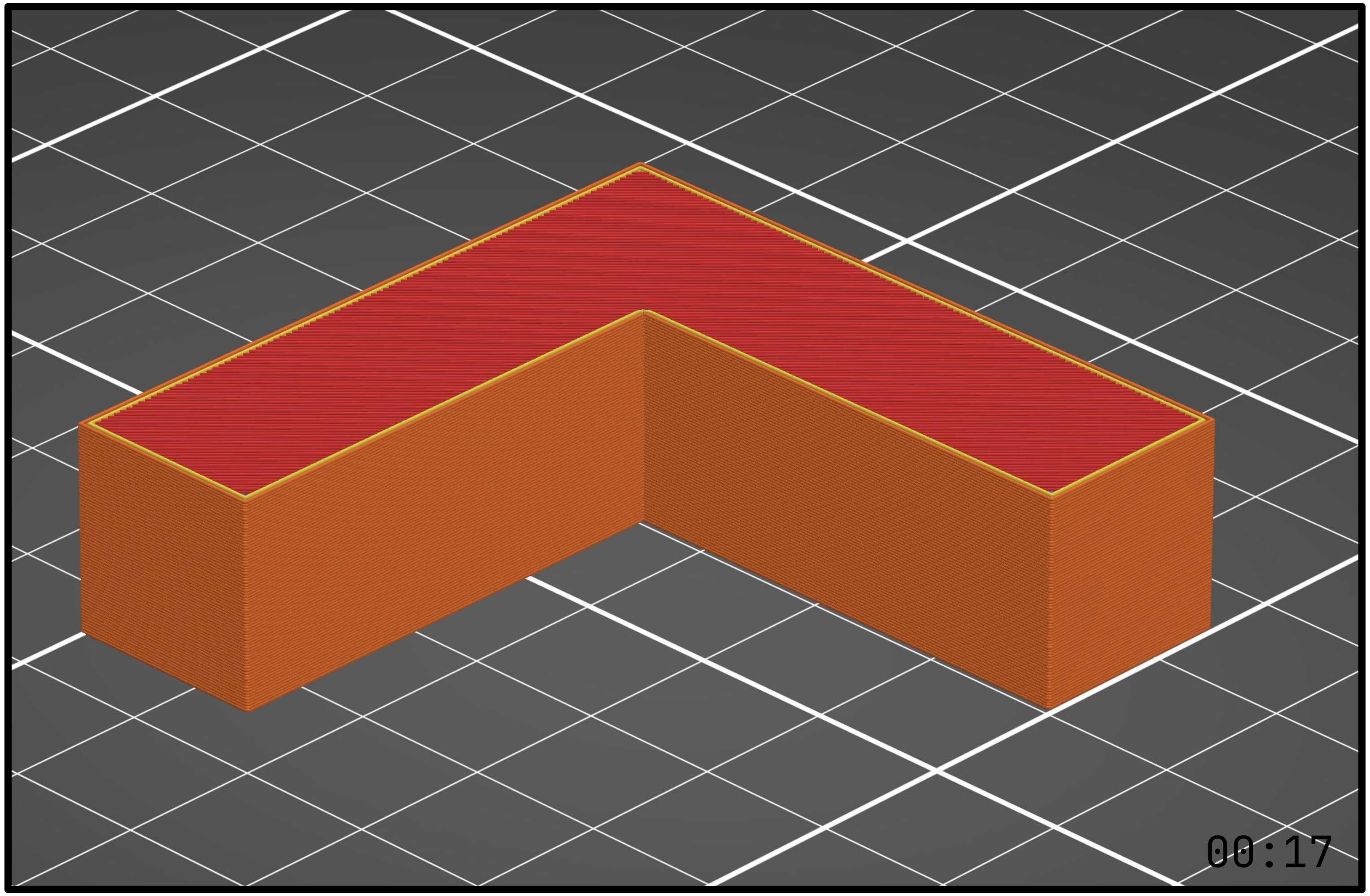
Support is meant to be broken off after the print finishes.



ALTERNATIVELY

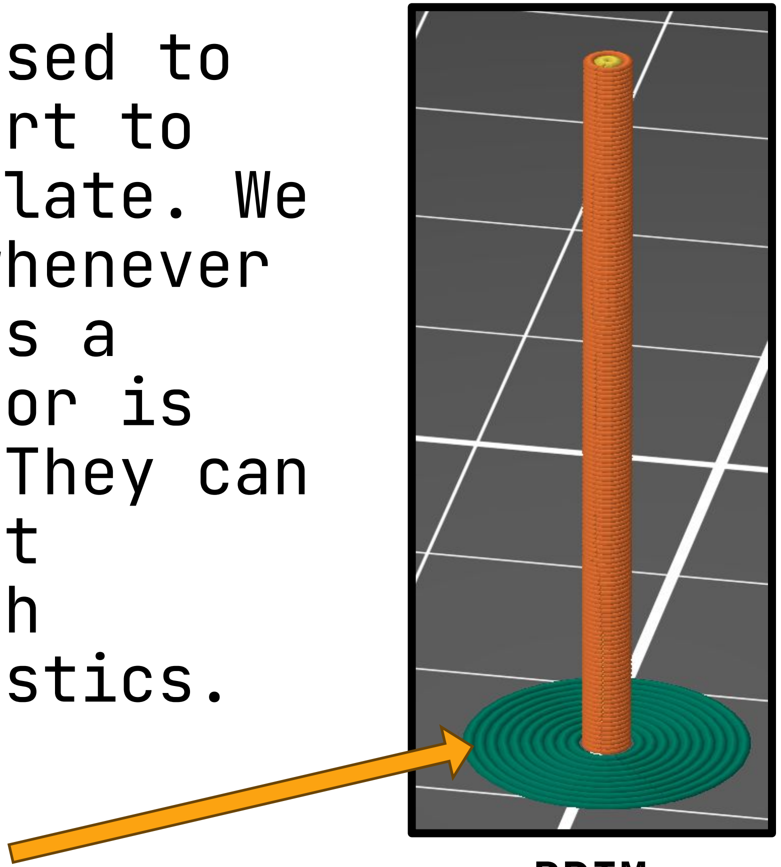
However, we can save a lot of plastic and time by just laying the part on the side.

Always try to reduce overhangs whenever possible.



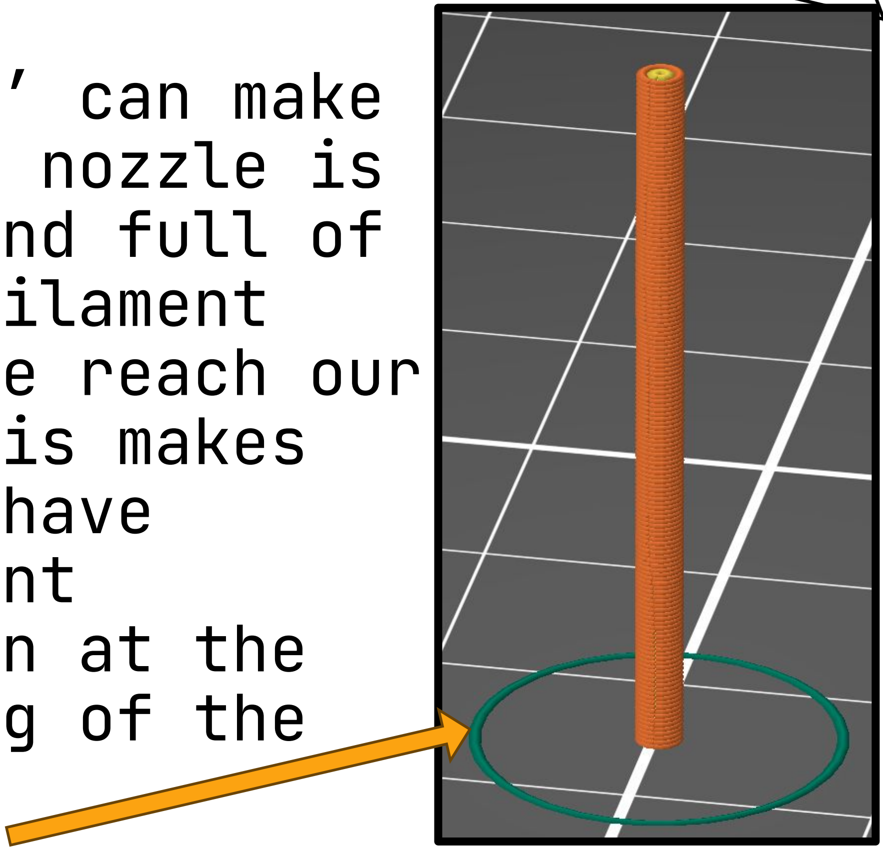
BRIMS AND SKIRTS

A brim is used to hold our part to the build plate. We use these whenever our part has a small base or is very tall. They can also prevent warping with certain plastics.



BRIM

A ‘skirt’ can make sure the nozzle is primed and full of molten filament before we reach our part. This makes sure we have consistent extrusion at the beginning of the print.

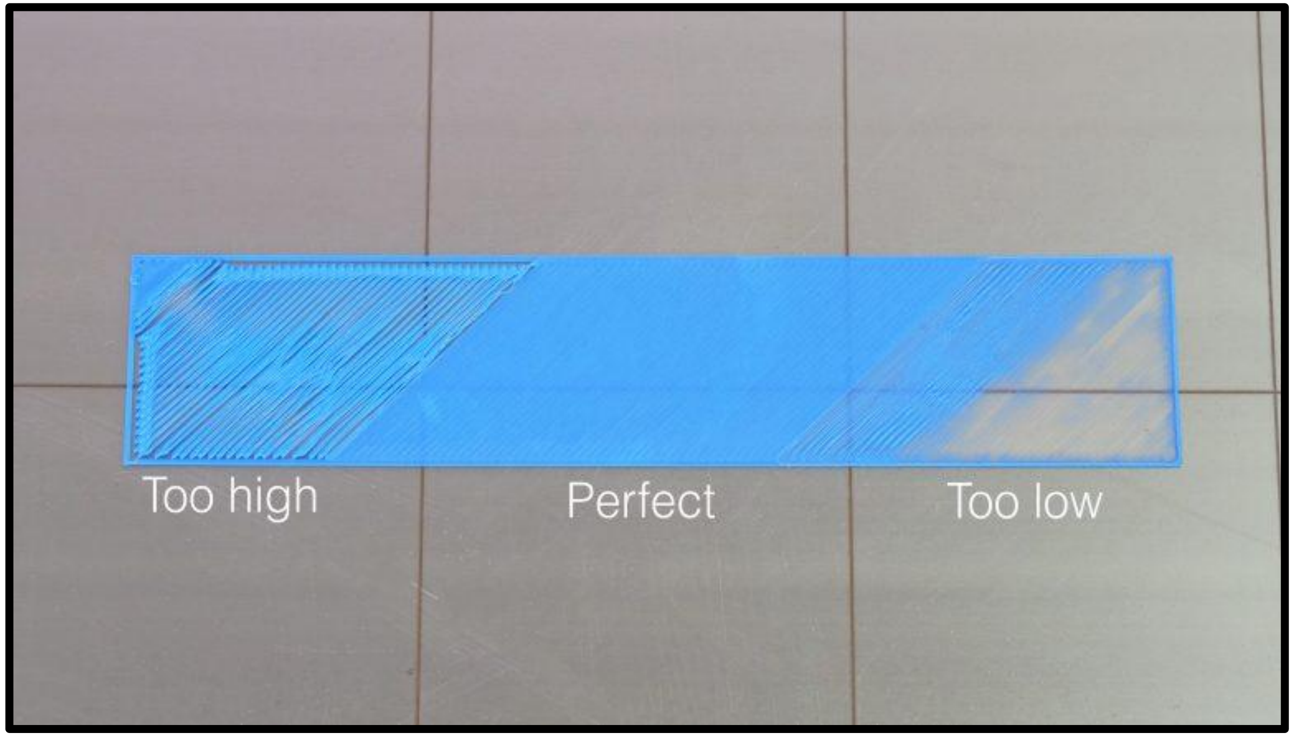


SKIRT

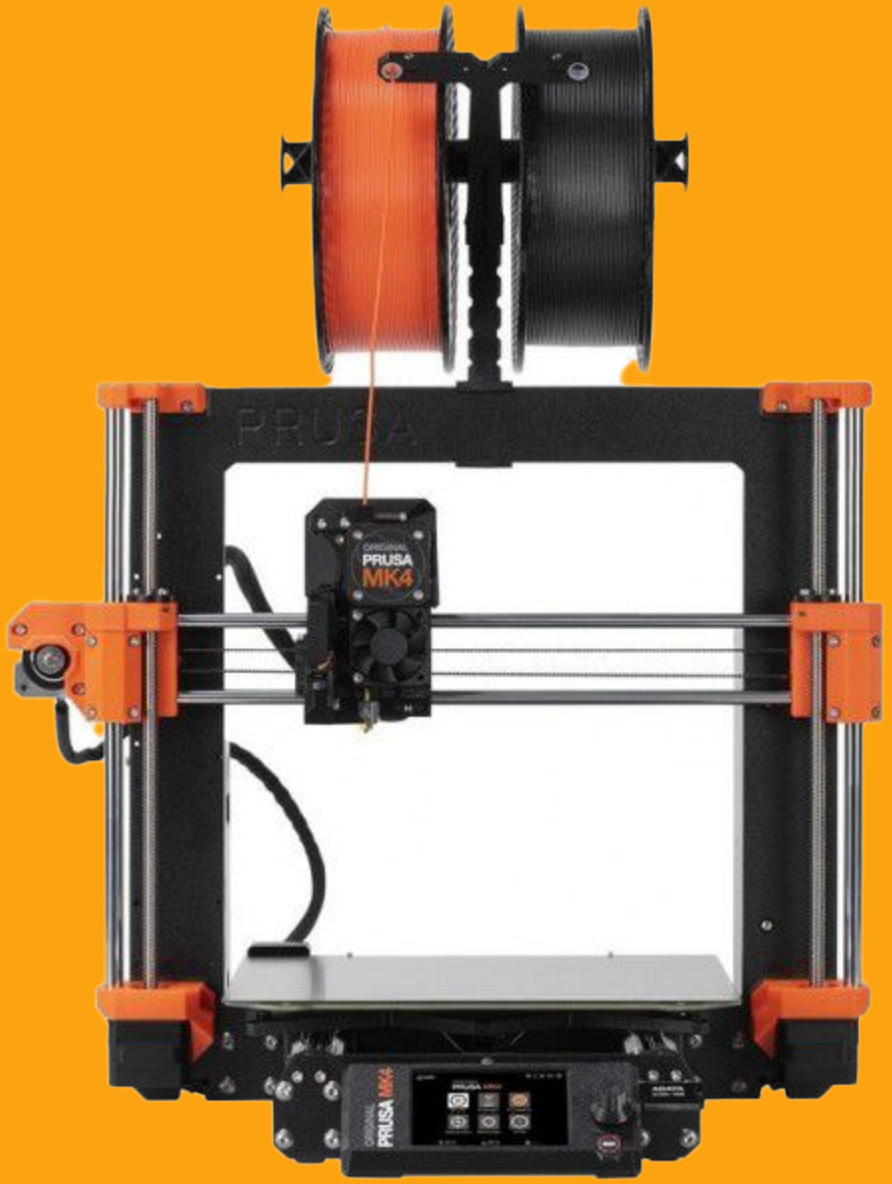
00:20

FIRST LAYER

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



00:22



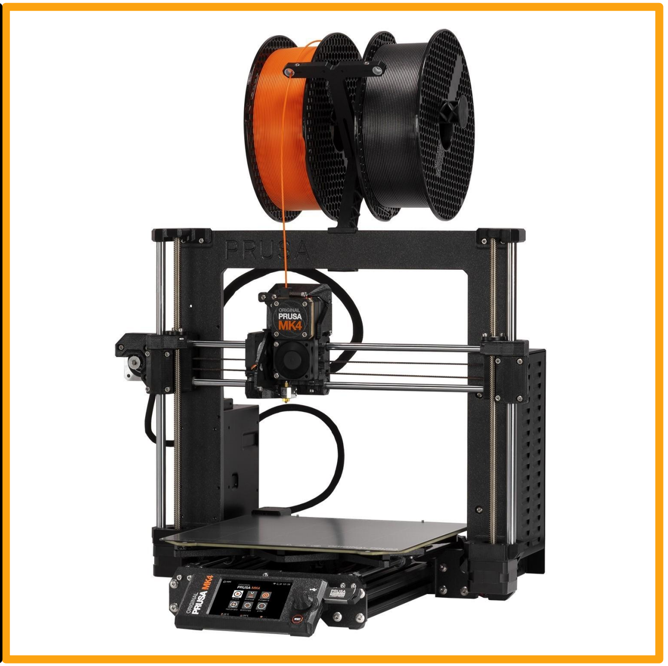
PRINTER

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.

00:24

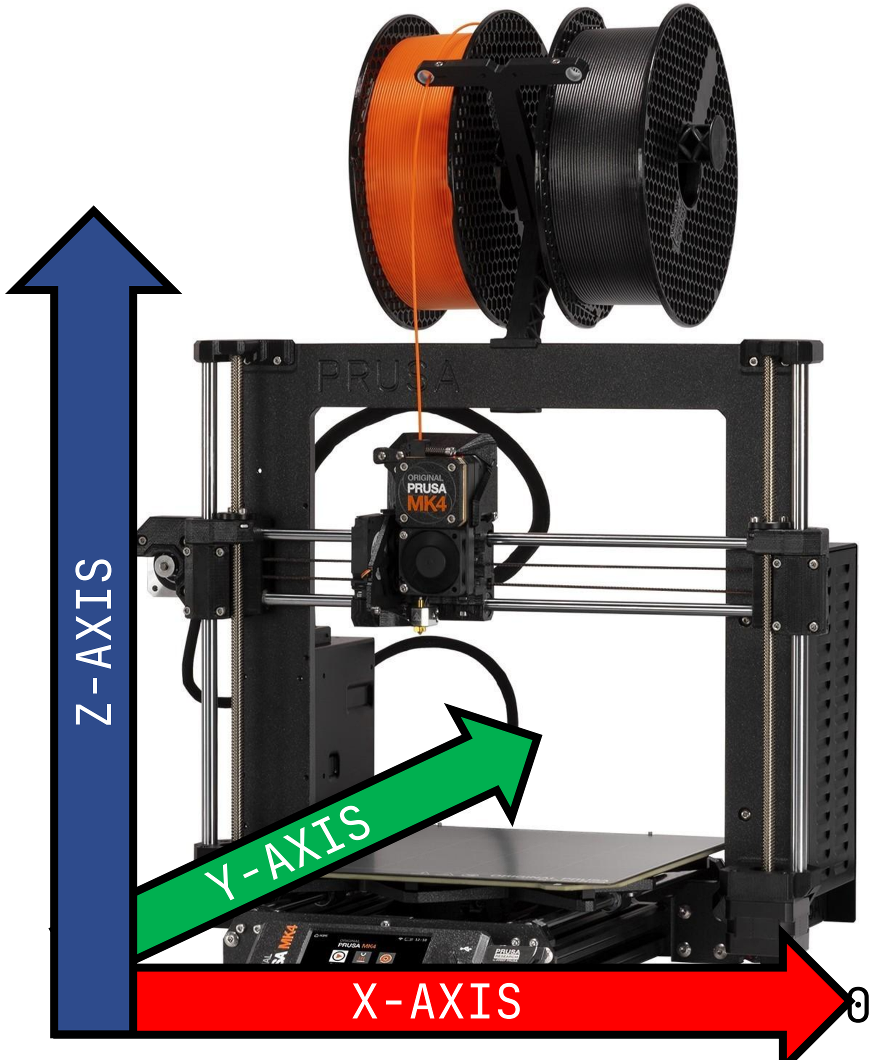
PRINTER AXES

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.

These instructions are called ‘G-code’



00:25

GCODE

Gcode is a kind of ‘language’ that our printers can understand. It might look like gibberish at first, but we can pretty quickly get the gist of things.

The G__ command tells the printer what to do (like move in a line).

The X__,Y__ and Z__ all represent coordinates.

And F, I, J, are going to dictate things like speed, offsets, or other parameters.

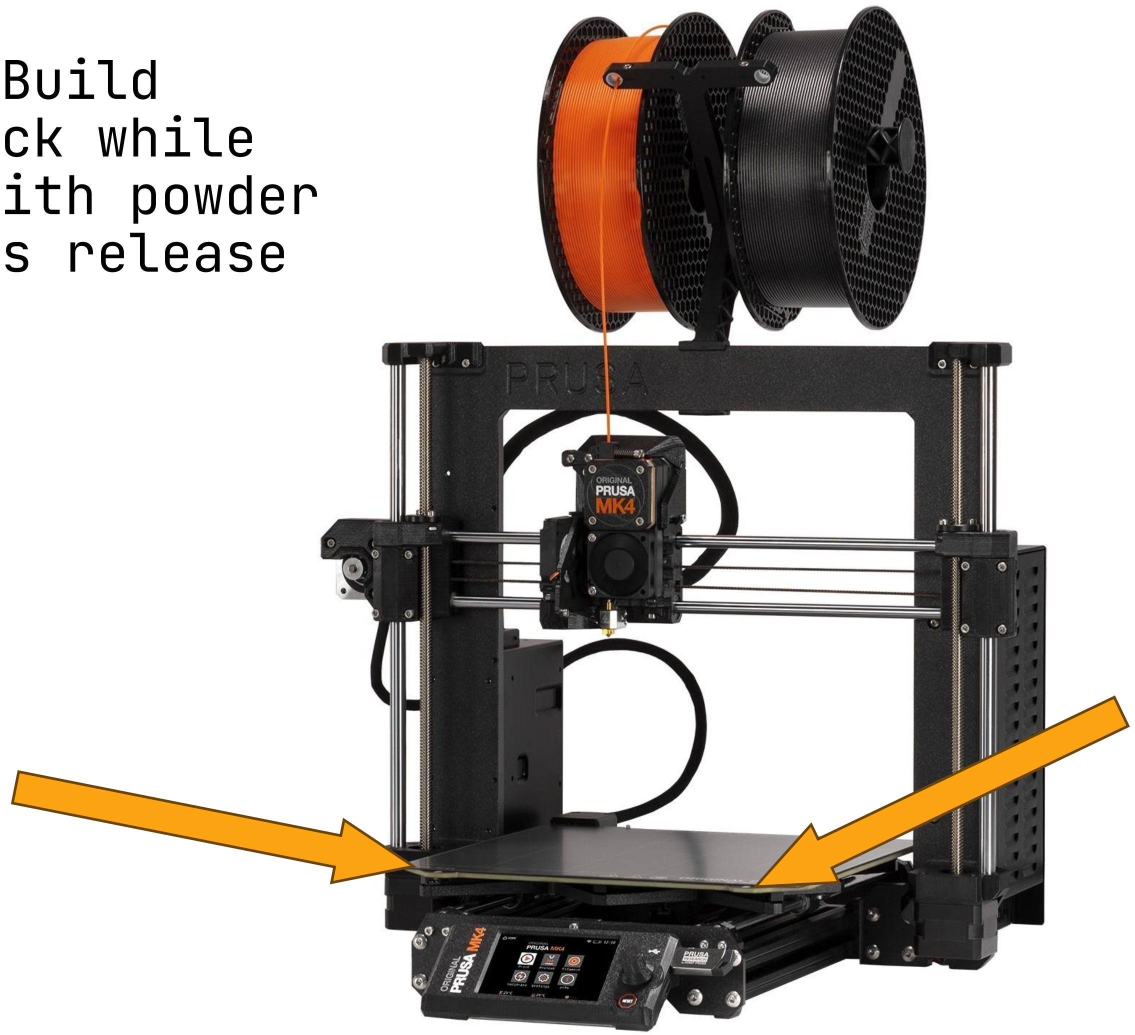
```
G01 Z-1.000000 F100.0(Penetrate)
G01 X247.951560 Y11.817060 Z-1.000000 F400.000000
G01 X247.951560 Y30.935930 Z-1.000000
G01 X106.963450 Y30.935930 Z-1.000000
G03 X106.587404 Y32.243414 Z-1.000000 I-7.576860 J-1.471361
G03 X105.974610 Y33.458880 Z-1.000000 I-6.445333 J-2.487300
G03 X104.697090 Y35.083261 Z-1.000000 I-7.601246 J-4.663564
G03 X103.141830 Y36.435630 Z-1.000000 I-10.087550 J-10.030472
G03 X102.969400 Y38.107779 Z-1.000000 I-20.252028 J-1.243405
G03 X102.369430 Y39.685740 Z-1.000000 I-3.842423 J-0.557919
G03 X100.419761 Y41.664361 Z-1.000000 I-6.181245 J-4.140917
G02 X98.333794 Y43.482560 Z-1.000000 I7.045018 J10.188229
G02 X95.783544 Y47.017541 Z-1.000000 I9.647185 J9.647199
G02 X94.101654 Y51.024620 Z-1.000000 I28.957871 J14.510988
G03 X92.872672 Y54.561719 Z-1.000000 I-340.631289 J-116.371936
```

https://howtomechatronics.com/tutorials/g-code-explained-list-of-most-important-g-code-commands/

00:30

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.



BUILD PLATE MATERIALS

Interestingly, we use plenty of materials to print on top of. In the past, it was very common to use glass, blue painters tape, or a special plastic called Kapton tape.

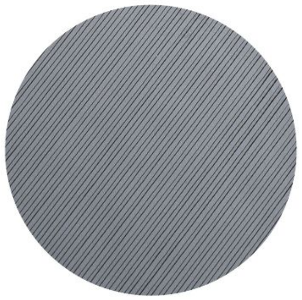
However, most modern build plates are actually removable and are most often made of steel.

Printing directly on steel is pretty useless, so we coat them in different materials.

BUILD PLATE MATERIALS (CONT.)

We generally have **two** main types of build plates.

Smooth build plates (PEI) are very useful for prints that don't want to stick. The smooth surface keeps prints stuck where they are.



Close-up of a printed object on the Smooth sheet

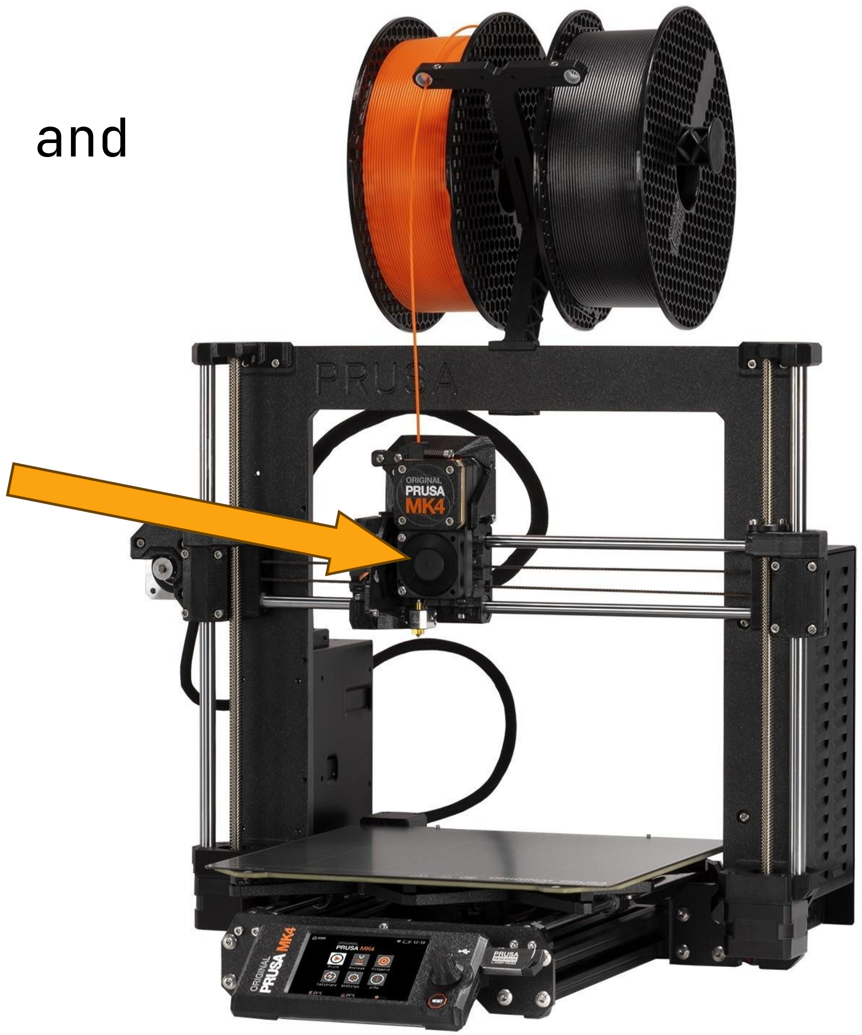
Close-up of a printed object on the Textured sheet

Rough build plates have a rough surface that is pretty hard to stick to. This is useful because some materials are so sticky they will damage a smooth plate in the process of removing them. They also give the bottom of our prints a cool texture.

00:36

HEAD

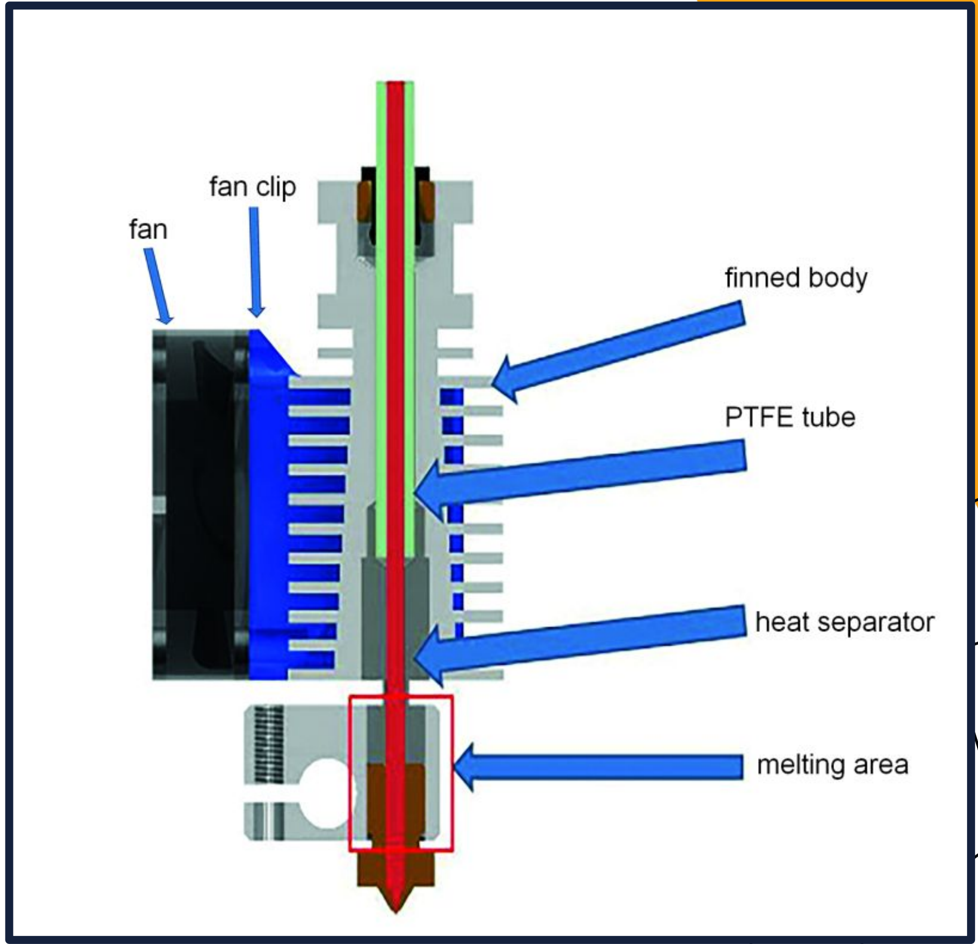
This is the most important part of the printer. It grabs the filament, heats it, and pushes it through the nozzle.



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.



https://www.open-electronics.org/3d4040-the-big-3d-printer-with-a-small-price/

00:39

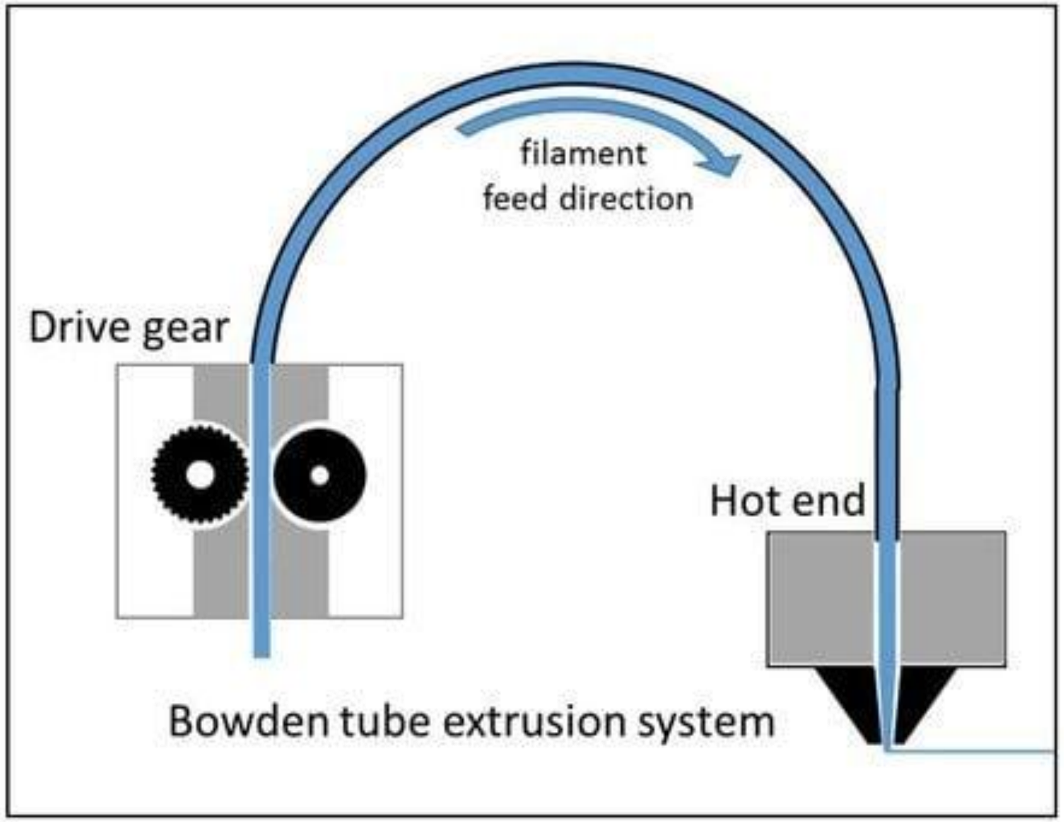
HEAD (CONTINUED)

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.

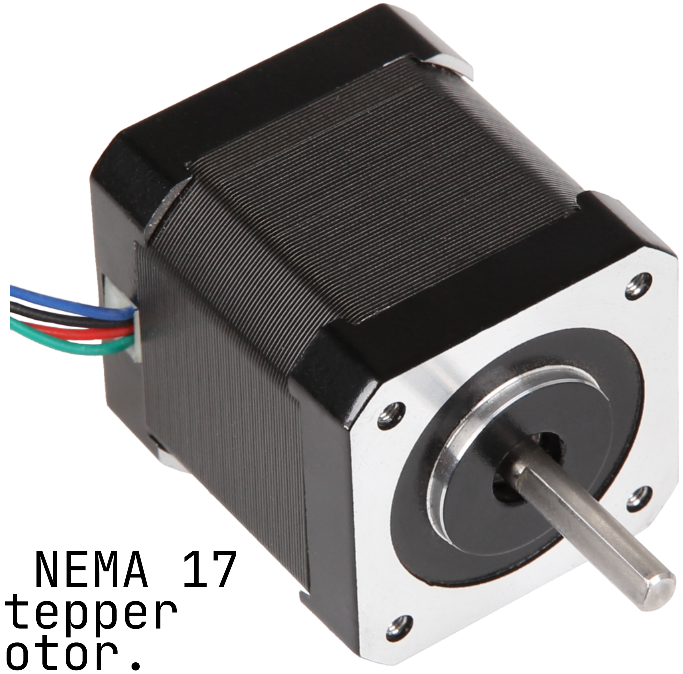
Premium and modern printers use direct drive (as shown on the previous slide). They pull the filament.



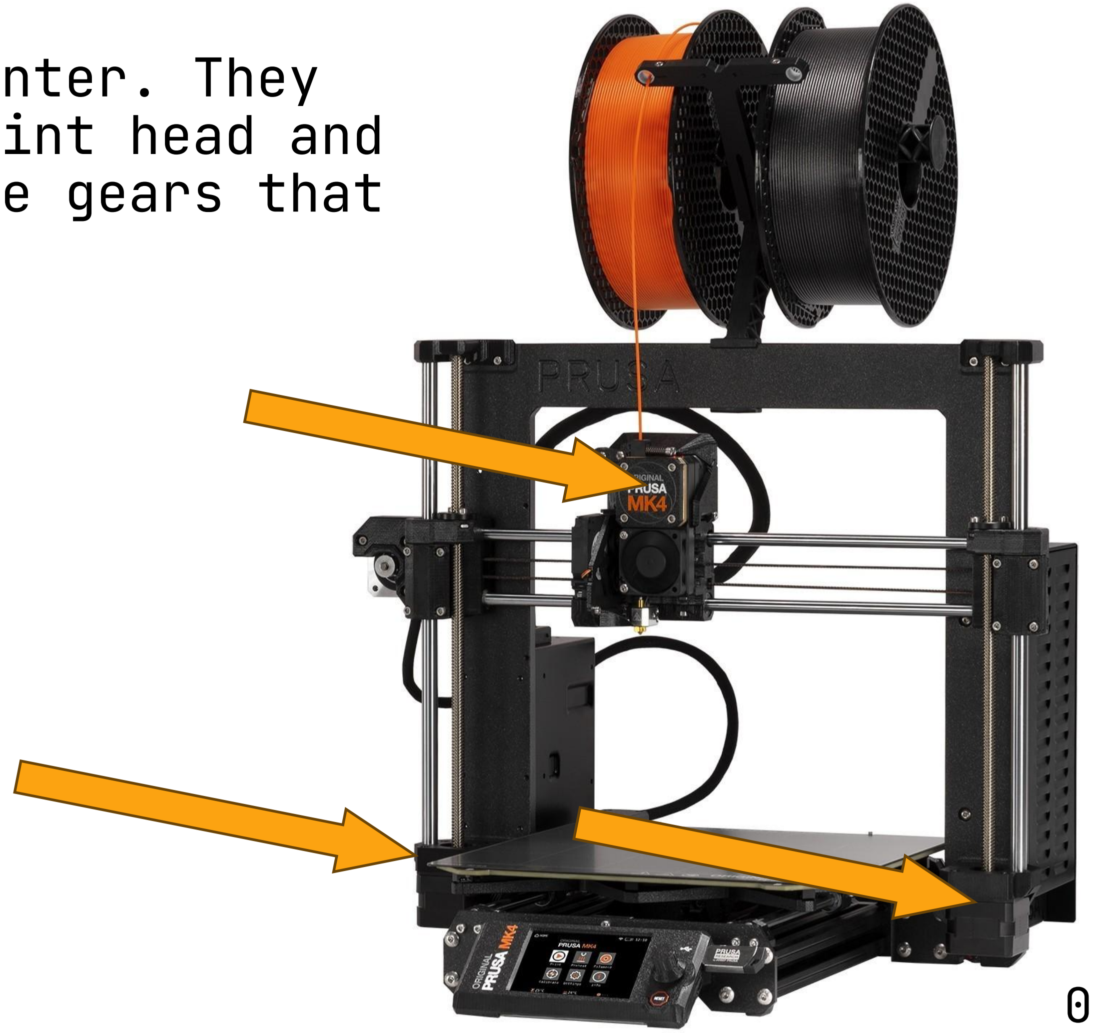
00:42

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.



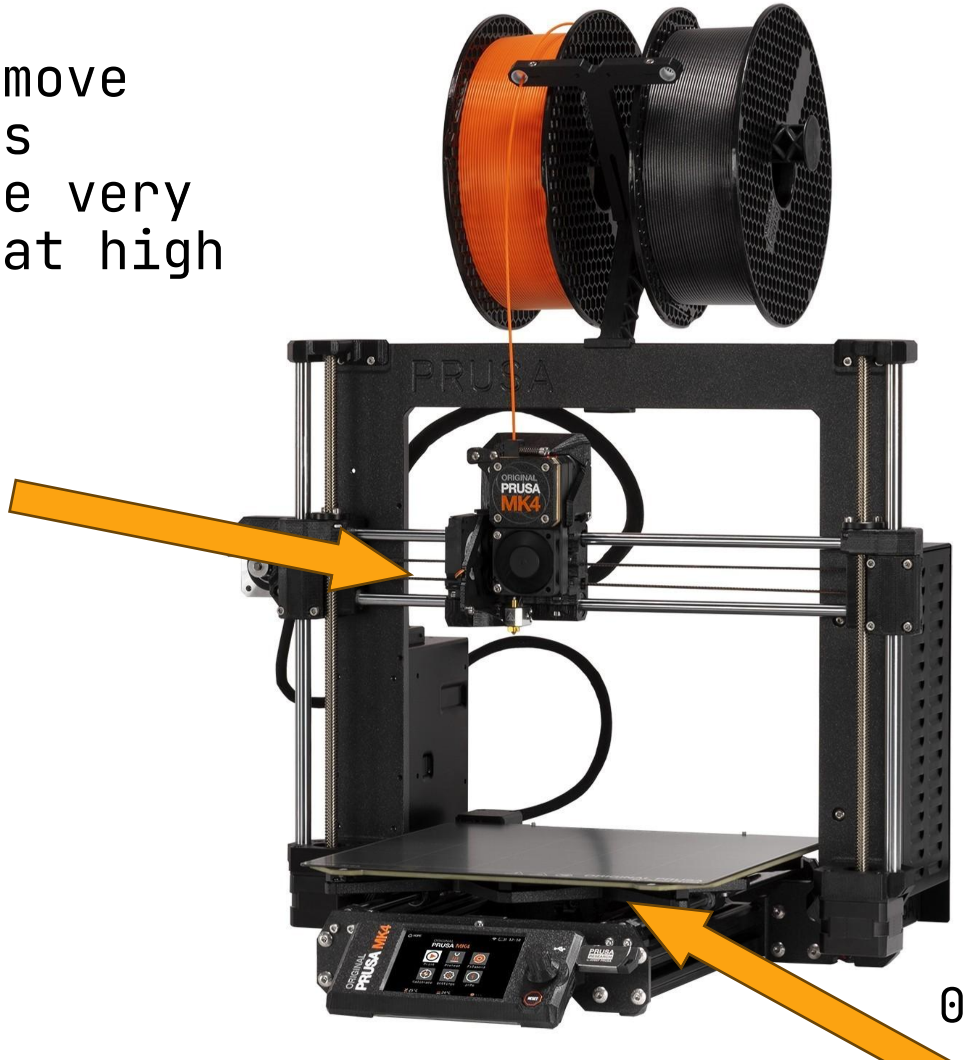
A NEMA 17 stepper motor.



00:44

BELTS AND PULLEYS

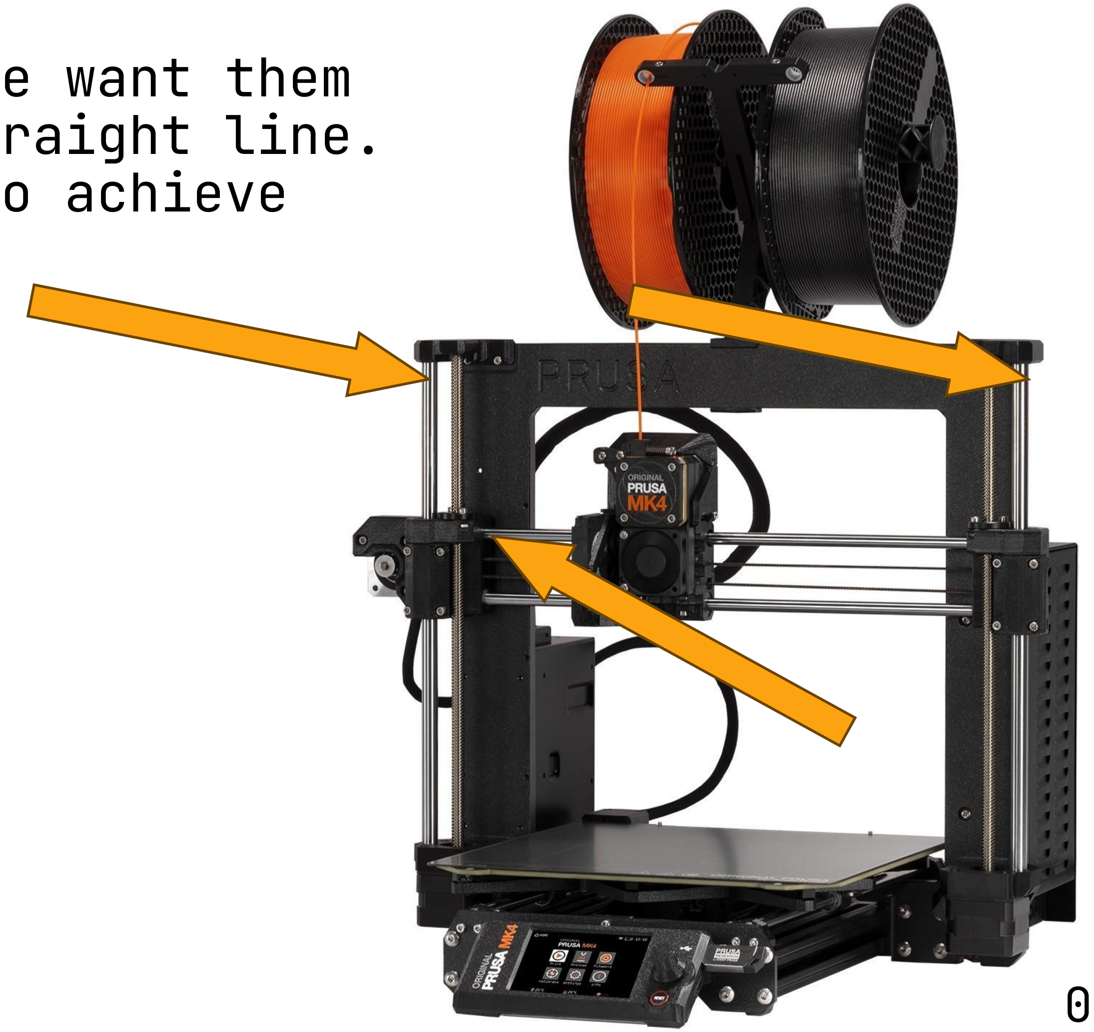
For parts of the printer that need to move rapidly, we often use belts and pulleys attached to the motors. They tend to be very lightweight and resistant to slipping at high speeds.



00:45

LINEAR RAILS & BEARINGS

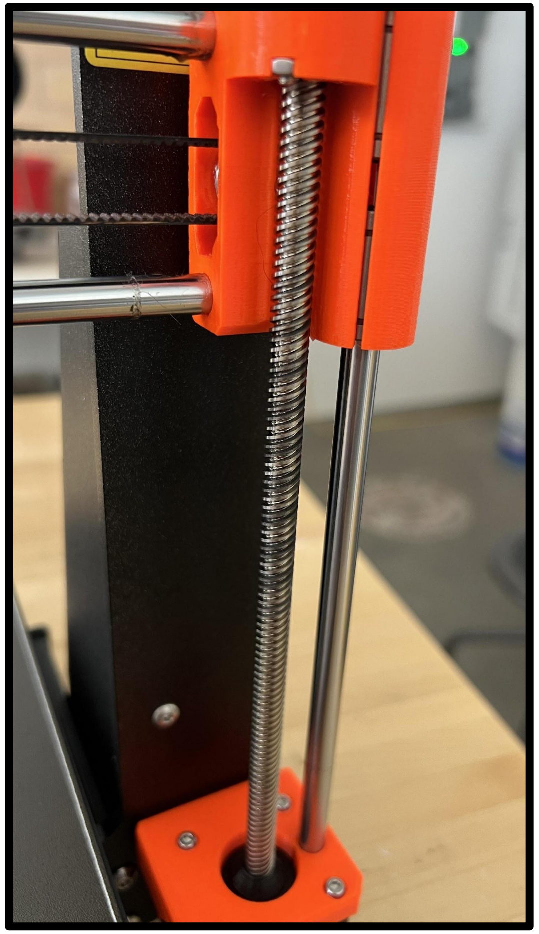
When we move our printer’s axes, we want them to be stable and move in a very straight line. We use linear rails and bearings to achieve this.

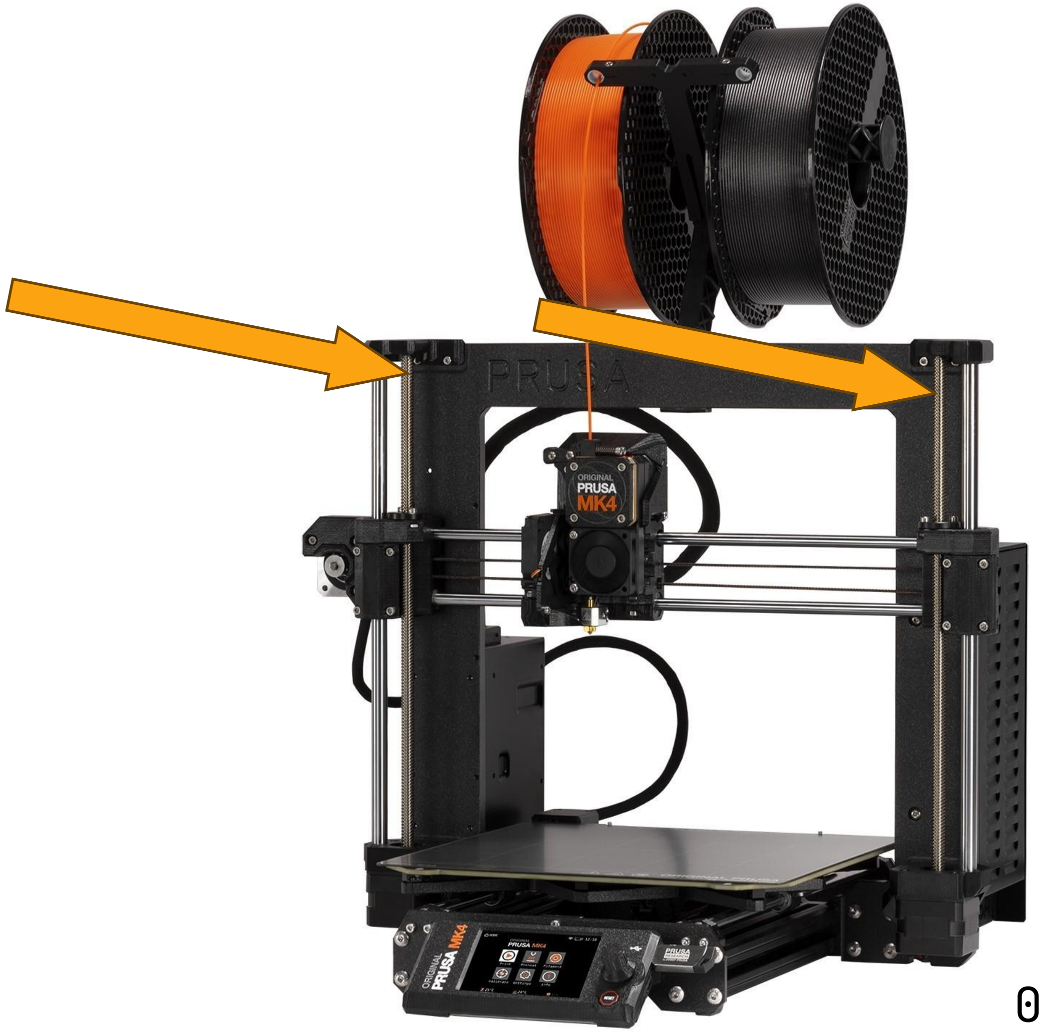


00:48

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.





00:50

KAHOOT TIME!

- ‘YEA 3D Printing Introduction’

- Begin break

01:00

SESSION 2

- Common printing materials
- Calibrating and Operating the printer

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).

00:00

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"

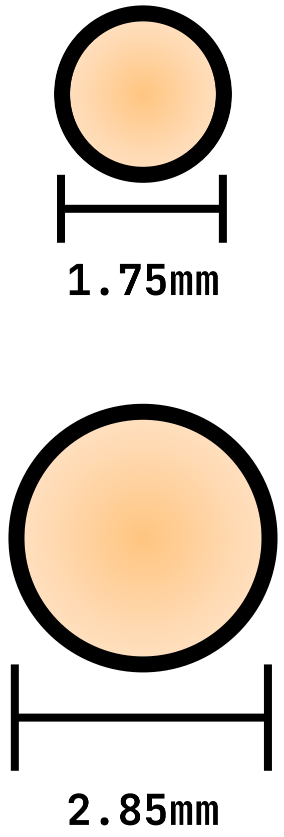
00:2

FILAMENT DIAMETERS

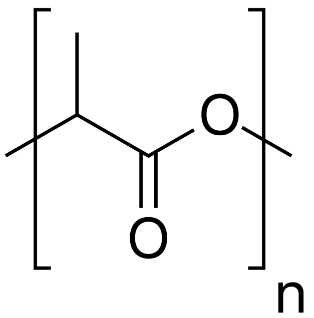
A quick note about diameters of filaments used in FDM printers.

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



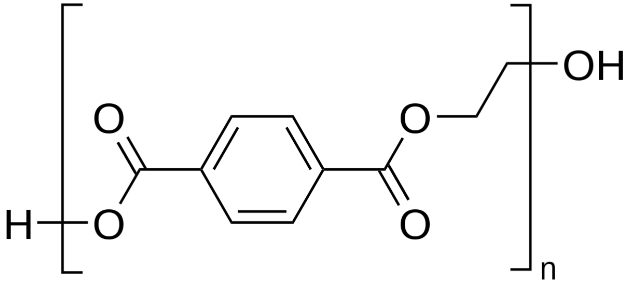
PLA is the most commonly used plastic for FDM printing. This is due to many factors such as:

- 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.

00:5

PETG - POLYETHYLENE TEREPHTHALATE GLYCOL



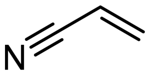
PETG is the second most popular plastic behind PLA for 3d printers. It prints at a higher temperature but is most notable for being less brittle. This makes it very useful for thin prints for parts you need to bend slightly during use.

00:6

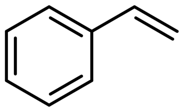
FLEXIBLE

There are many flexible filaments that we might want to 3d print. However, they can be some of the most challenging to print. Because the filament is prone to bending, flexible materials often get stuck inside the machine. A direct drive extruder is more or less REQUIRED, as a Bowden system will get gummed up almost immediately.

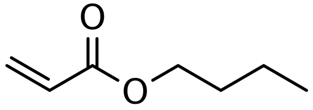
ASA - ACRYLONITRILE STYRENE ACRYLATE



acrylonitrile



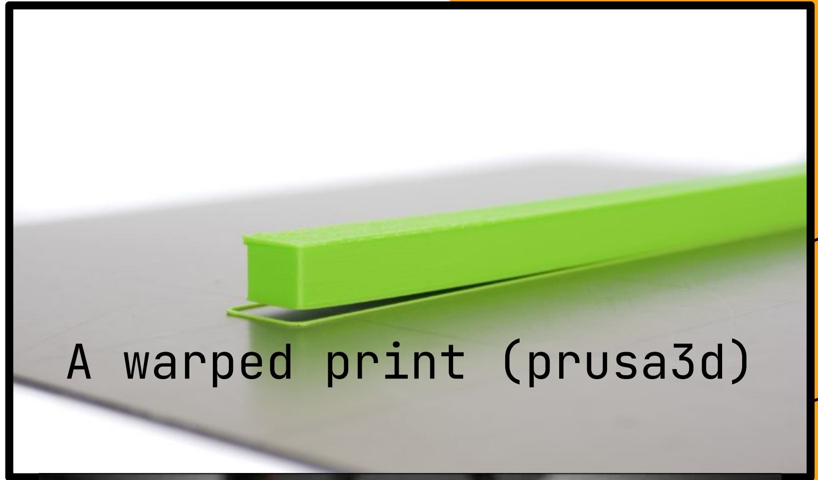
styrene



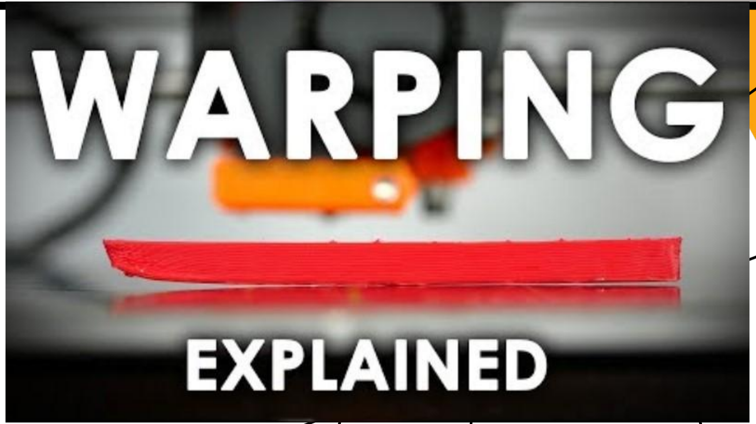
butyl acrylate

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)



A warped print (prusa3d)



00:10

SPECIALTY ADDITIVES

It is very common to add an assortment of additional substances to change the properties of a filament. For example, many places sell a PLA+, PLA pro, or PLA Impact material. This is usually PLA filament with some kind of softener added to make the material less brittle.

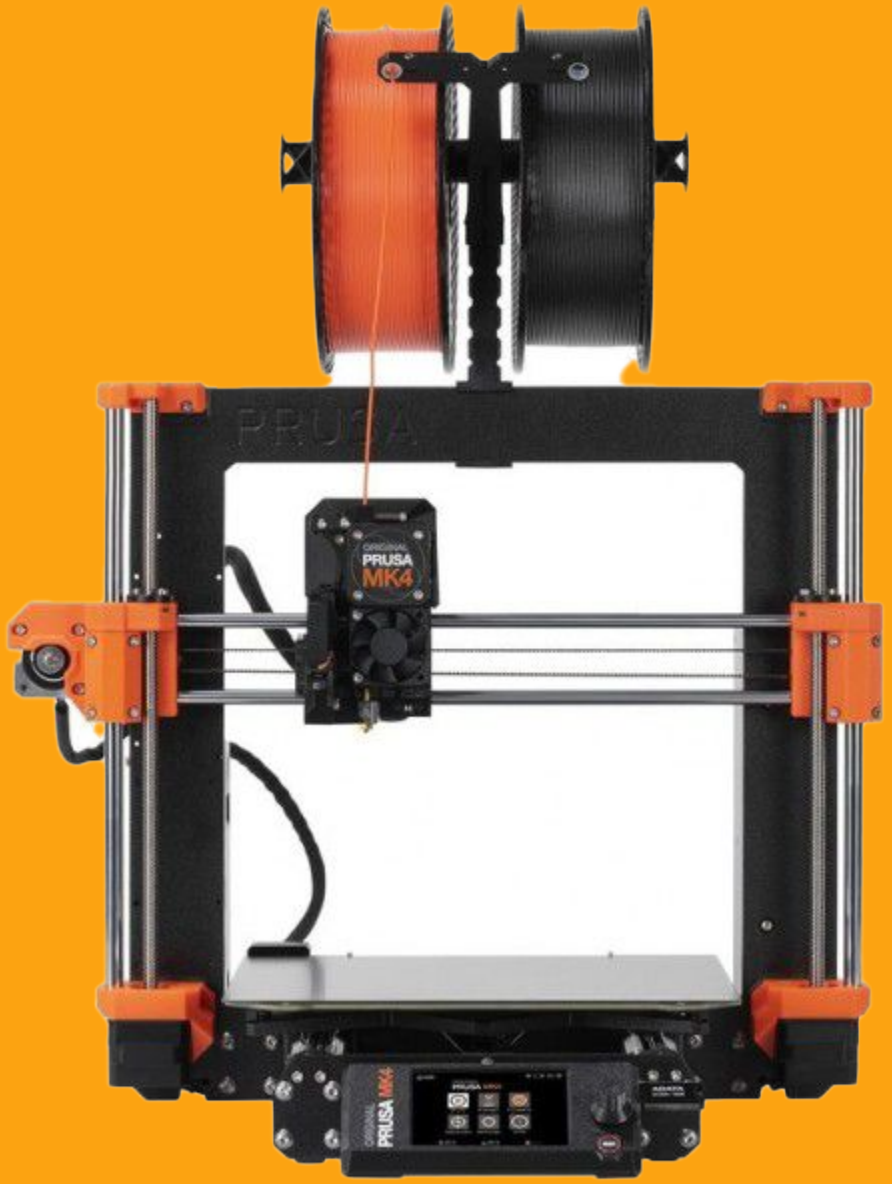
For example, ASA (discussed on the last slide) is very similar to ABS (Legos are ABS!) but has stuff added to make it better for 3d printing.

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.



OPERATING THE PRINTERS

How do we actually interact with the printers, how do they behave, and how can we understand what they are doing?

INTERNET CONNECTIVITY

One of the most common features of new hobbyist printers is the ability to connect to the internet. This means you can start, stop, and monitor prints on your phone or computer.

However, you can also interact with printers entirely locally. By using flash drives and SD cards, you can transfer files from your computer's slicer to the printer with no internet connection required.

00:30

CALIBRATION

Notably, the printer doesn't know where it is in space when we first start it up. That's why they always perform a 'homing' operation before a print and before any important actions.

Homing: the printer returns to the 'home location'. This is where the printer has its X, Y, and Z coordinates set to '0'.

It usually knows it has reached zero by hitting physical switches that tell the printer to stop, called 'limit switches'.

We'll walk through a homing cycle now.

Limit Switch

00:35

MATERIAL CHANGE

Most printers can only have one filament loaded at a time, meaning we have to unload the current filament to print with other materials.

This process involves first heating up the nozzle, then ejecting the old filament.

Once ejected, the printer heats to the temperature of the new material (which may be higher or lower) before loading the new material type.

We'll walk through a material change now.

00:40

NOZZLE CHANGE/TYPES

During our overview of the printer, we mentioned that filament is squeezed through a hot ‘nozzle’.

Nozzles come in all kinds of sizes and materials for different use cases. They also wear out and need to be occasionally replaced.

Most common is a 0.4mm brass nozzle □

There are also nozzles made of stainless steel, with unique coating like Teflon (common on nonstick pans), or even materials like diamond on the tips.

We’ll walk through a nozzle change now.

New and Old Brass Nozzles

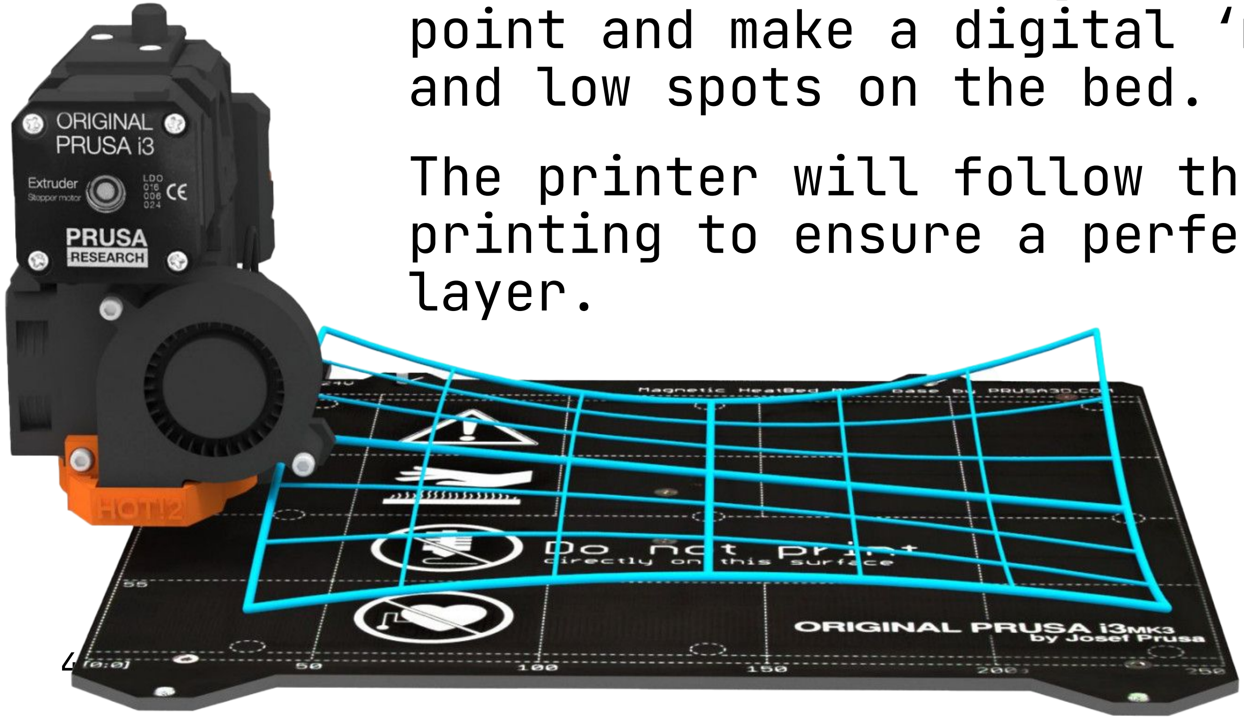
00:50

(MESH) BED LEVELING

For a print to be successful, it is vitally important that our first layer be consistently thick. However, most print beds aren't completely flat.

We can check the height of the bed and several point and make a digital 'mesh' of the high and low spots on the bed.

The printer will follow these contours while printing to ensure a perfectly flat first layer.



LET'S SEE IT IN ACTION

Run a quick print (Benchy or otherwise), describe features of startup, calibration, brim, infill, etc.

01:00

SOLIDWORKS

This is the software we'll use to make a model. It contains tools to create and manipulate shapes and bodies.

WHAT IS SOLIDWORKS

Solidworks is a parametric CAD software. Let’s break down some of those words.

CAD – Computer aided design

This describes the modern method of designing digitally with computerized design tools (rather than drawing stuff on paper)

Parametric Design

Parametric design focuses on the relation between lines and shapes rather than absolute values. This means we can change the length or angle of something, and the rest of the model will update accordingly.

MAKING A PENCIL HOLDER

To start modelling, we’re going to make a pencil holder.

One of the most important things right now is getting you used to the tools.

This part doesn’t need to be good or usable at all, we’re just having some fun.

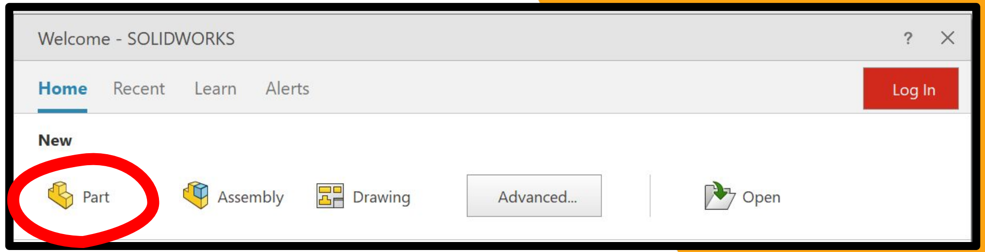
OPENING SOLIDWORKS

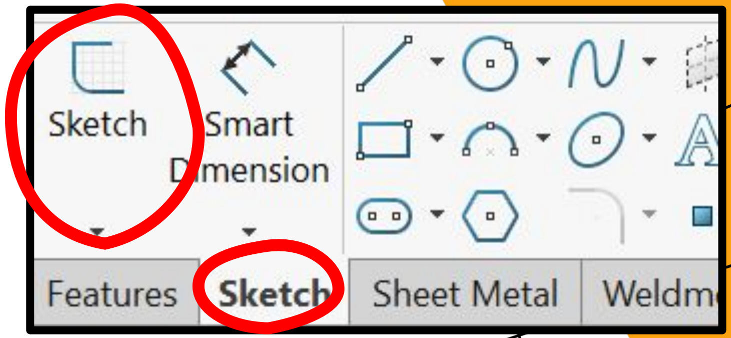
When we first open Solidworks, we're prompted with a file creation window.

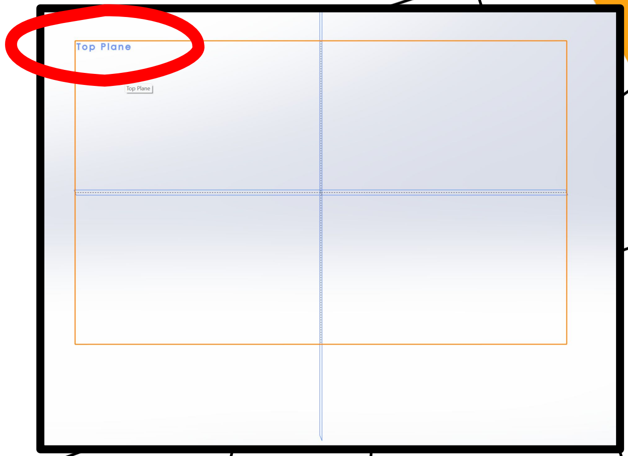
Create a new Part.

Under the sketch tab, create a new sketch.

We need to select a plane to draw on, I would choose 'Top Plane' but any plane will work.







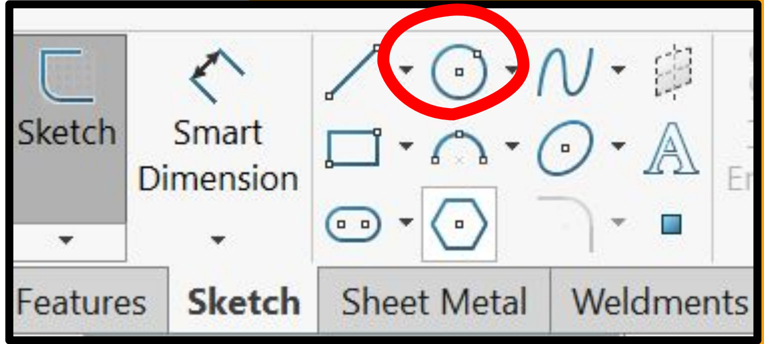
DRAW A CIRCLE

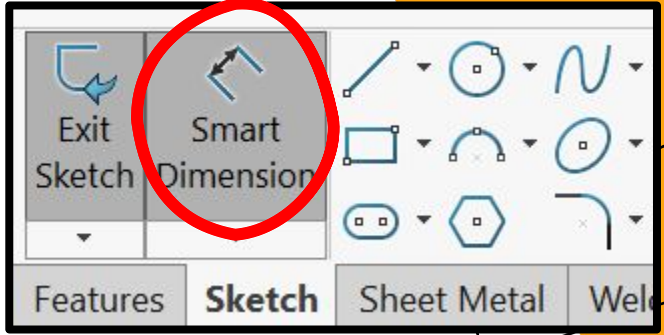
Select the circle tool by either clicking the button at the top under 'Sketch' or holding down right click and dragging downward before releasing.

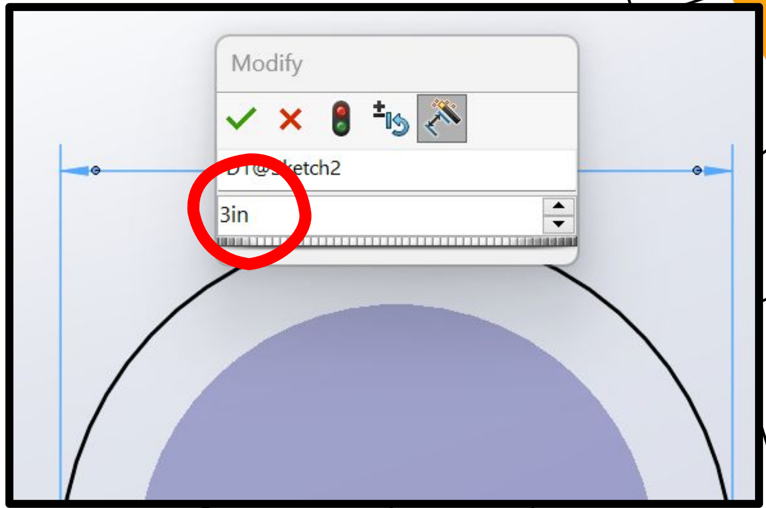
First select the center and the outside. Don't worry about how big you made it.

Once drawn, selecting 'Smart Dimension' will let us say how big we want the circle.

Click the outside of the circle and type in your desired diameter. I'll make it three inches across.



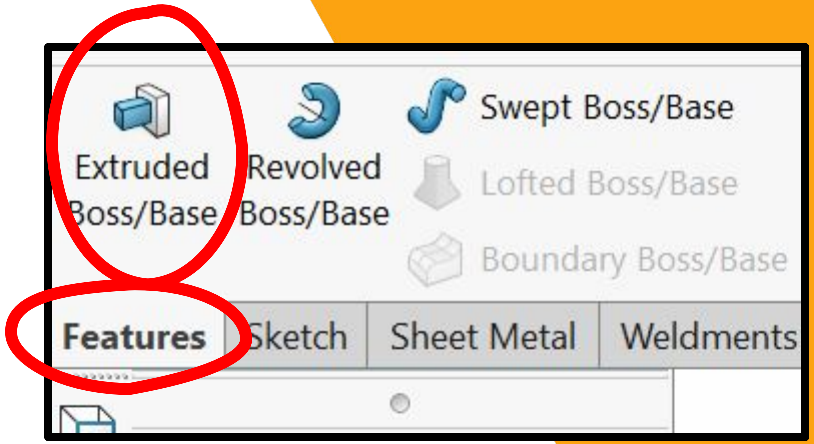


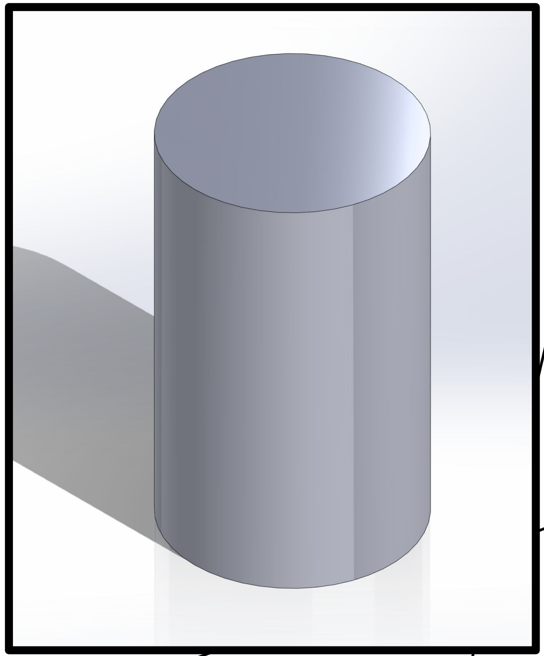


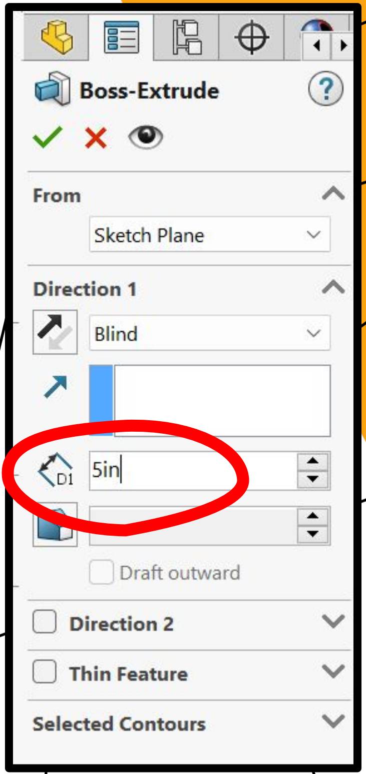
EXTRUDING

Once we have our circle, we can extrude it into a 3D object.

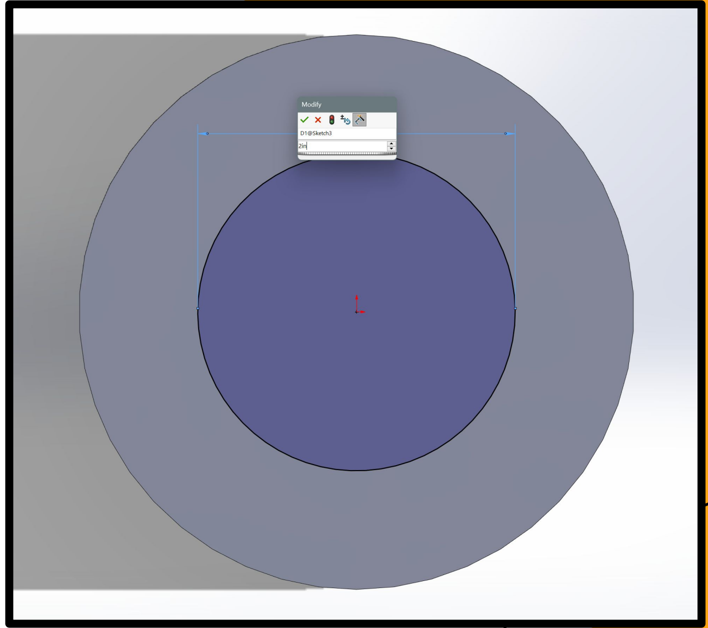
Go to the 'features' tab and the 'Extruded Boss/Base' tool. This will let you select your circle and type in how far you want to extrude it. I'm going to make it five inches tall.







MAKING A HOLE

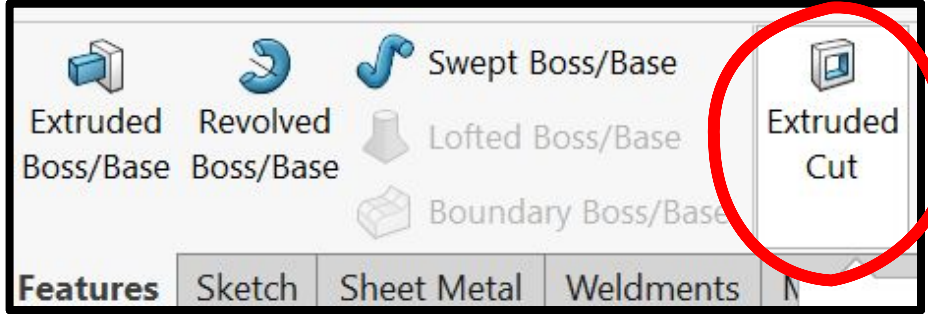


Now our cylinder doesn't really look like a pencil holder yet. In order to finish it, we'll need to cut a hole in it.

Going back to the sketch tab, we can create a new sketch and click one of the circular faces on our body. This will allow us to draw on the top of our part.

Again we'll draw a circle but a bit smaller. I'll make mine two inches across.

Now, back in the 'features' tab we will select the extruded cut tool.

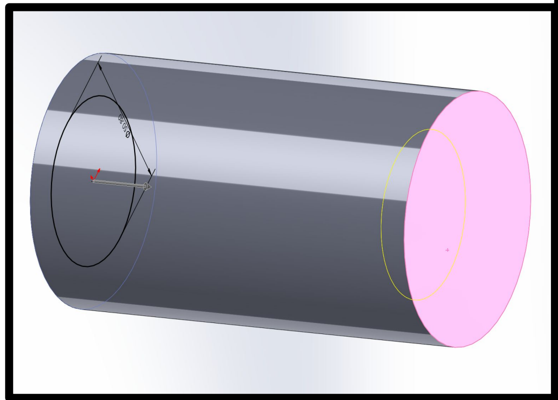


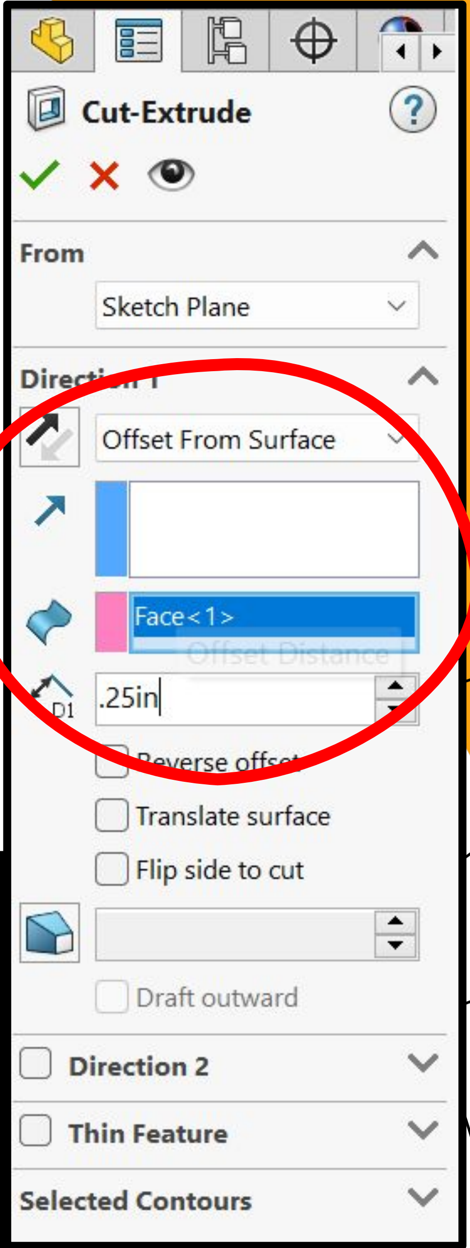
MAKING A HOLE (CONT.)

Once we pull up the extruded cut context menu, we have several option in the dropdown.

If we wanted the hole to go all the way through the part, we *could* select ‘through all’.

However, we want to leave some thickness at the bottom of the part, so our pencils don’t fall out. To do this we can use ‘Offset From Surface’, select the bottom face, and tell it to leave .25” from the bottom.



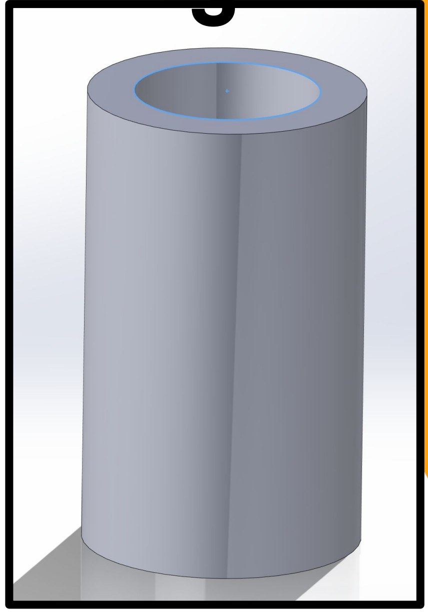


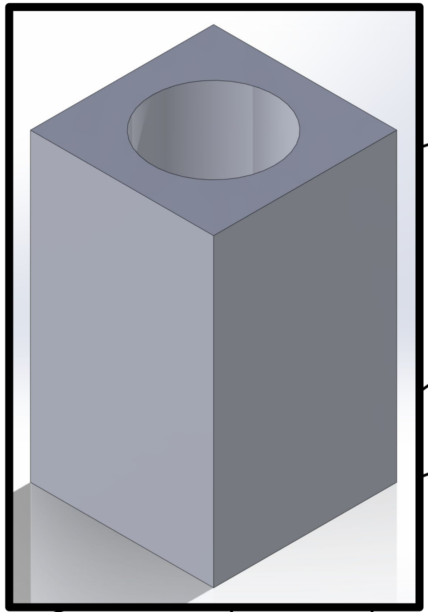
MAKING IT SQUARE (ON YOUR OWN)

What if we **actually** wanted the outside of the pencil holder to be square, rather than round?

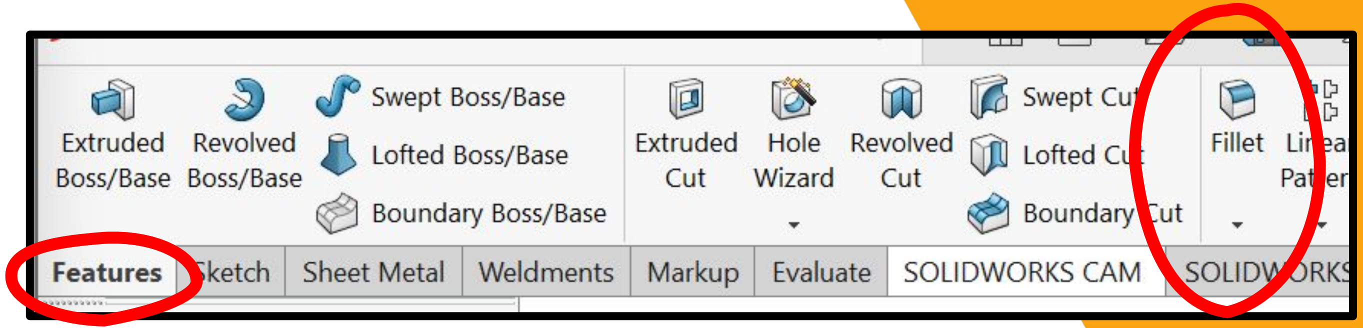
We can go back and change the initial sketch!

In your timeline, right click the first extrusion feature and click the 'edit sketch' button. Click and delete the circle and replace it with a square. This is what makes parametric design software so powerful. We can go back and change dimensions and other features without remaking the entire part.





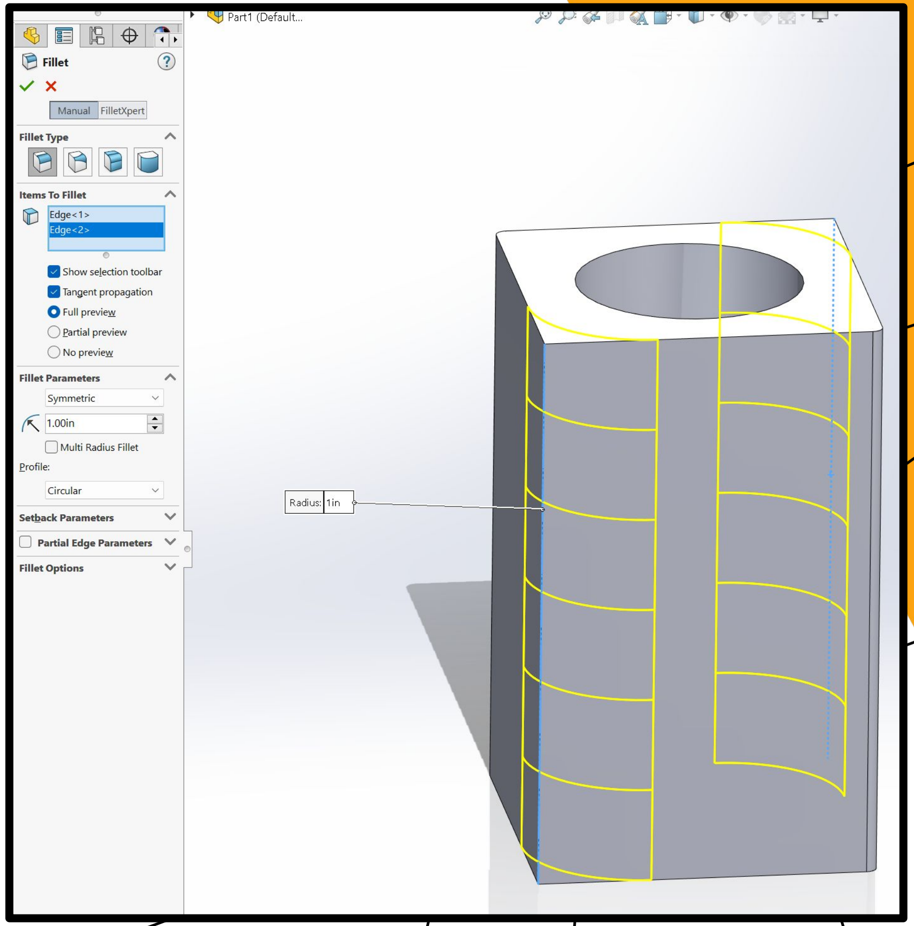
MAKE IT PRETTY



Let's be honest, our pencil holder doesn't really look that great. How can we make it look nicer?

Fillets!

Under the features tab, click the 'Fillet' button and select some edges. You can tell it the radius.



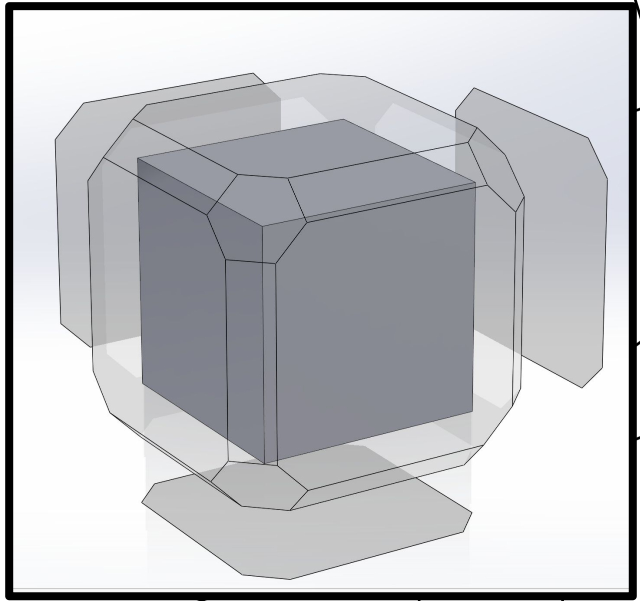
Try giving different values to different edges on your pencil holder.

MOVING AROUND

One of the hardest parts of learning a new CAD software is getting used to moving around in 3d space.

- (+/-) Scroll wheel to zoom in and out.
- (□) Middle Mouse Button and drag to orbit.
- (⇔⇕) Ctrl + MMB to pan.

If you ever get lost, press   to pull up orientation view. Simply click a face to orient your camera accordingly.



SAVING AND EXPORTING

First, always save your part as a .SLDPT file. This is what you'll open next time in Solidworks to change or edit your part.

Anything you want to print, you'll need to export as an .STL file. This changes your part to a mesh and lets PrusaSlicer read it.

WHAT DO I NAME STUFF?

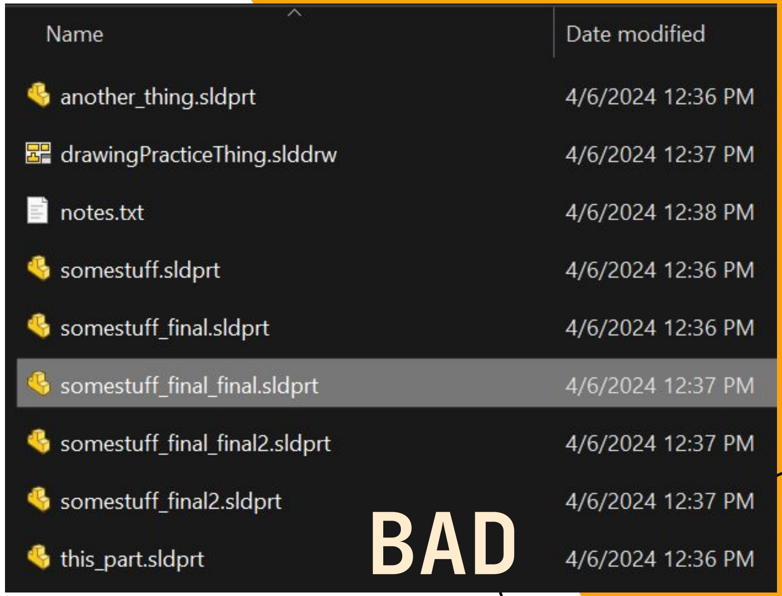
Please name your files in the format 'LastFirst_____' with the underlines the part name.

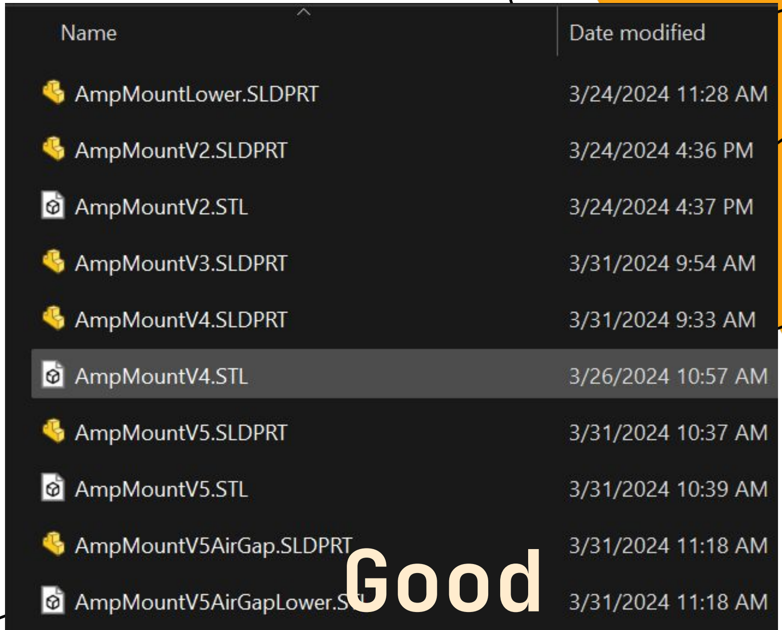
Keep file names consistent and use version numbers. Don't name things **'final'**. We back ourselves into the wall if we change it again (which we do a lot).

What would we name it next?

'finalbutreallythistime'??

Note: These lab computers wipe every time we log out. You **NEED** to save stuff somewhere safe like a thumb drive or Google Drive.

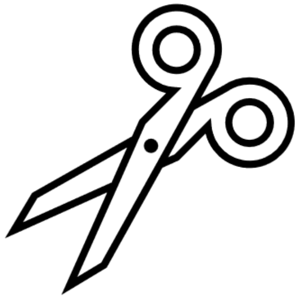




PRUSA SLICER

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.

ALTERNATIVE SLICERS

In this class, we'll be using PrusaSlicer. However, there are many slicers out there to use (and all the popular ones are free).

Consider Ultimaker's Cura slicer, which has been the most popular for years but has been less common lately.

There is also Bambulab's 'Bambu Studio' which is a copy of PrusaSlicer with a custom layout and some small changes.

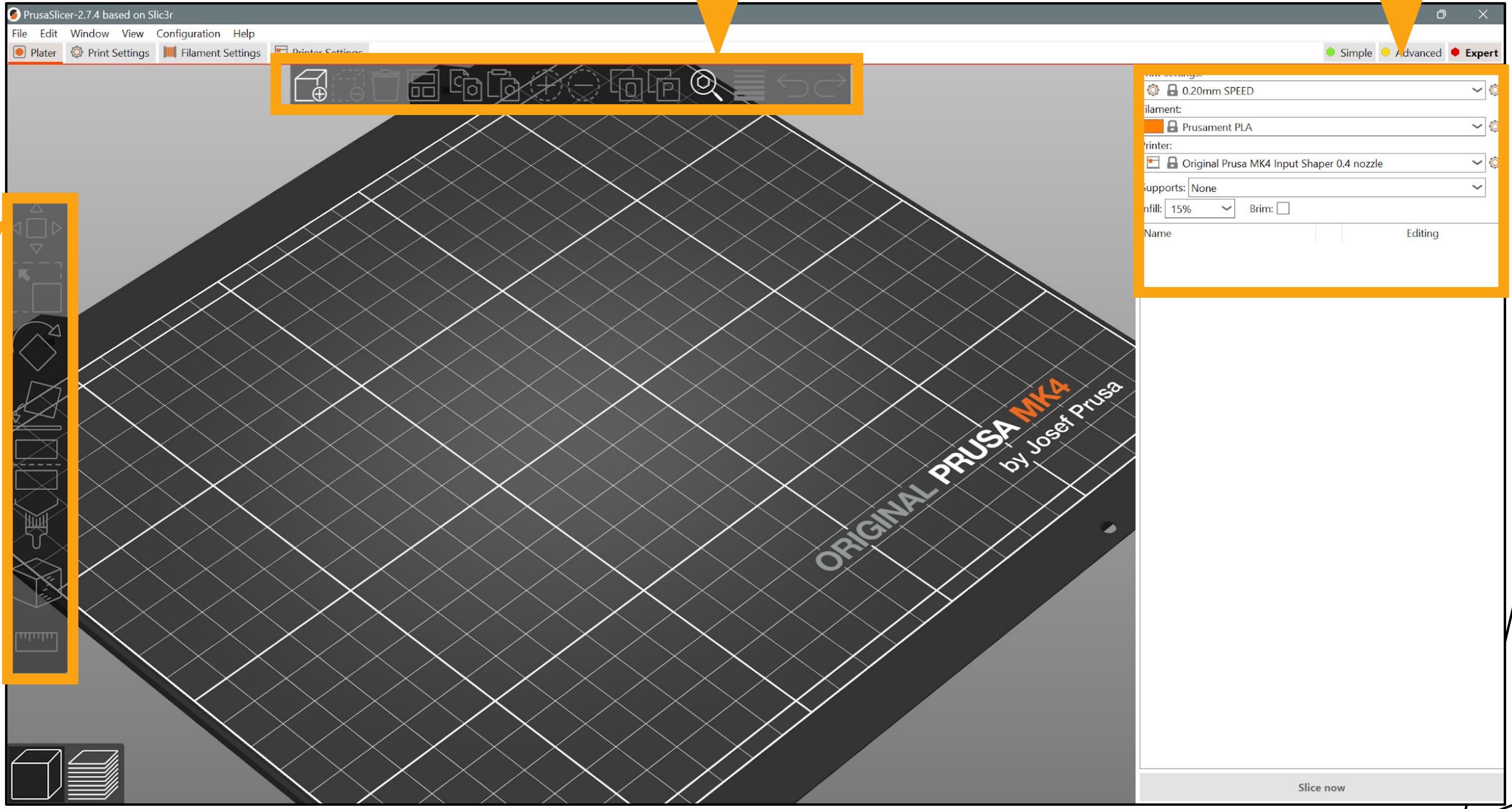
That's not even mentioning slicers for other types of 3d printer like SLS or SLA technologies.

OVERVIEW

Plate Controls

Slice Settings

Part & Object Controls



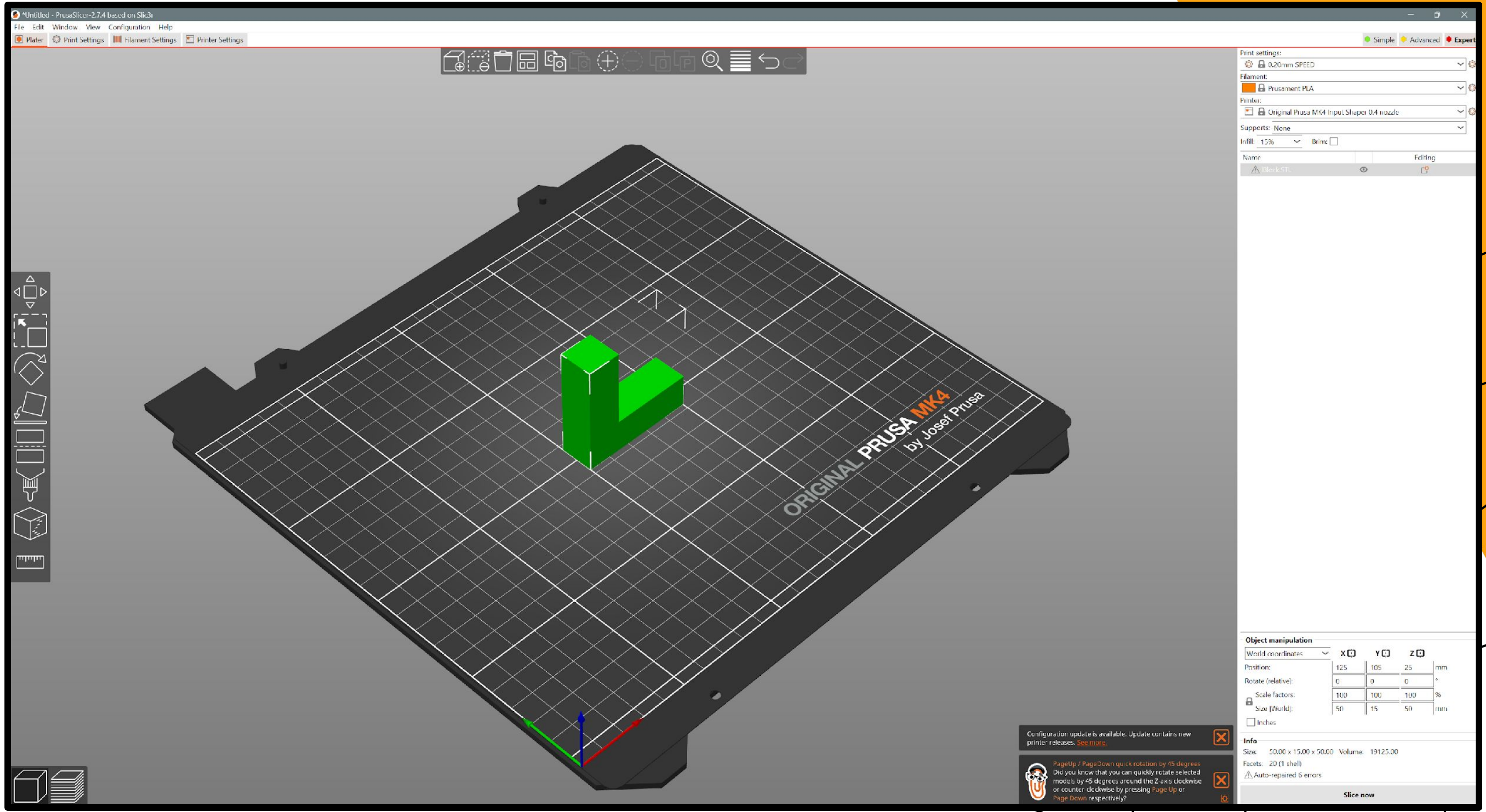
IMPORT A PART

First, import your part (.stl file).

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

OR

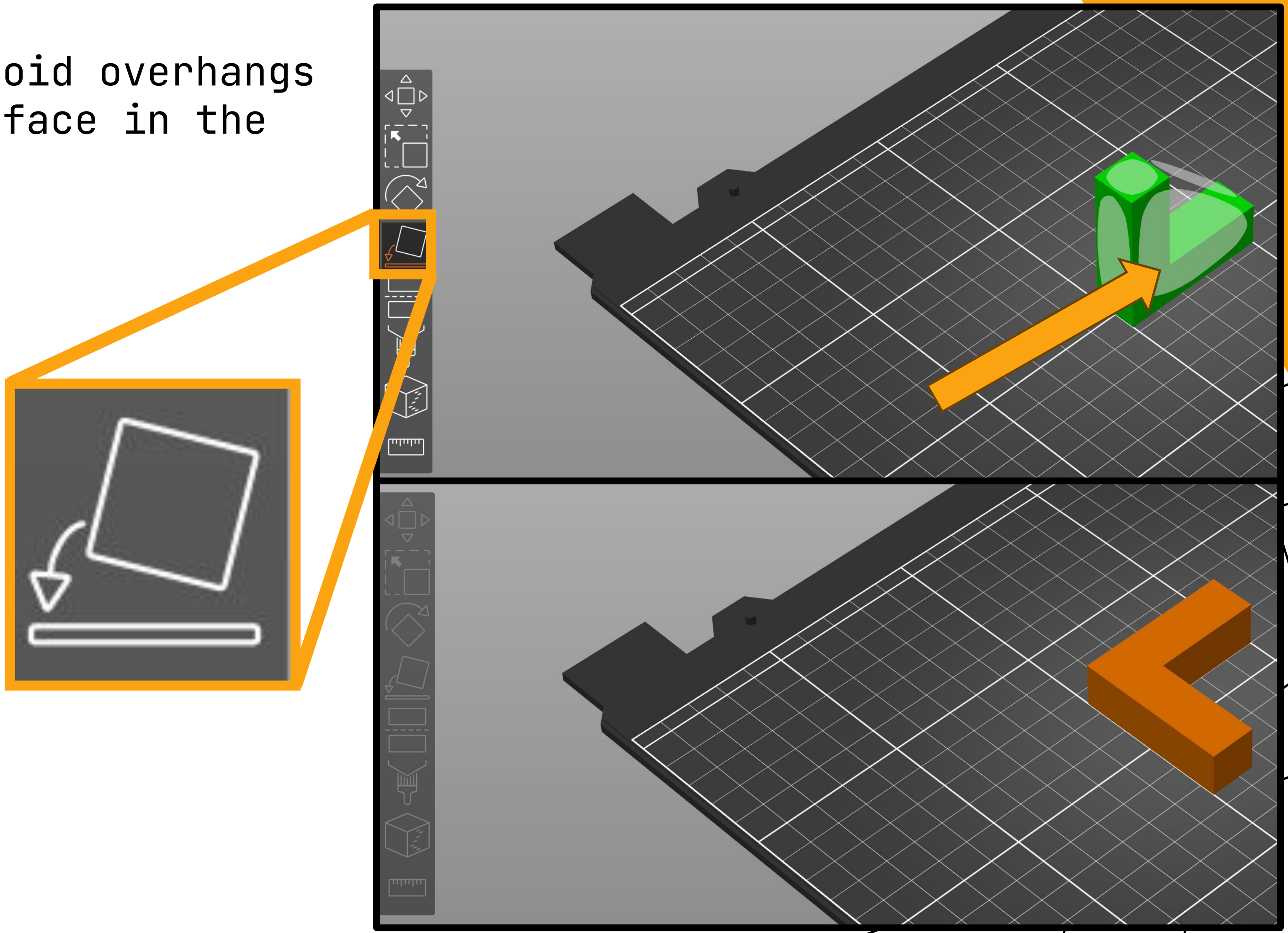
File > Import > Import STL/3MF…



ORIENT YOUR PART

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

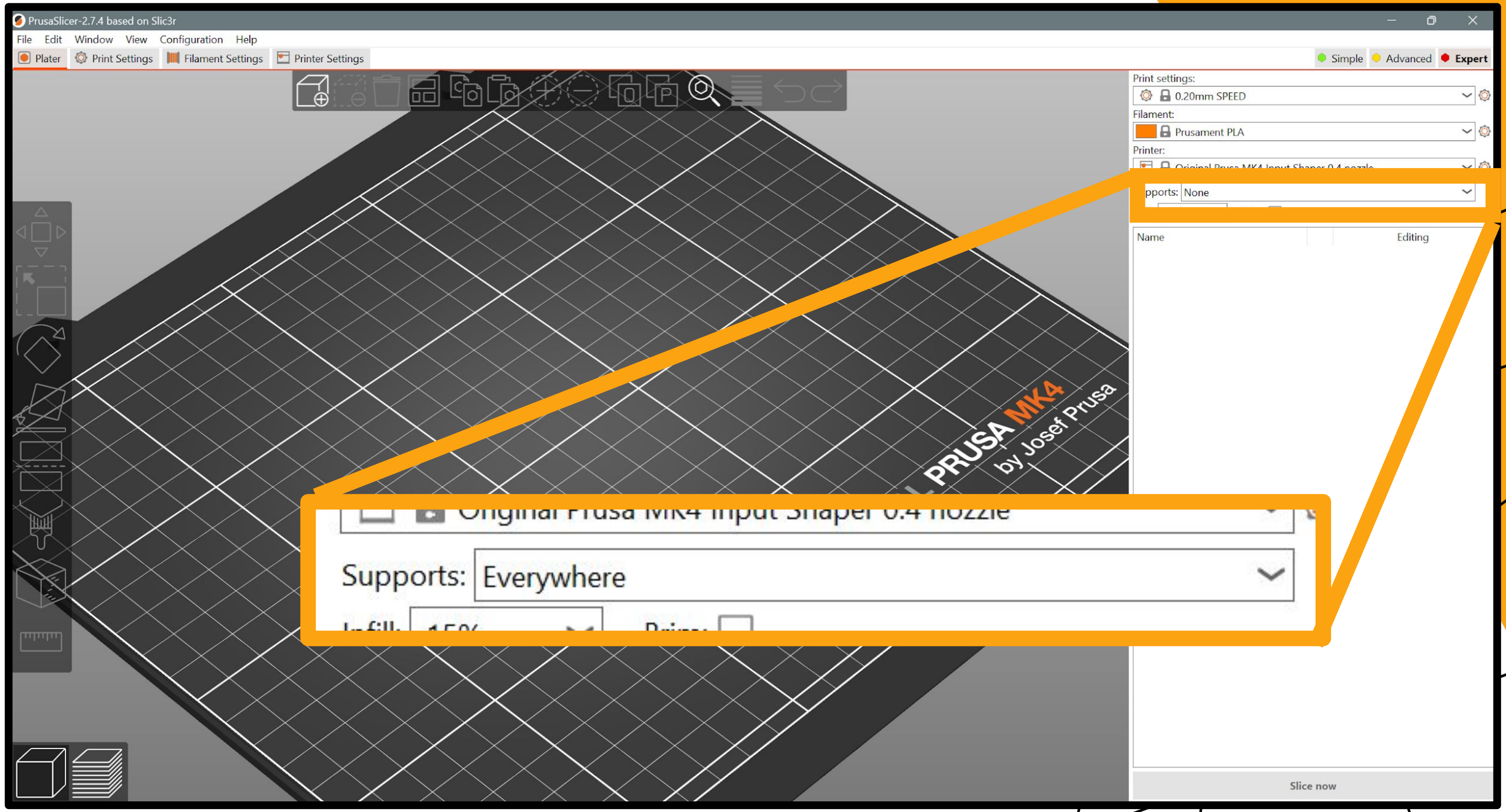
First select your part.
Then, using the ‘lay flat’ tool, click a face to align it to the plate.



ADDING SUPPORT (IF NEEDED)

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

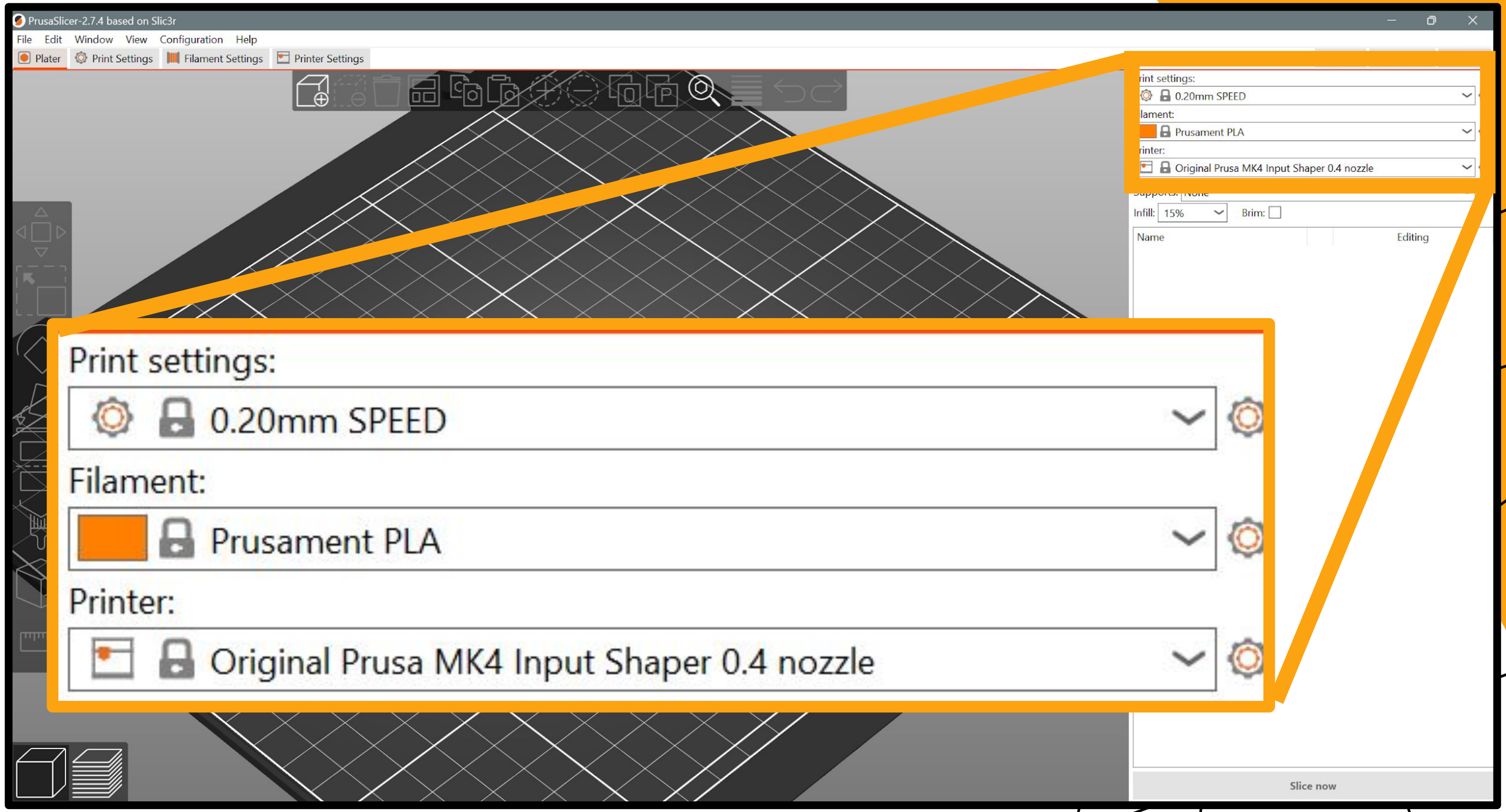
On more complicated parts, we can add them manually if necessary.



FINAL SETUP

Select the printer, filament, and print settings profile.

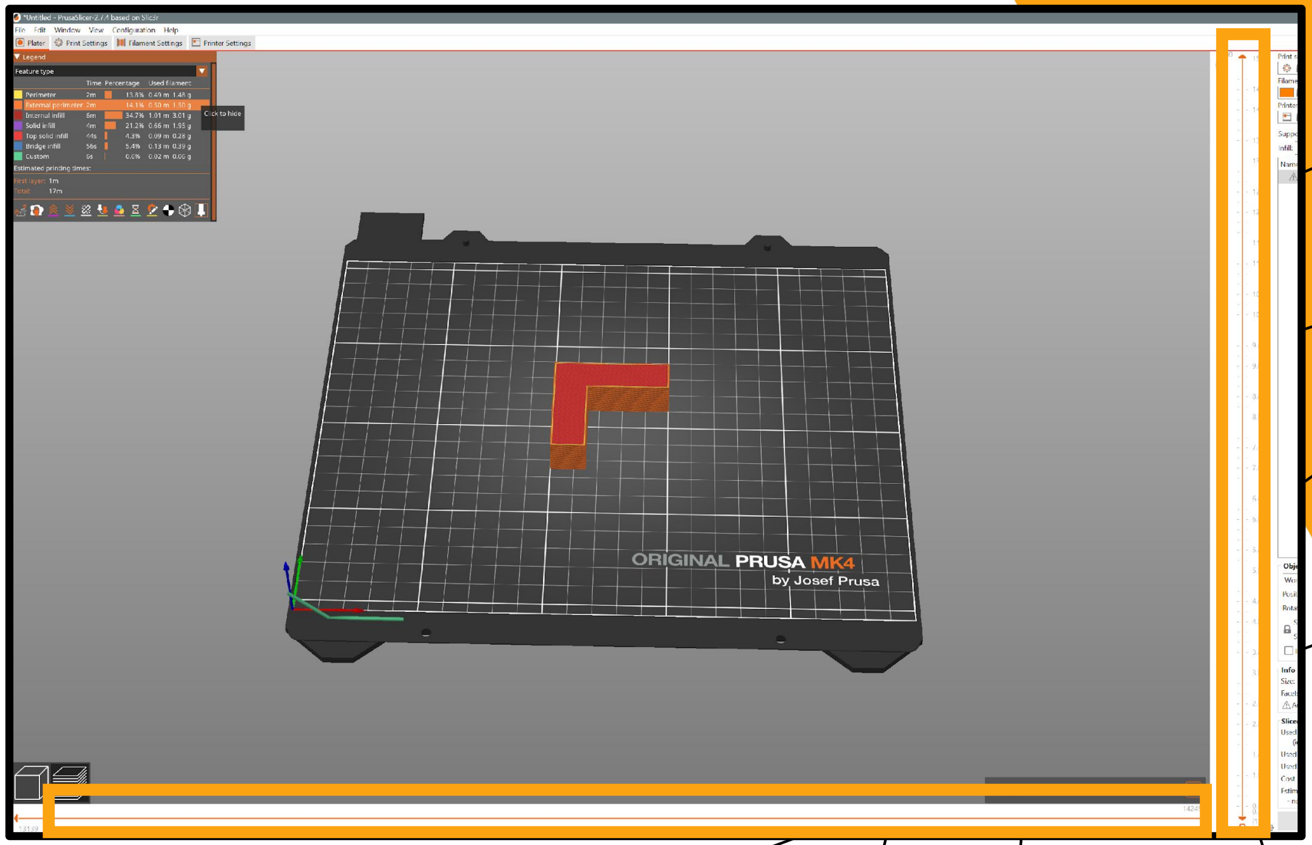
Note the print settings change depending on the printer.



SLICING

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.

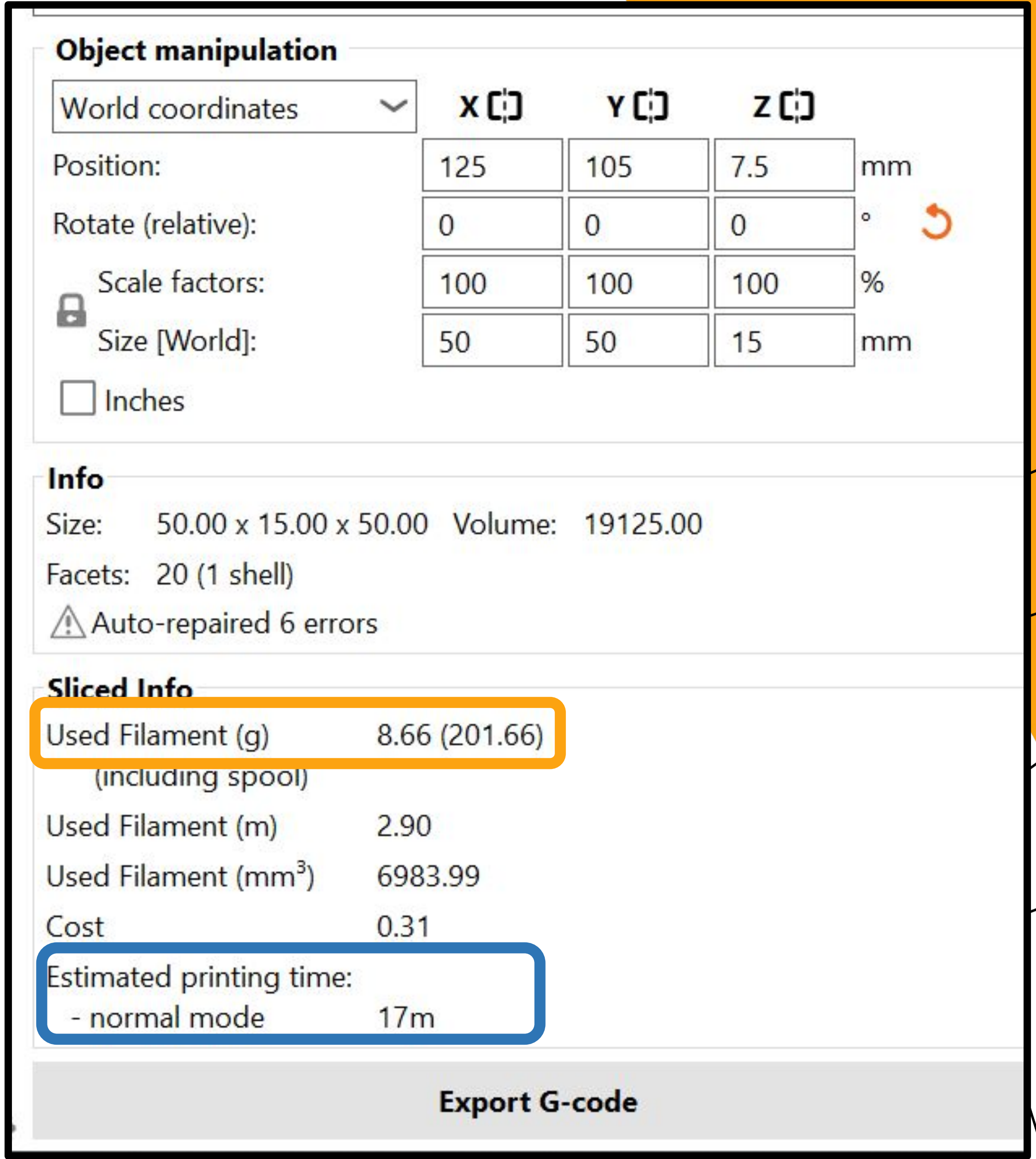


SLICING

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.

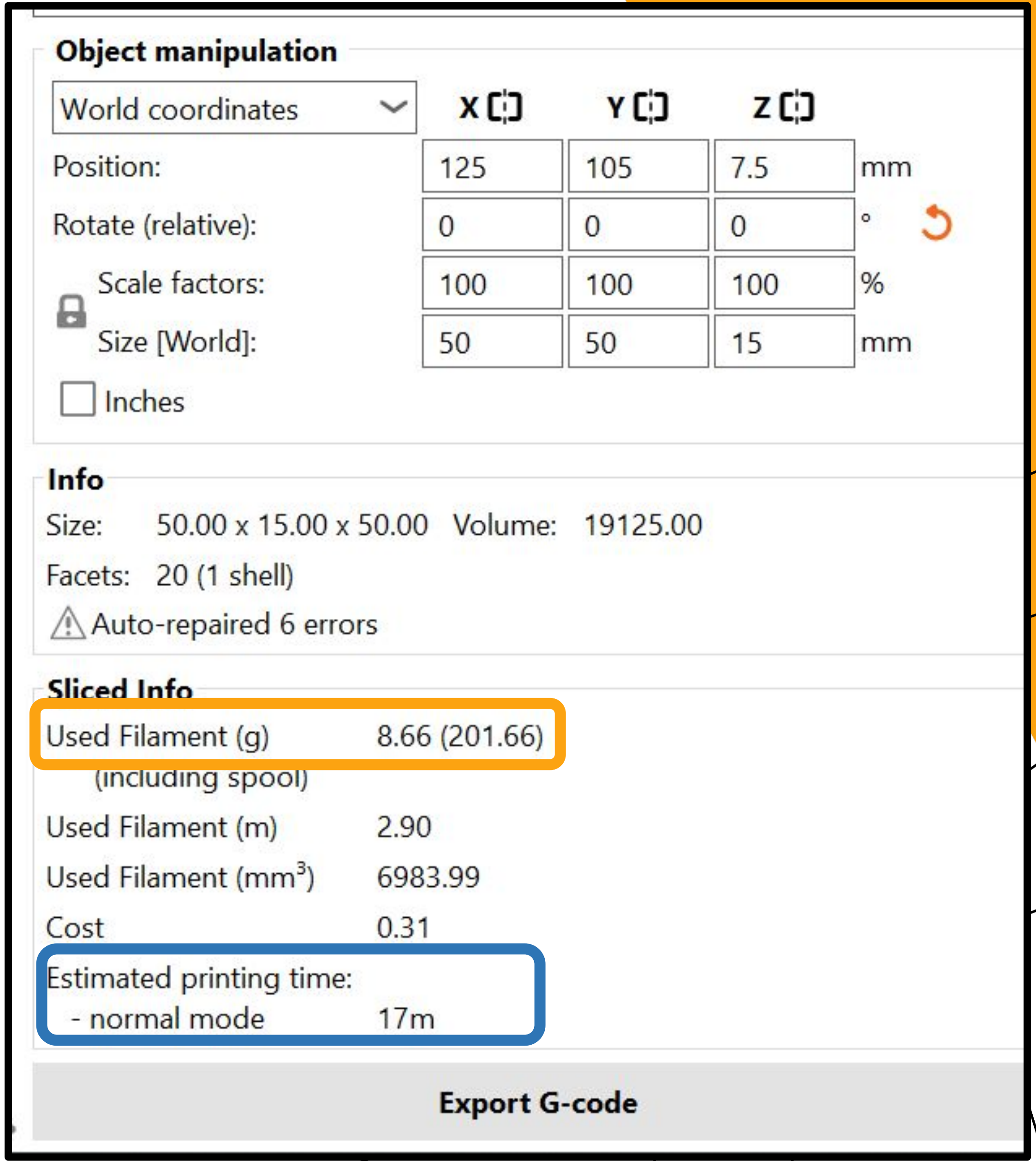


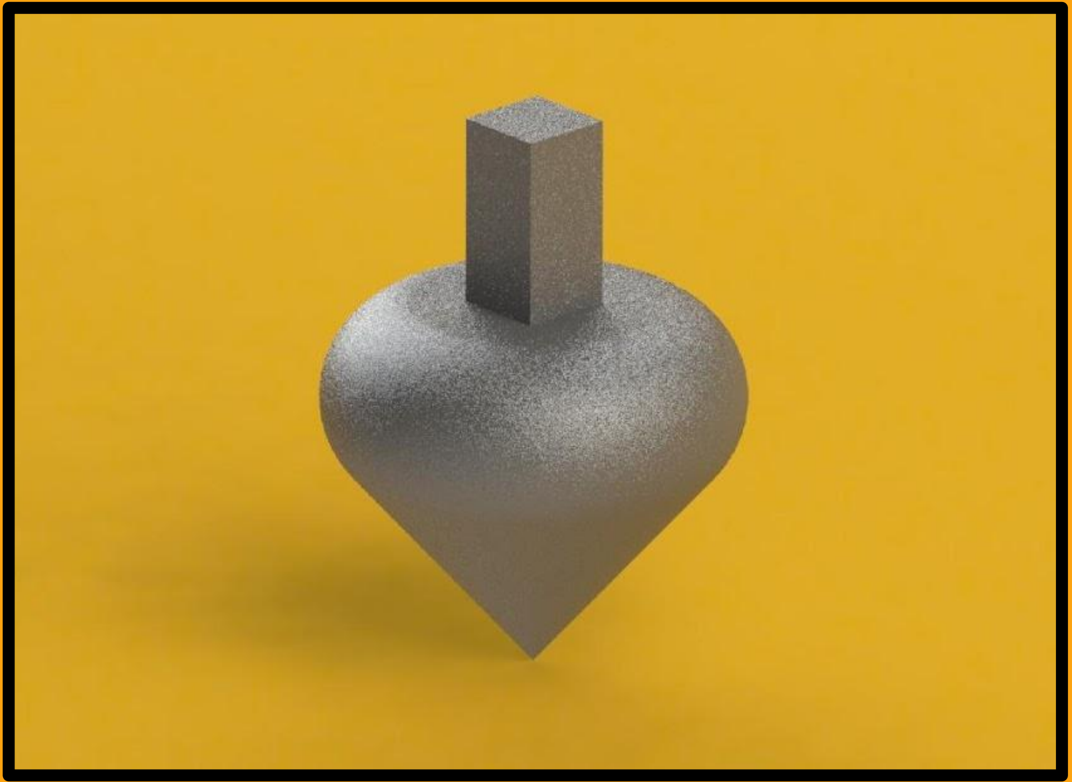
SLICING

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.





FINAL PROJECT

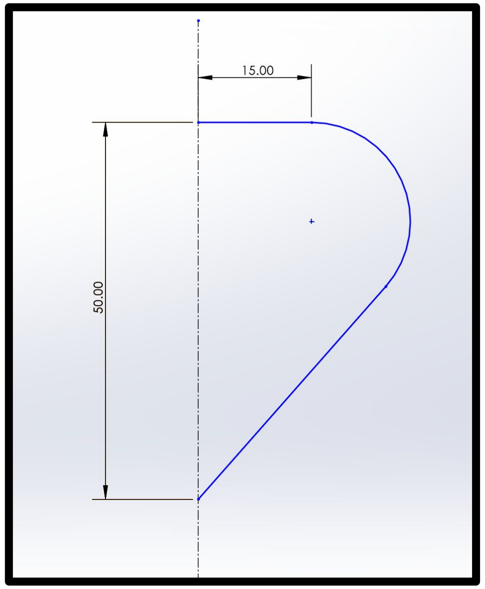
Let’s make some spinning tops!

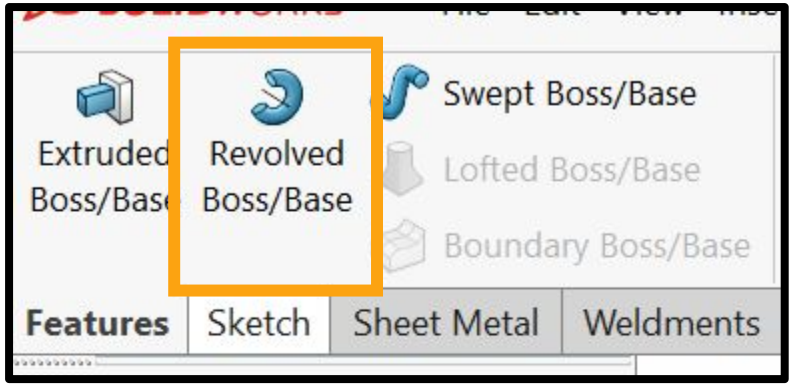
THE REVOLVING TOOL

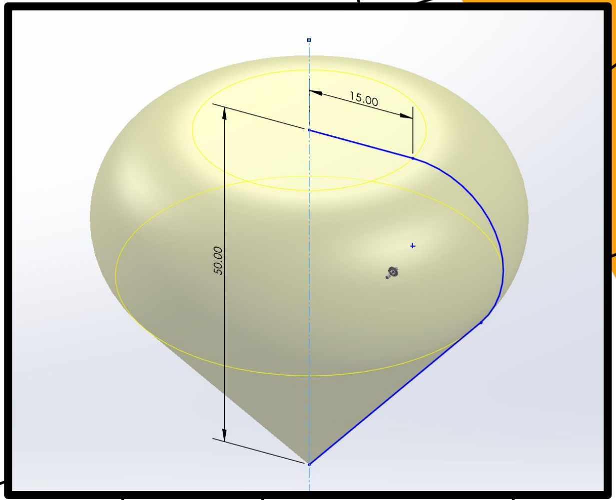
Now, using our existing toolset would make this basically impossible. Luckily, we have the handy revolve tool.

As before, start with a sketch. However, this time we'll sketch the 'profile' of our part.

This tool is pretty weird. Take a few minutes to play around with it.





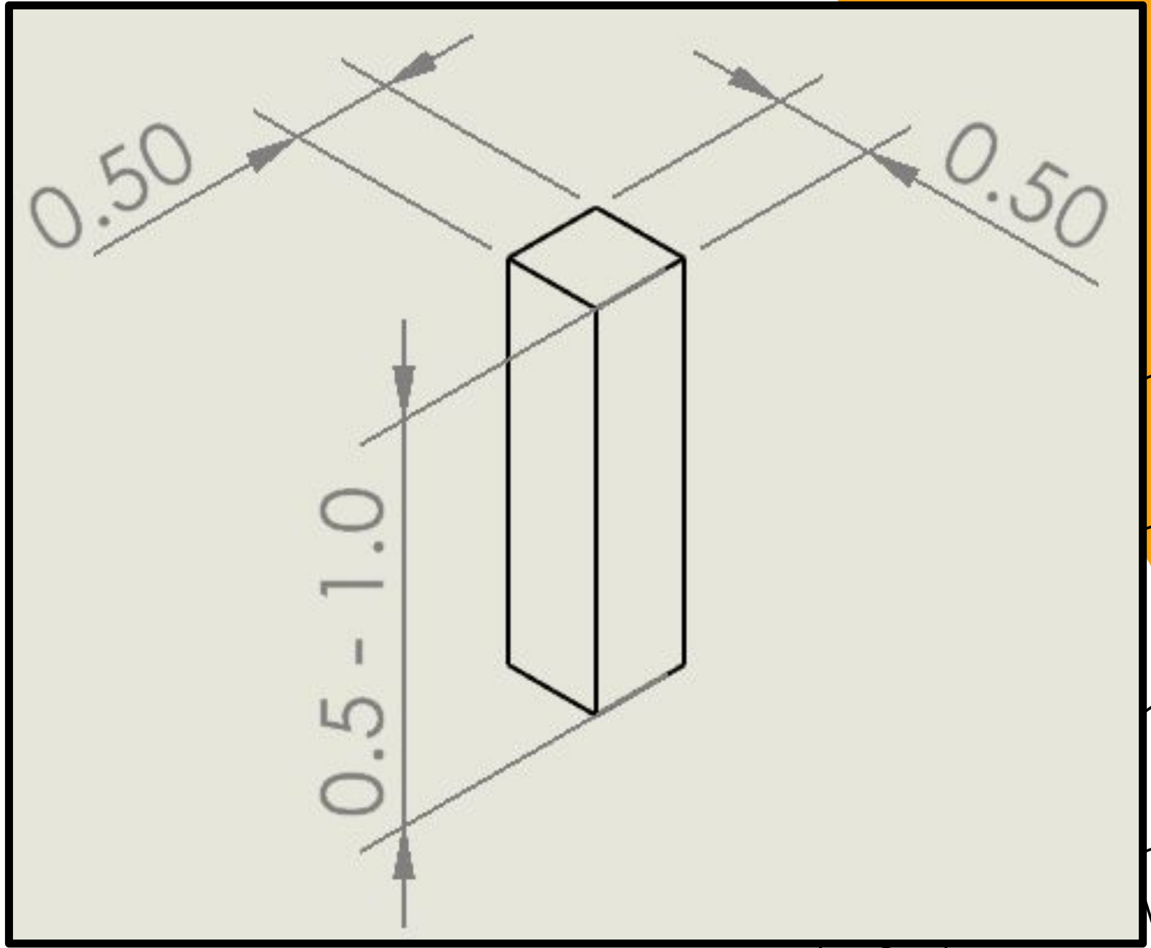


ADD THE SHAFT

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"x0.5" 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

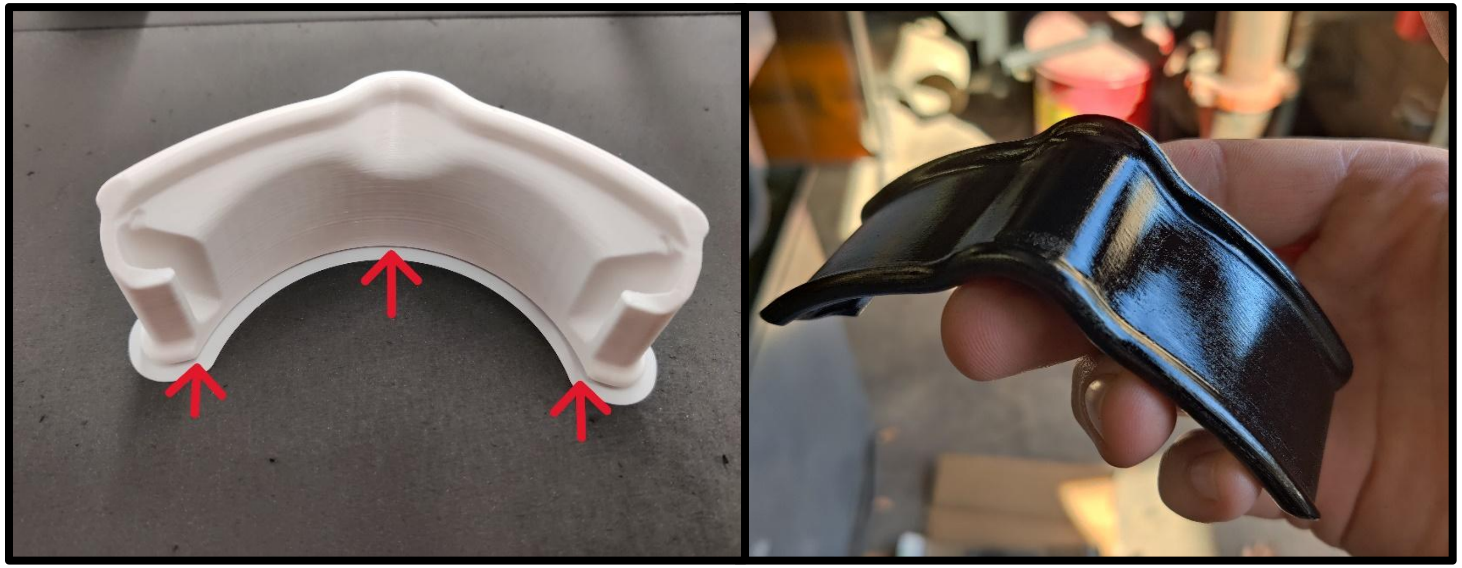
In order to check print viability and estimations of material used (in PrusaSlicer), you’ll be changing and exporting file quite often. This is the intended goal, to get you comfortable with the workflow.

POSTPROCESSING

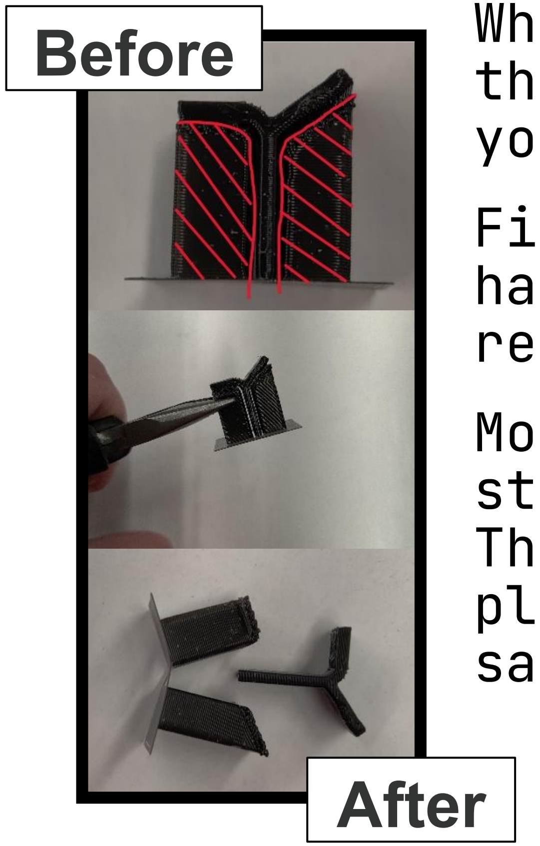
Making prints look nice.

WHAT IS POSTPROCESSING?

When parts come off the 3d printer, they are rarely ready to be used and will often need some extra work to finish them. Depending on the part this can include support removal, gluing, painting, and more.



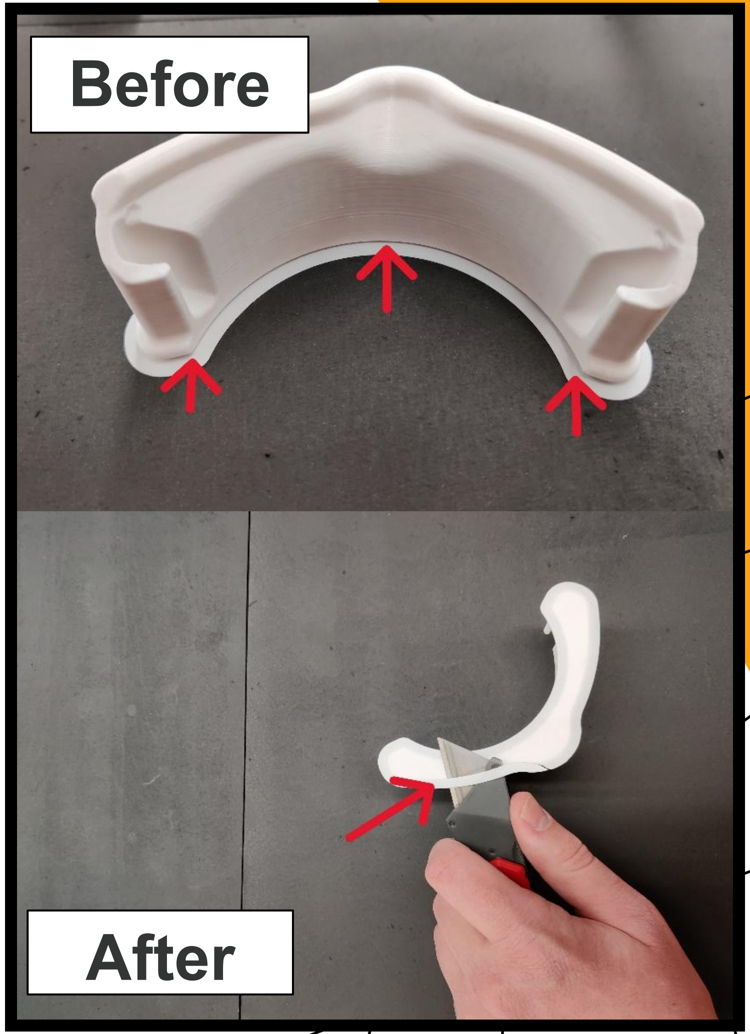
BRIM AND SUPPORTS



When your print is first removed from the printer, there are two things you'll commonly see.

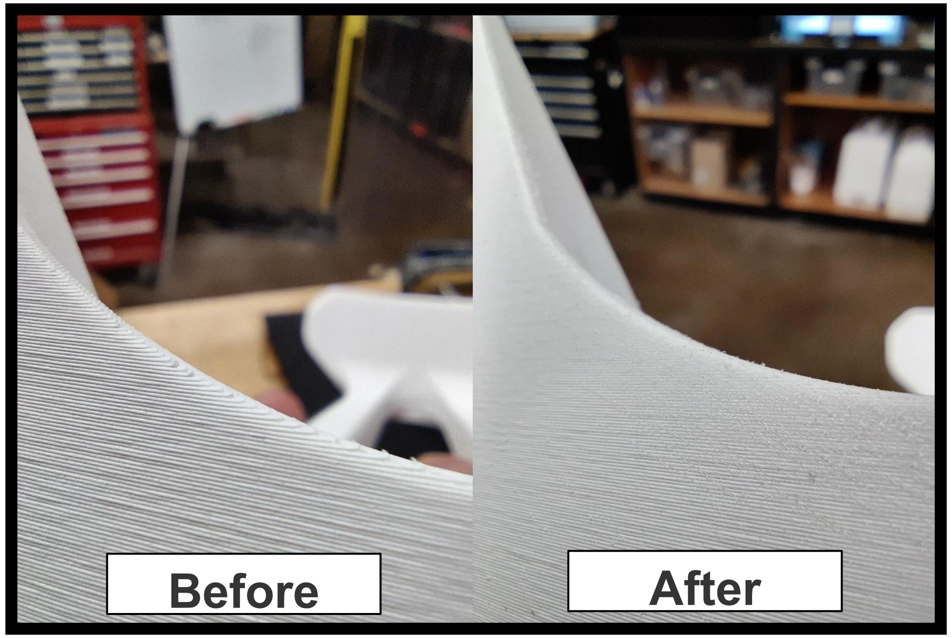
First, large and flat prints often have brims added. These can be easily removed with a sharp knife.

More complex parts also have support structures that need to be removed. These are easily pulled off with pliers and cleaned up with a knife or sandpaper.



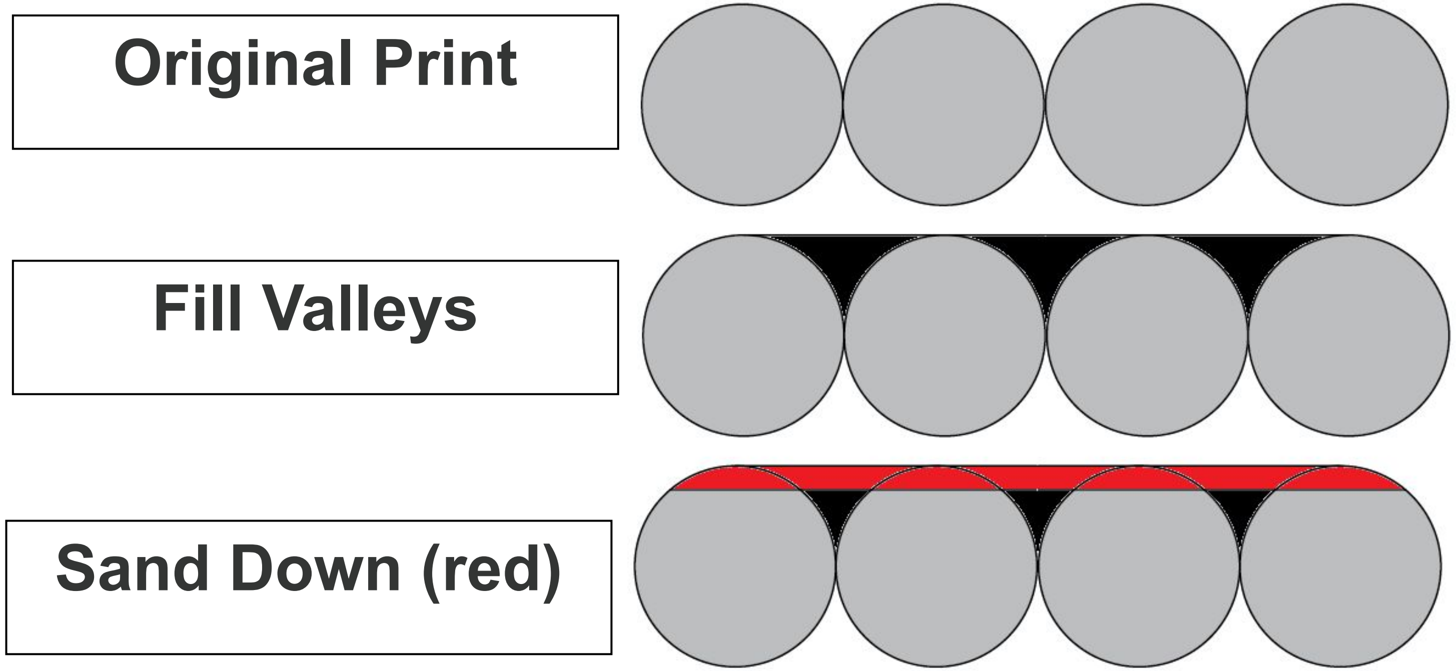
FIRST SANDING

Once we've removed the brim and supports from the part, we want to sand down the most aggressive layer lines and flaws. Starting with a small piece of ~220 grit sandpaper is a good grit to knock down edges and large flaws. Remember, we aren't trying to get everything perfect right now. That will come in a later step.



FILL, PRIME, AND SECOND SANDING

Once we've gotten the worst of the flaws out, we can start to get rid of the finer lines. Importantly, we don't want to simply sand until they are removed because this will severely reduce the dimensions of the part and will remove some of the detail in the print. Instead, we can fill in the valleys before sanding again.



FINAL RESULTS

Unfortunately, we won’t have time to bring our parts to a full gloss finish. However, we can get the worst of the layer lines out with some quick sanding.