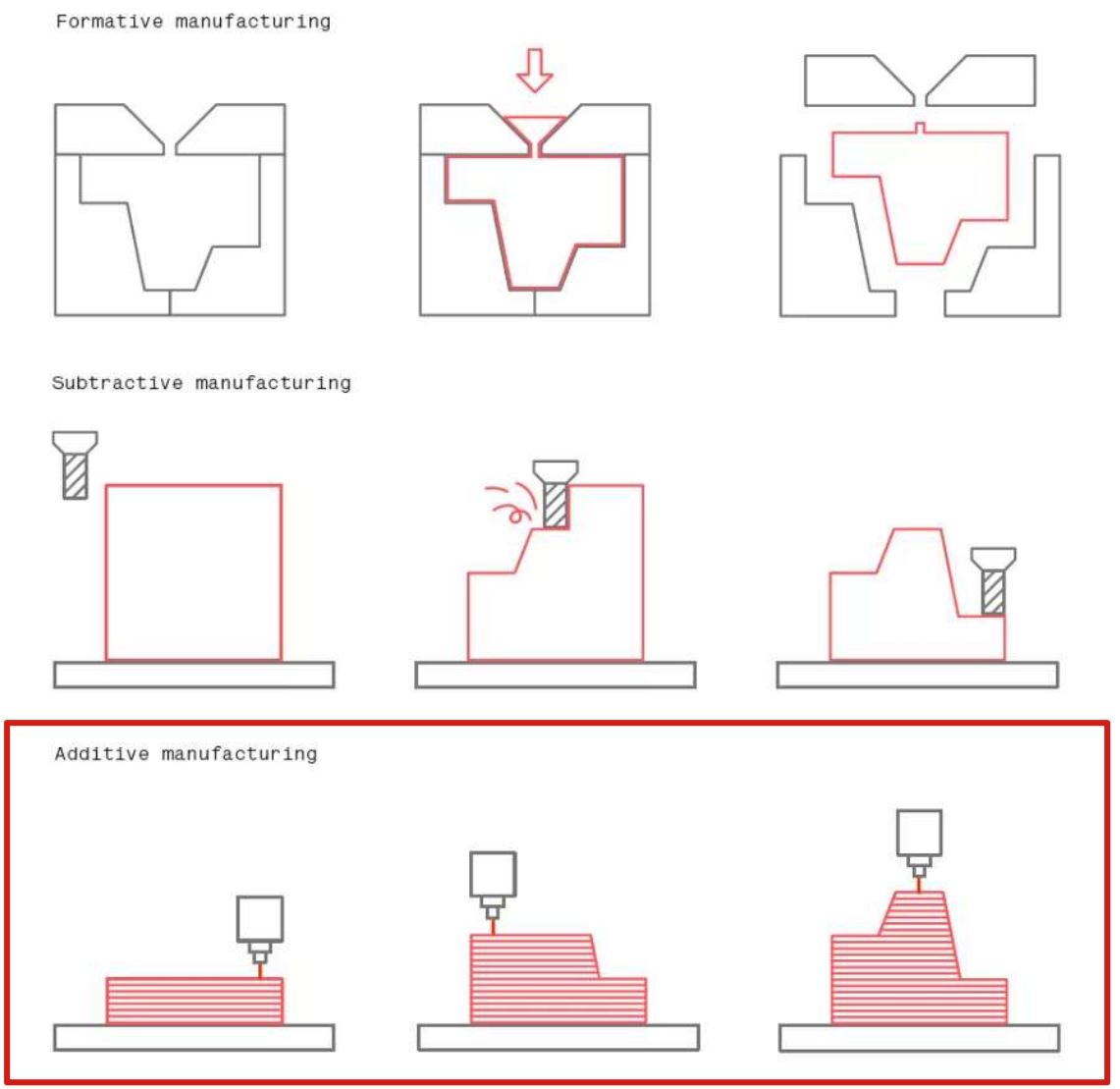

NJIT
Makerspace

Intro to 3D Printing

SLA-1 (1983)

What is 3D Printing?

- 3D Printing is an additive manufacturing process that creates a physical object from a 3D digital model
- Regardless of the technology used, 3D printed objects are created layer-by-layer



Pros and Cons of 3D Printing

Pros

- Create complex designs
- Iterate quickly and inexpensively
- Less waste
- No need for tooling, molds, or fixturing

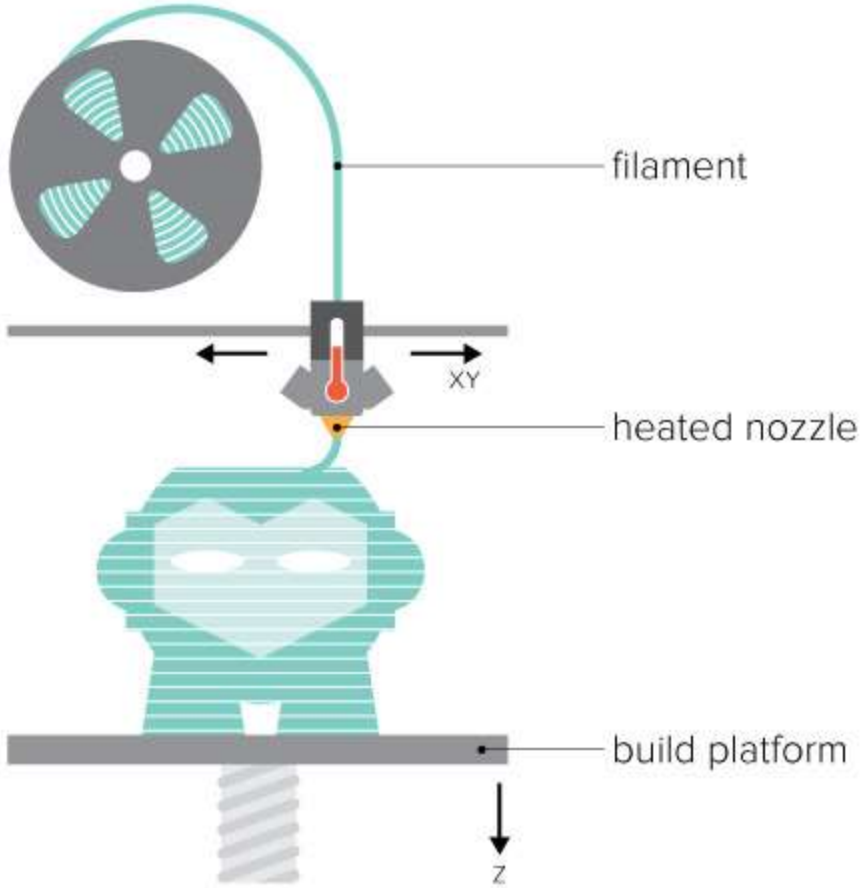
Cons

- Production runs (greater # of parts) are costly
- Less material choices
- Limited part strength/longevity
- Less precision than other manufacturing methods

The FFF Process Explained

Fused Filament Fabrication

- Also referred to as Fused Deposition Modeling (FDM)
- A spool of solid plastic filament is fed into a heated nozzle via motor, where it is **extruded** in a melted form
- Linear motors move the extrusion head over the build plate in “x” and “y” directions, laying down melted filament in the shape of the 3D model and cooling it on top of the **build plate**
- A lead screw moves the build plate in the z-direction



What Can I Do With the 3D Printer?

Prototype

Complex internal geometries

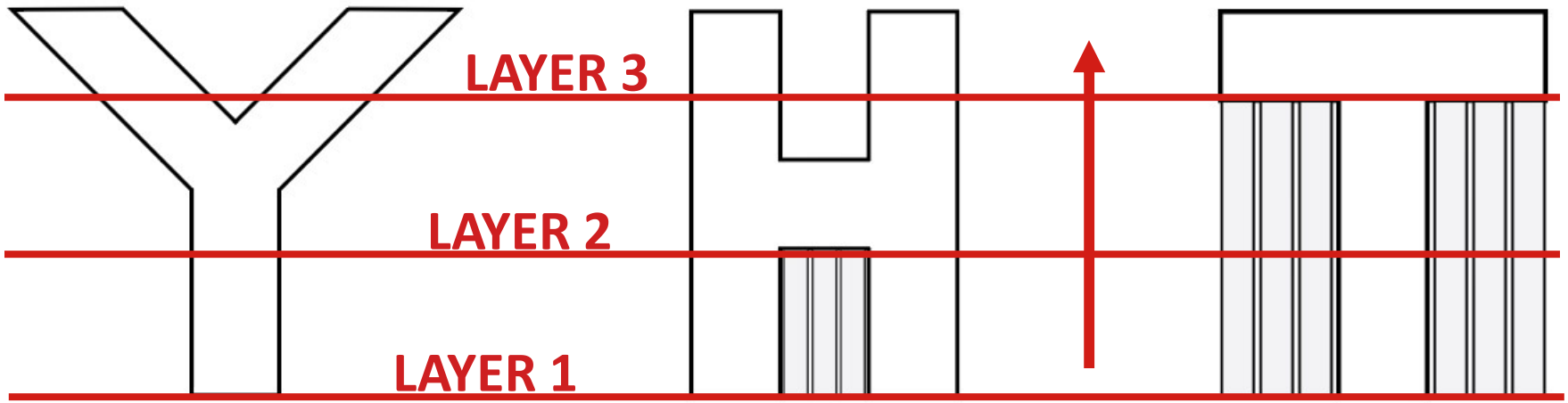
End-use Assemblies

Basic Design Considerations

- Using Supports
  - Overhangs
  - “Bridging”
- Vertical Axis Holes
- Build Direction & Resultant Strength
- Warping/Build Plate Adhesion (Cura)
- Layer-to-Layer Adhesion (Cura)

Basic Design Considerations

Supports

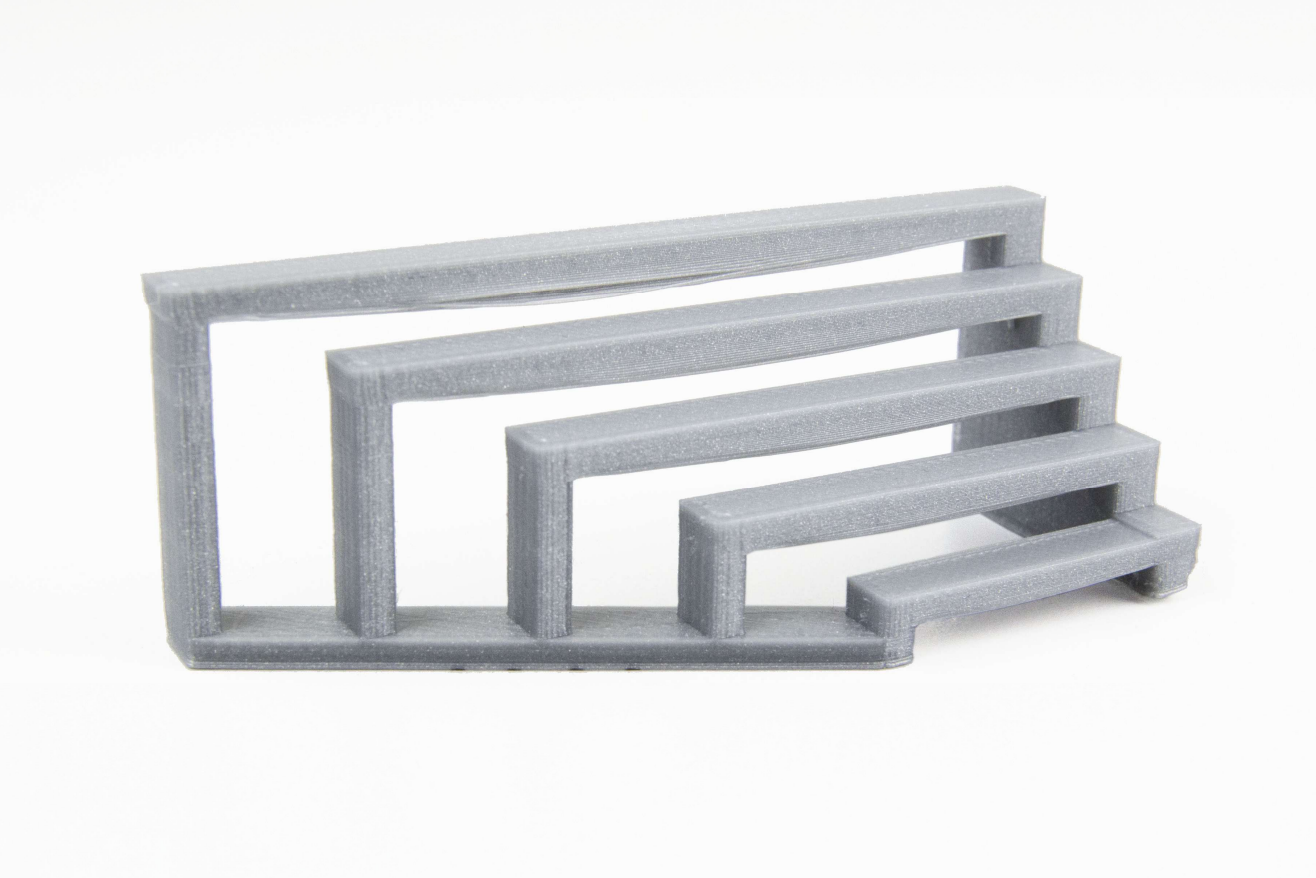


- In FDM printing, layers of melted plastic are laid down on top of subsequent solid layers and cooled to form a continuous solid
- **In the absence of a preceding solid layer of filament, a support structure is needed to support the next layer to be laid down**
- **Essentially, if part of a layer would be printed over "thin air," it will need a support structure underneath it to bond to**
- Supports can be easily removed once the print is completed

Basic Design Considerations

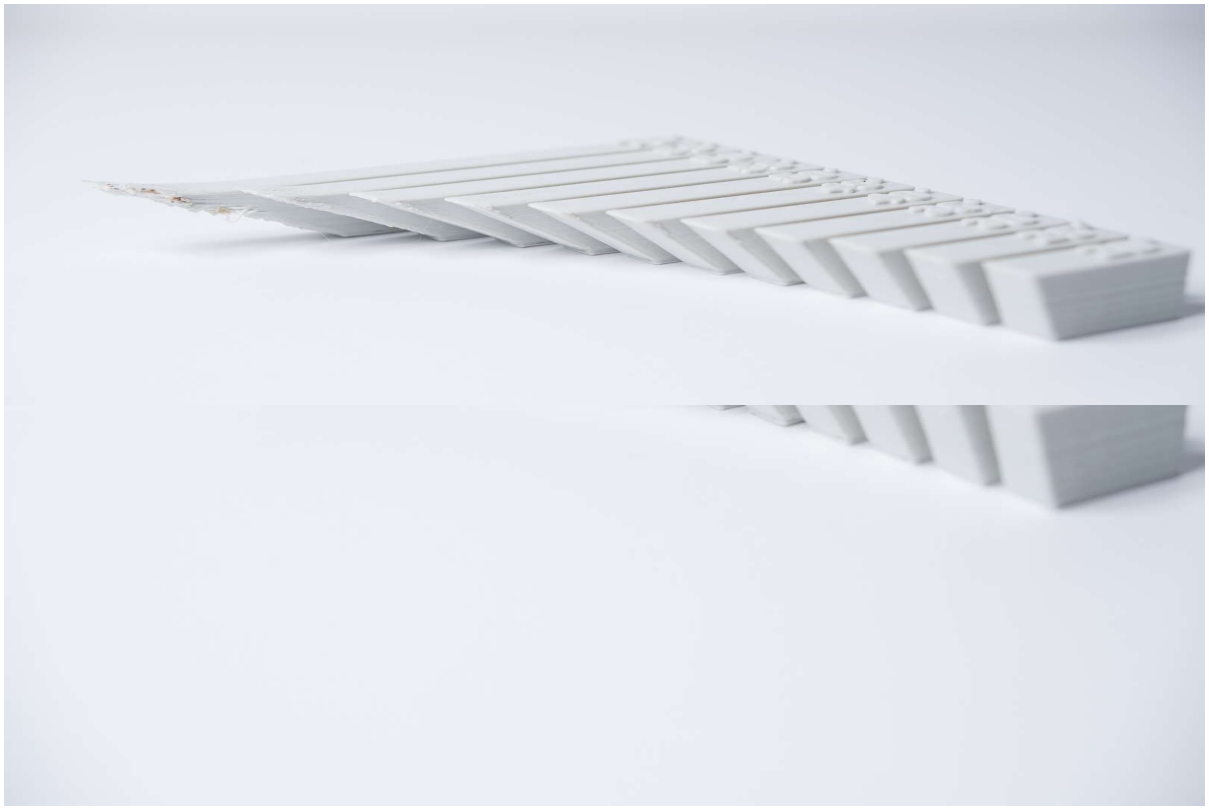
Bridging

- Short for “bridging a gap”
- Without a solid structure for the plastic to be extruded onto, it must be cooled and solidified mid-air
- For a shorter bridging distance, **support structures** may not be needed



Basic Design Considerations

Overhangs

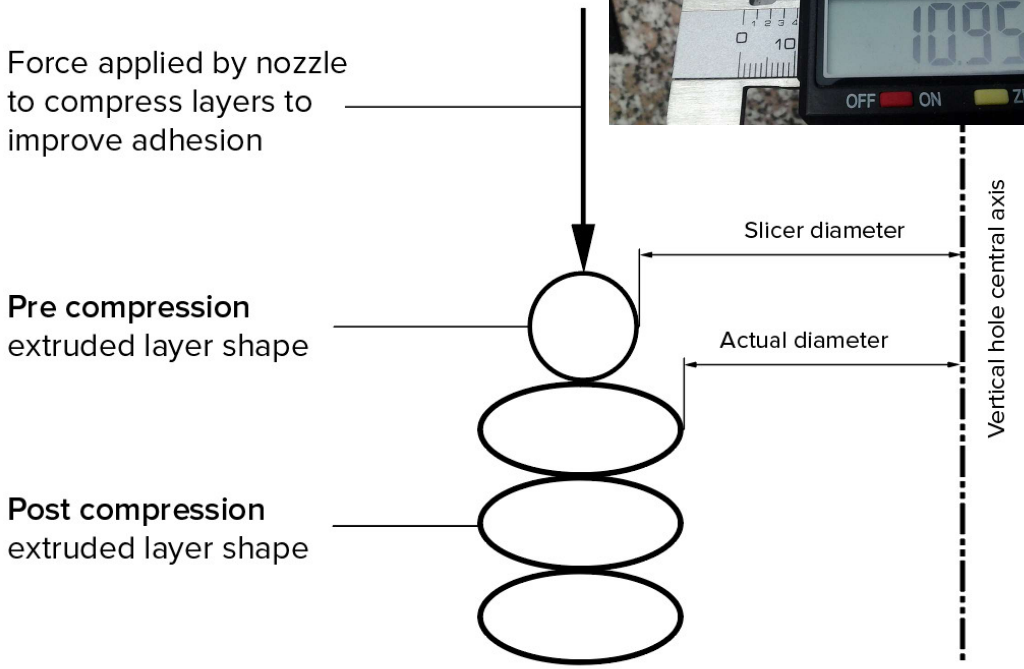


- Measured by the angle between your part geometry and a vertical line coming up from the build plate (like the lead screw)
- As this angle increases past **45°**, there is less material below the newly extruded layer to support it
- This results in poor layer adhesion or curling from differential cooling
- Again, **supports** solve this problem

Basic Design Considerations

Vertical Axis Holes

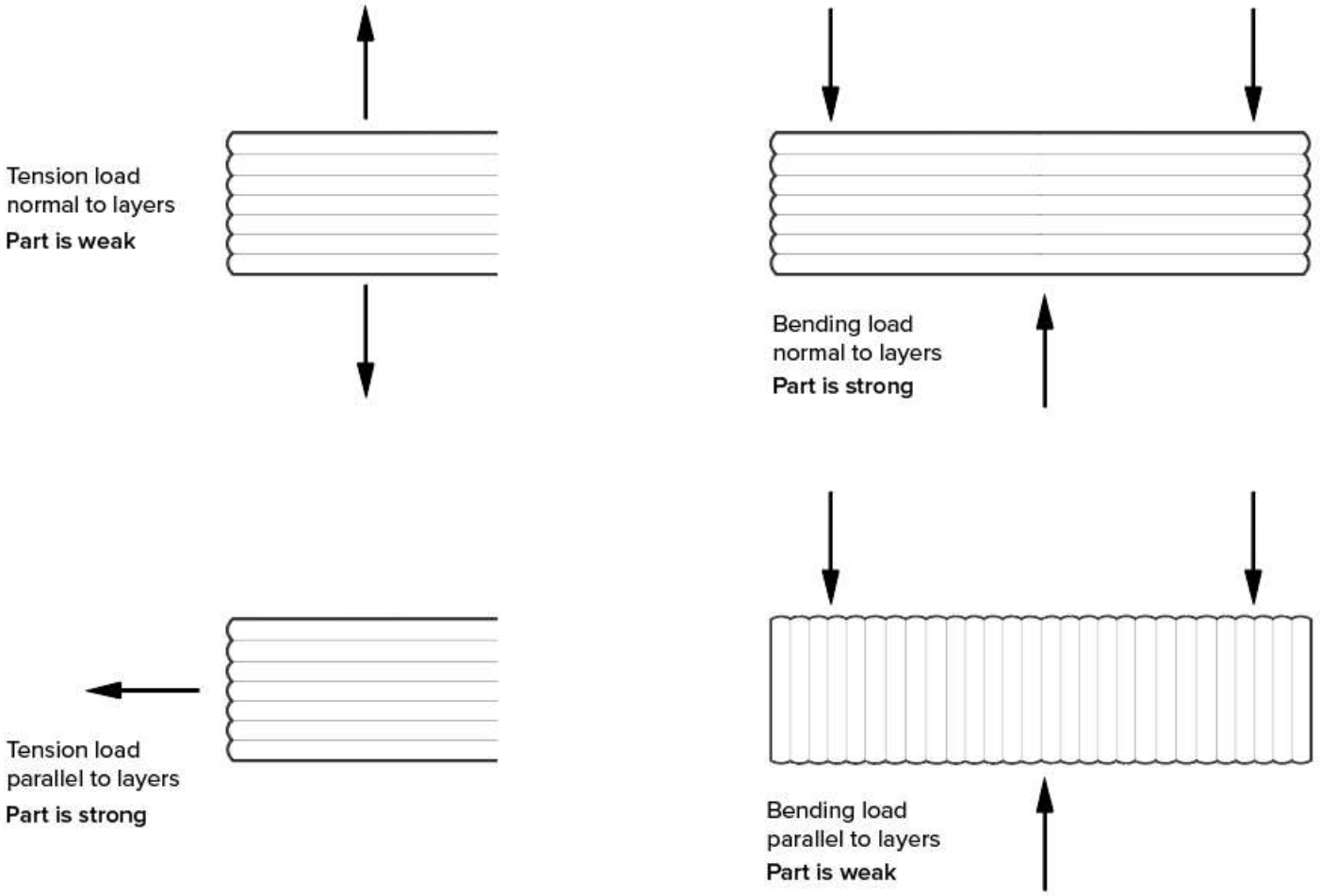
- The compression of extruded layers is not accounted for by our slicing software (Cura)
- This compression causes printed holes to be smaller in diameter than stated in the CAD design
- **Either chase vertical axis hole with a file or reamer OR oversize diameter in CAD design**

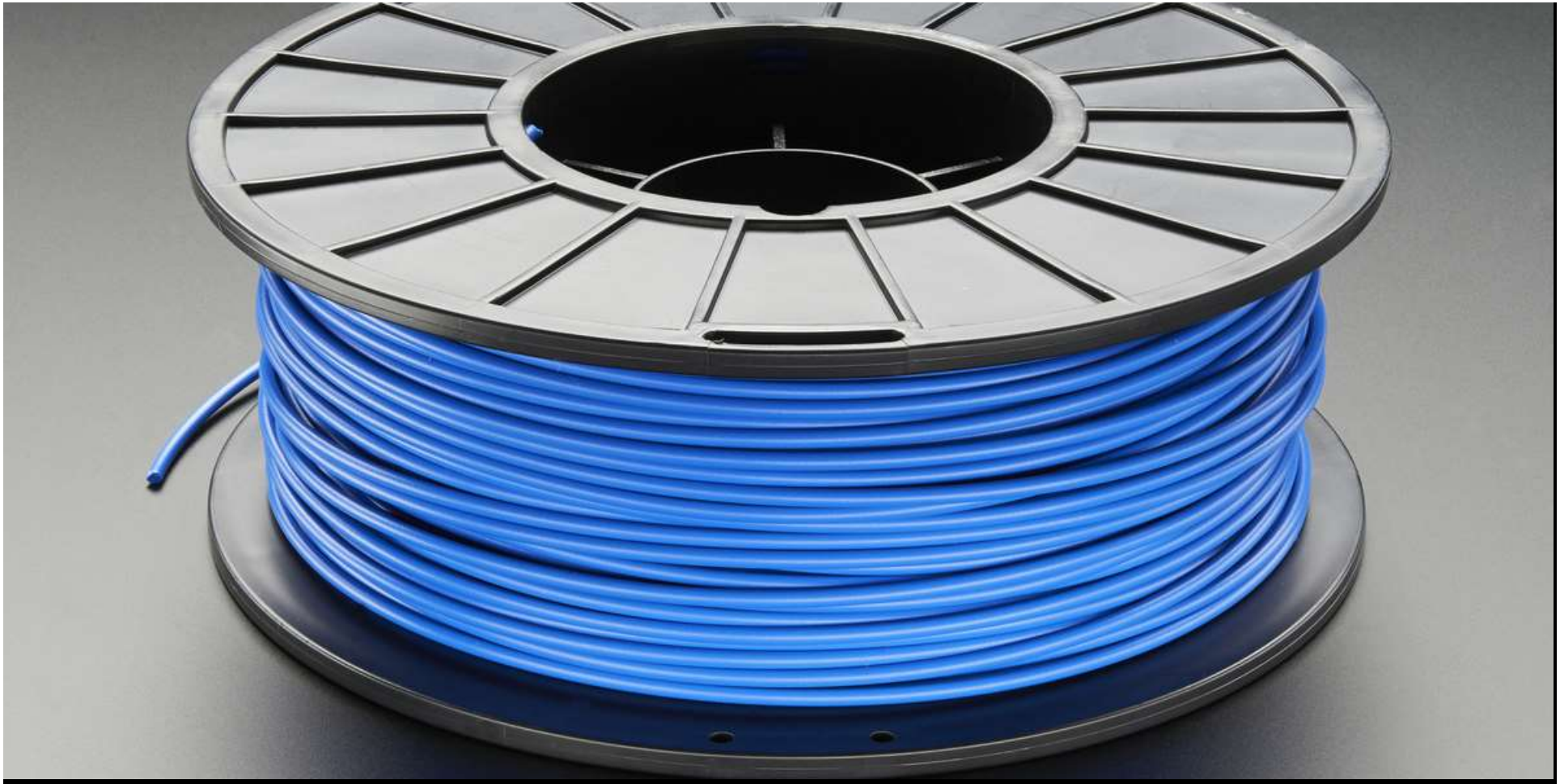


Cross-sectional view of extruded layers

Basic Design Considerations

Build Direction & Resultant Strength



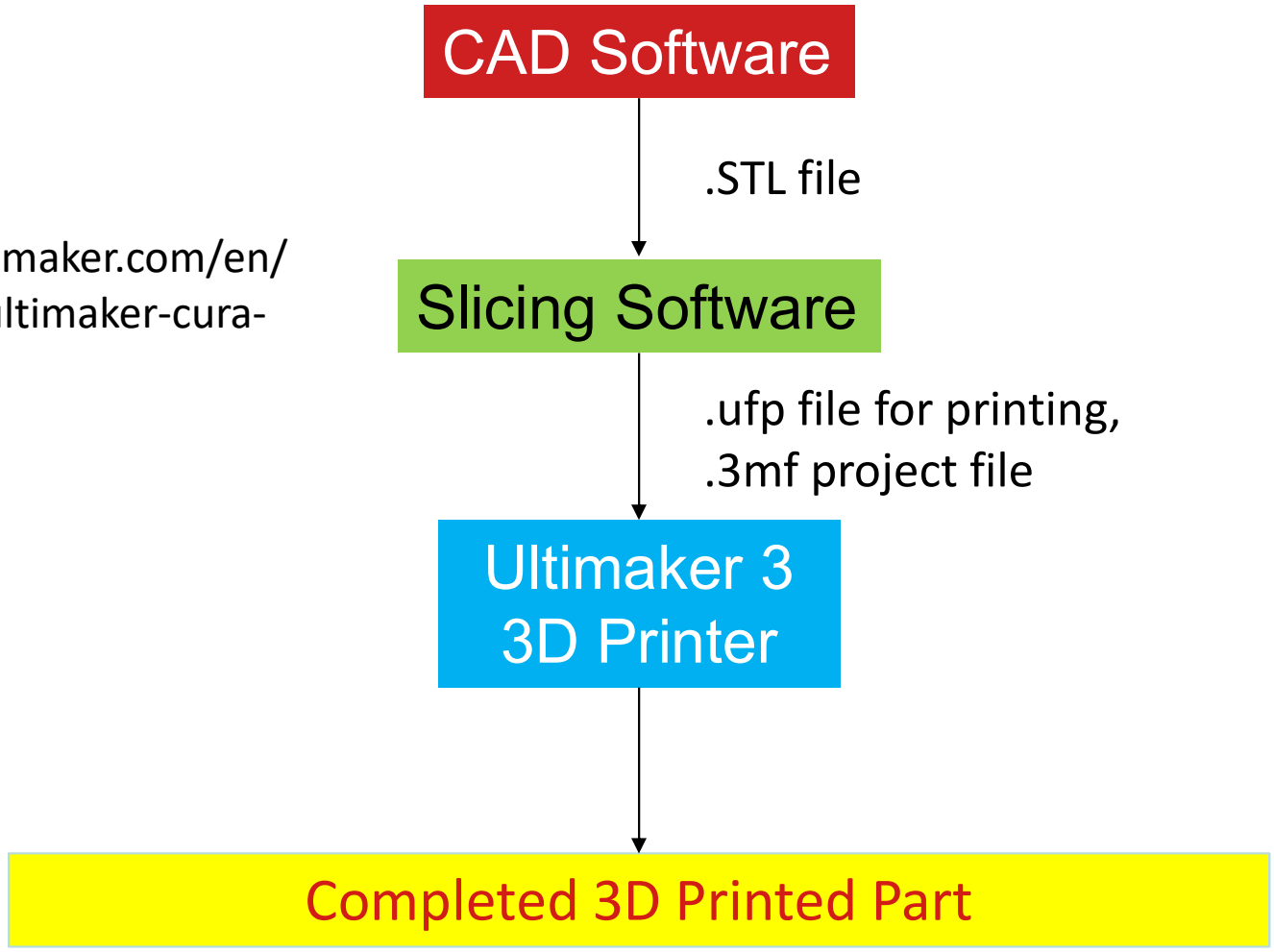


Available Materials

- **PLA** (Polylactic Acid)
- **PETG** (Polyethylene Terephthlate Glycol-Modified)

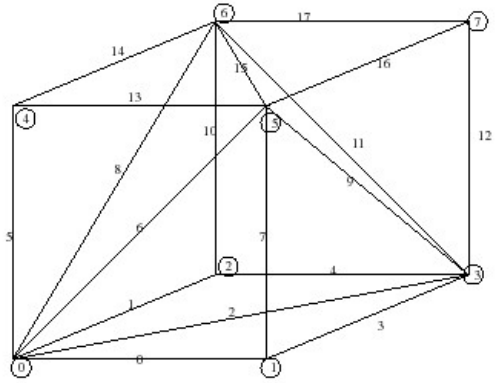
Workflow

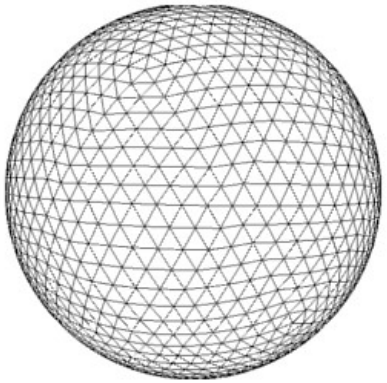
https://ultimaker.com/en/products/ultimaker-cura-software

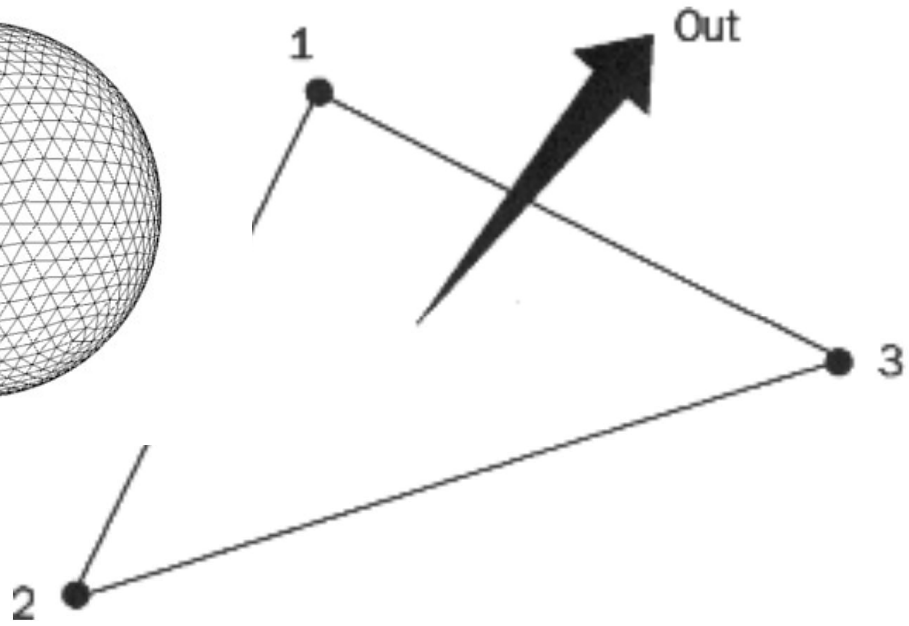


Exporting a .STL File

What is an .STL file?







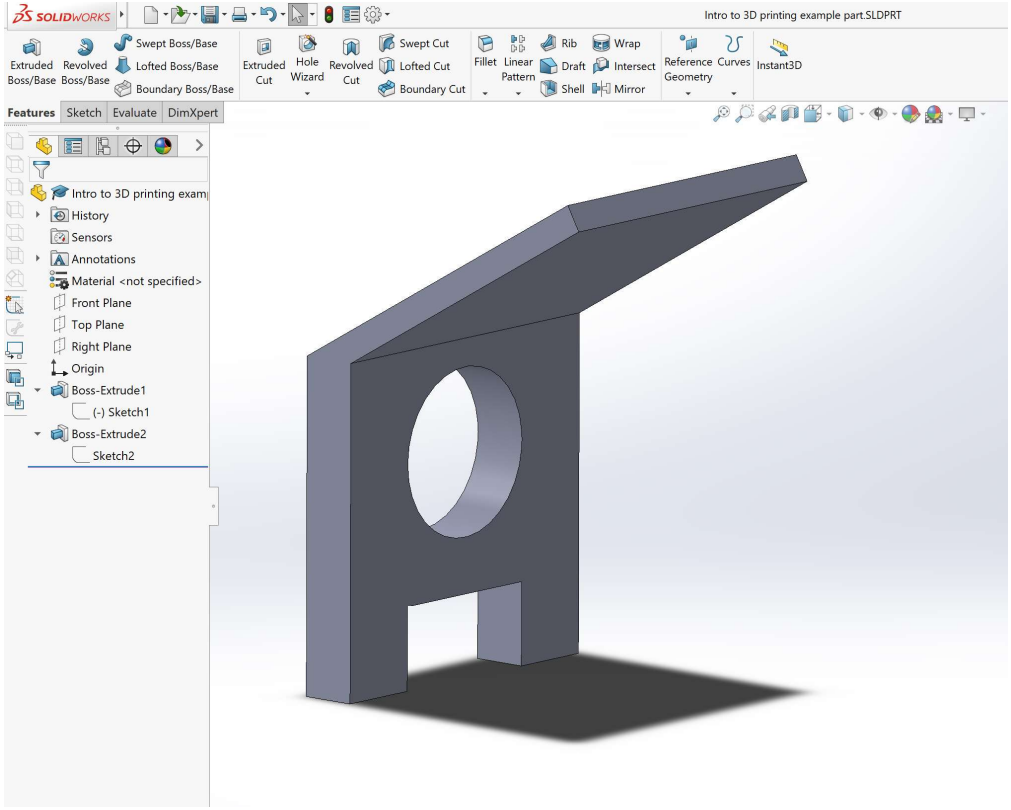
An STL file stores the co-ordinates of the vertices and the components of the unit normal vector to the facets

https://all3dp.com/what-is-stl-file-format-extension-3d-printing/#pointone

- A STereoLithography (STL) file uses triangle polygons to “describe” the surface of a 3D model
- 3 points in space and an outward direction (normal vector)
- This file will be interpreted by our slicing software, **Cura**

Exporting a .STL File

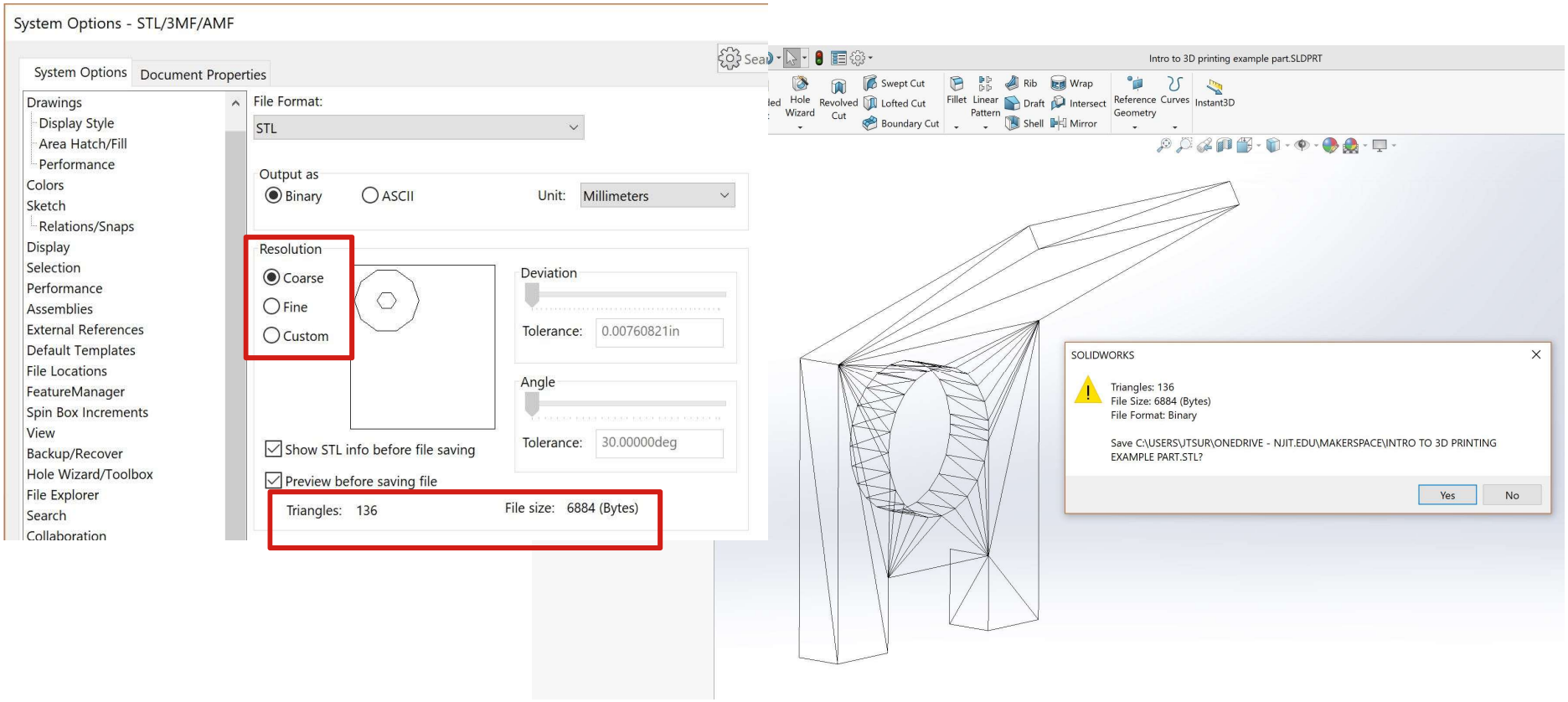
Coarse vs. Fine



- "Export" or "Save as" is typically where you can export as an .STL
- Most programs will have options for how your .STL will be created…

Exporting a .STL File

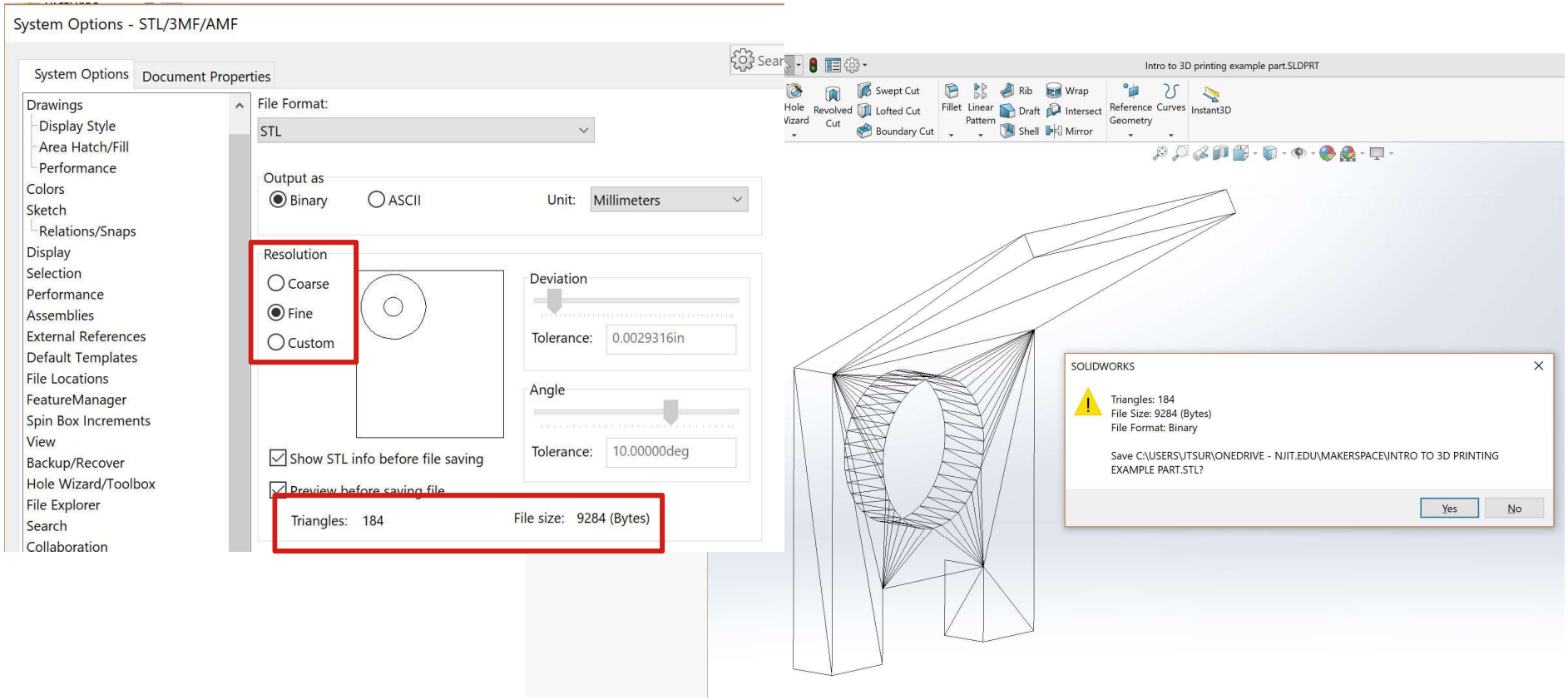
Coarse vs. Fine



- The “Coarse” setting will create an .STL with the fewest number of triangles
  - This decreases file size, but is a **less** accurate representation of 3D model

Exporting a .STL File

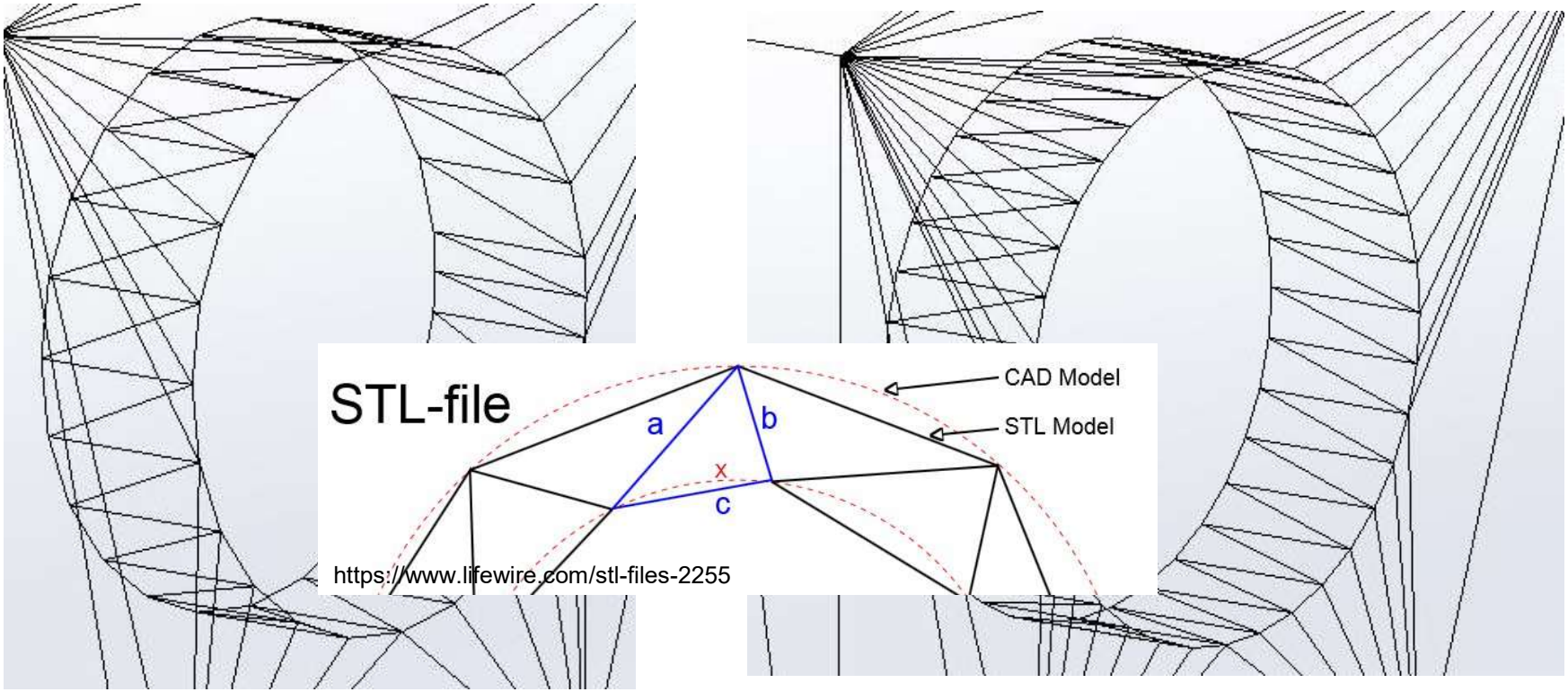
Coarse vs. **Fine**



- The “Fine” setting will create an .STL with more triangles
  - This increases file size, but is a **more** accurate representation of 3D model

Exporting a .STL File

Coarse vs. Fine & Effect on Holes

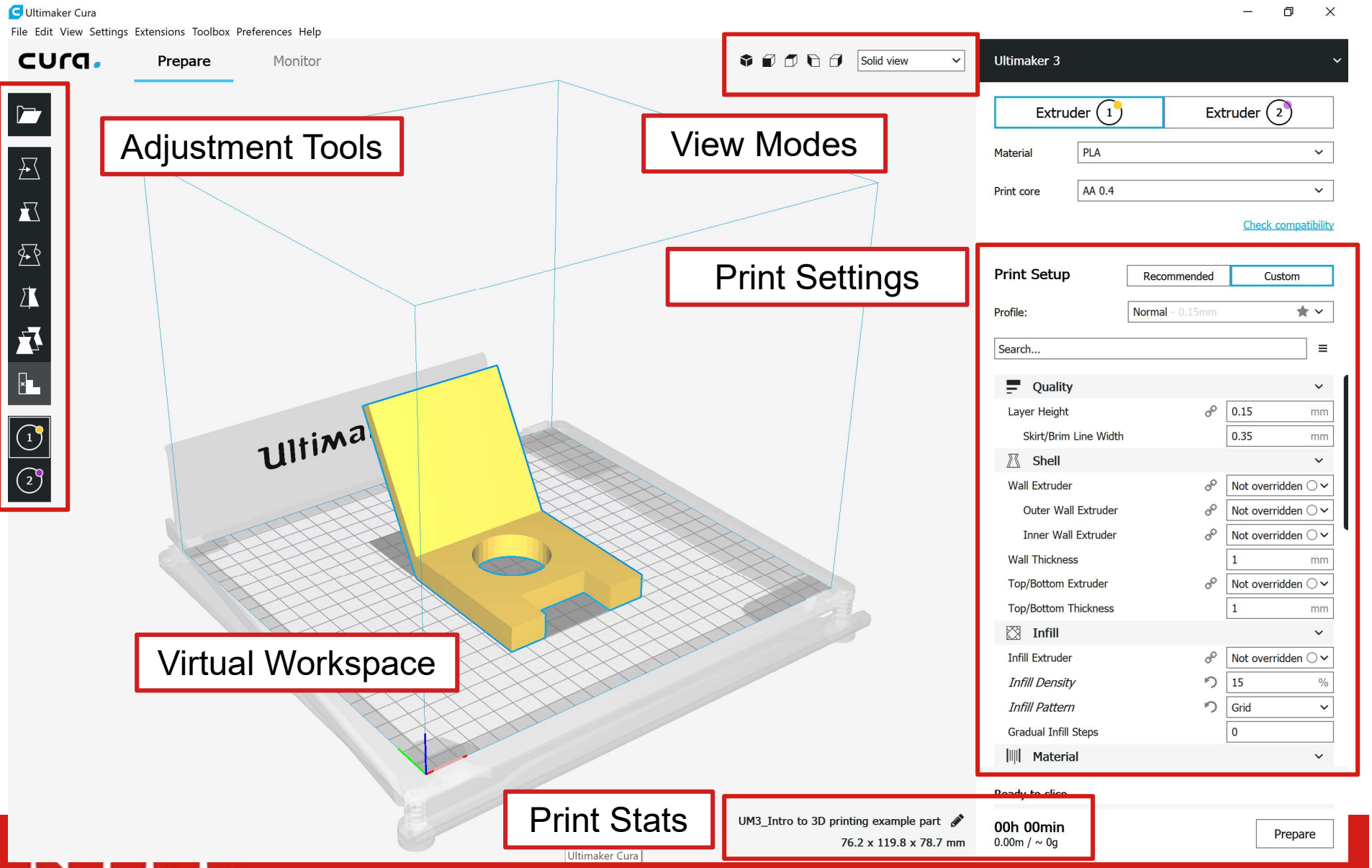


- Curved lines from a CAD model are represented with straight lines in a .STL
- **The more straight lines in your .STL, the more accurate your circle will be**

What is Cura?

- Open source, 3D printer slicing application
- Cura “slices” .STL files into layers to determine the optimal printing method and converts this into machine specific g-code
- This g-code is ultimately sent to the printer for fabrication

Cura Main User Interface



Adjustment Tools

Open File

Move

Scale

Rotate

Mirror

Per Model Settings

Support Blocker

Print Selected Model with Extrude 1

Print Selected Model with Extrude 2

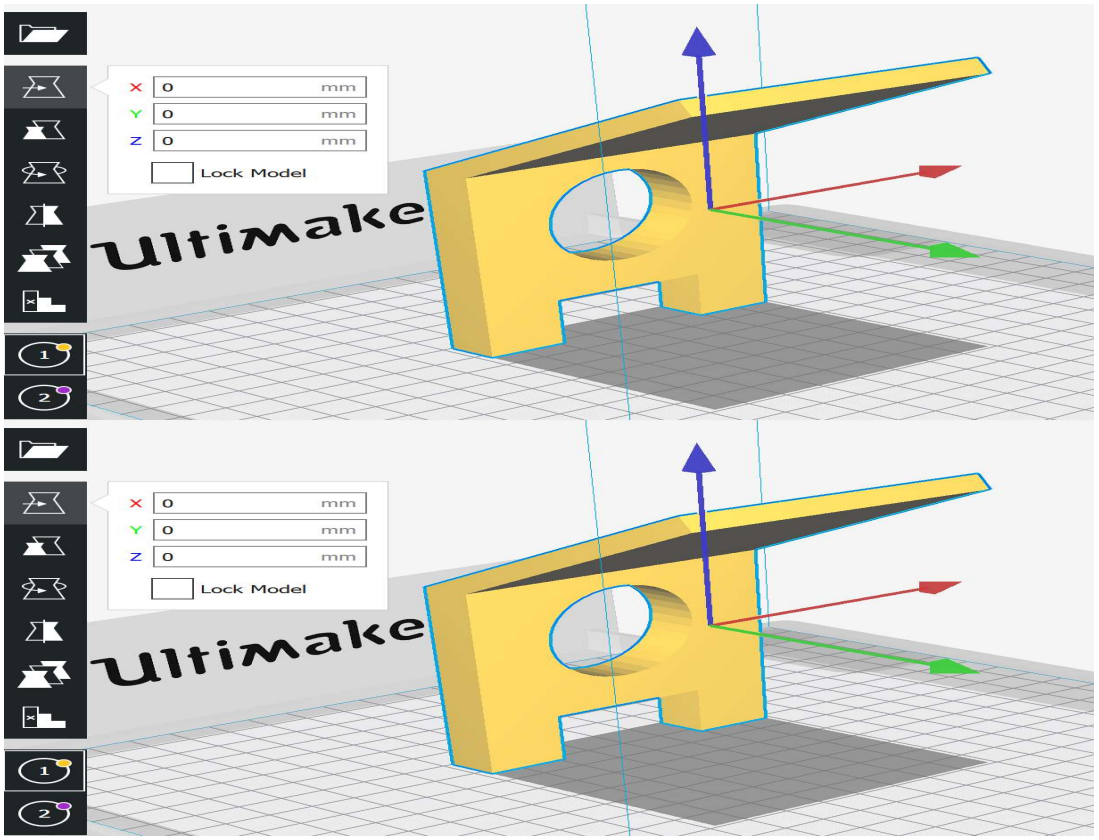
Adjustment Tools

Open

- **Select your .STL file and it is placed on the virtual build plate**
- More than one .STL file can be placed on the build plate at once
- Try opening a .JPEG!

Adjustment Tools

Move

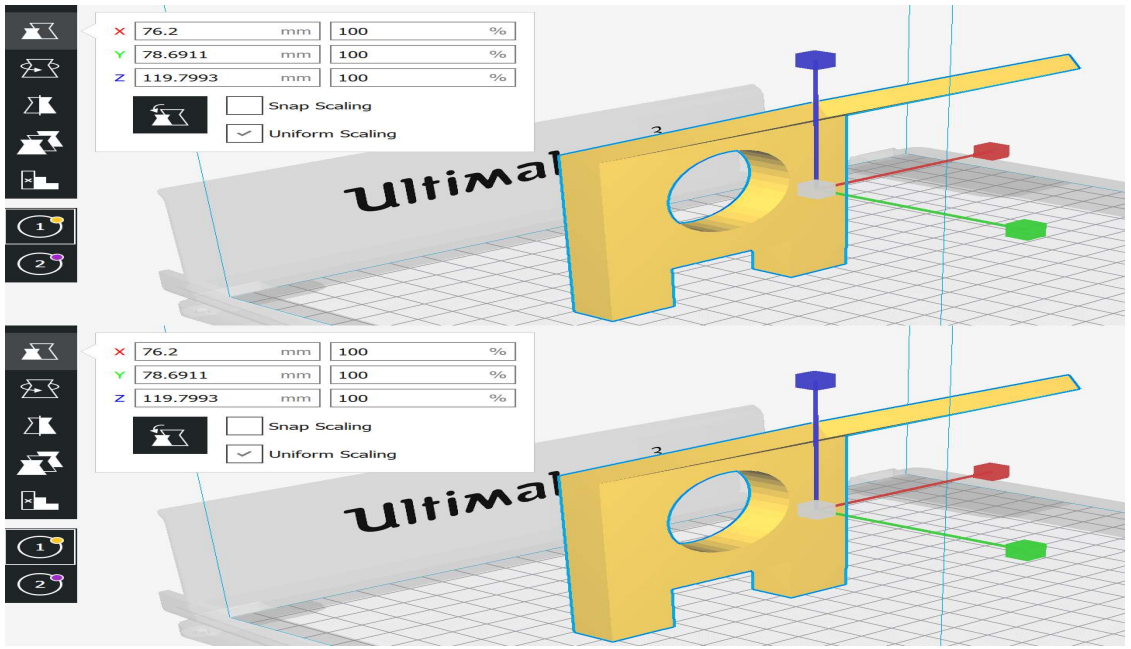


- **Part can be moved in the X & Y directions on the build plate**
- **A positive or negative number can be entered** to move the part relative to its original position, or the part can be **dragged via the red and green arrows**
- The part will always snap to the build plate (Z direction)

Adjustment Tools

Scale

- The size (scale) of the model can be uniformly changed
- The size ratios defined in the CAD model will stay true if **Uniform Scaling** is selected
- **Scaling can also be adjusted by percentage value**

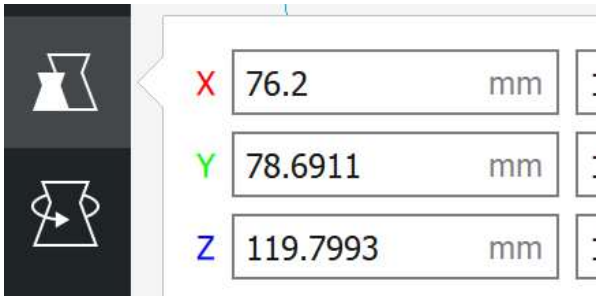


Adjustment Tools

Scaling Issues

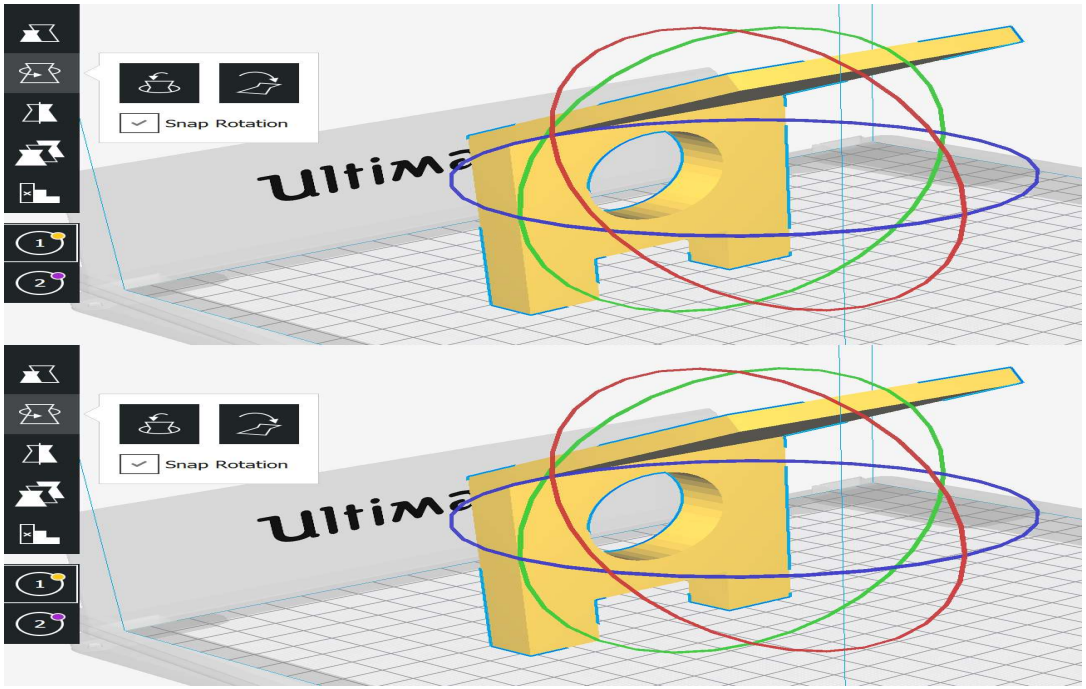
1 inch = 25.4 mm

- **Sometimes your .STL will be imported into Cura at a scale corresponding to standard units!**
- Determine a maximum dimension of your part in the X, Y, or Z direction from your CAD model (within your CAD software)
- Convert this dimension to millimeters
- Enter this value into the corresponding scale entry box



Adjustment Tools

Rotate

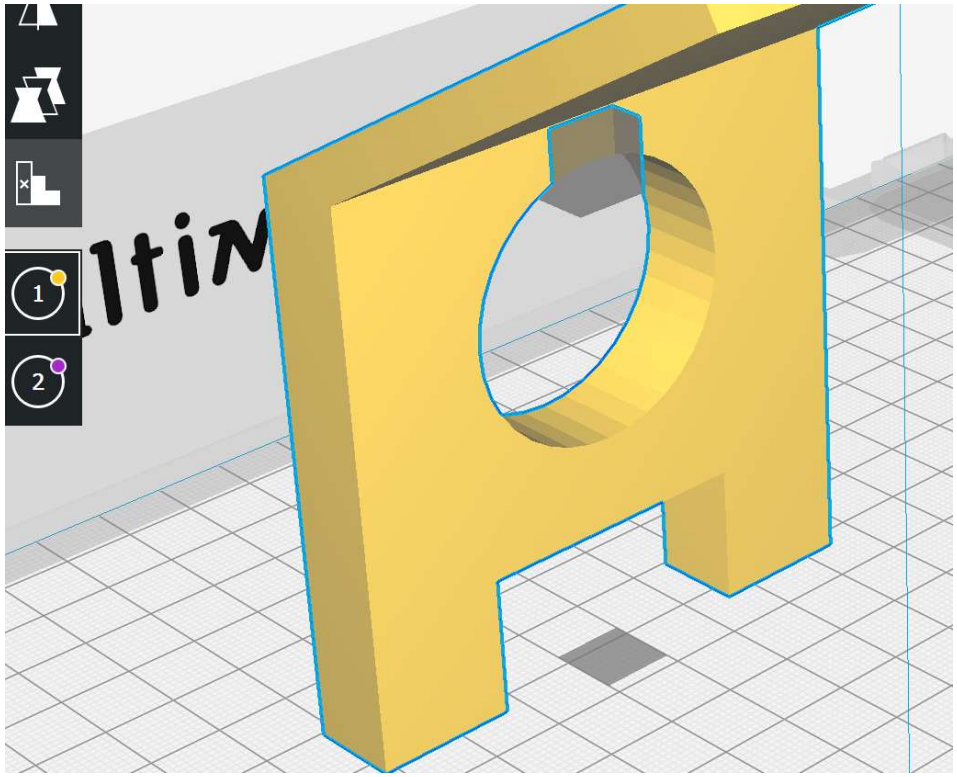


- The part can be rotated by clicking on a dragging 1 of the 3 circles around the part
- The left button resets the rotation back to its original position
- The right button lays the surface nearest to the build plate flat to the build plate
- **Snap Rotation** snaps the rotation to 15^{o} increments

Part Control Tab

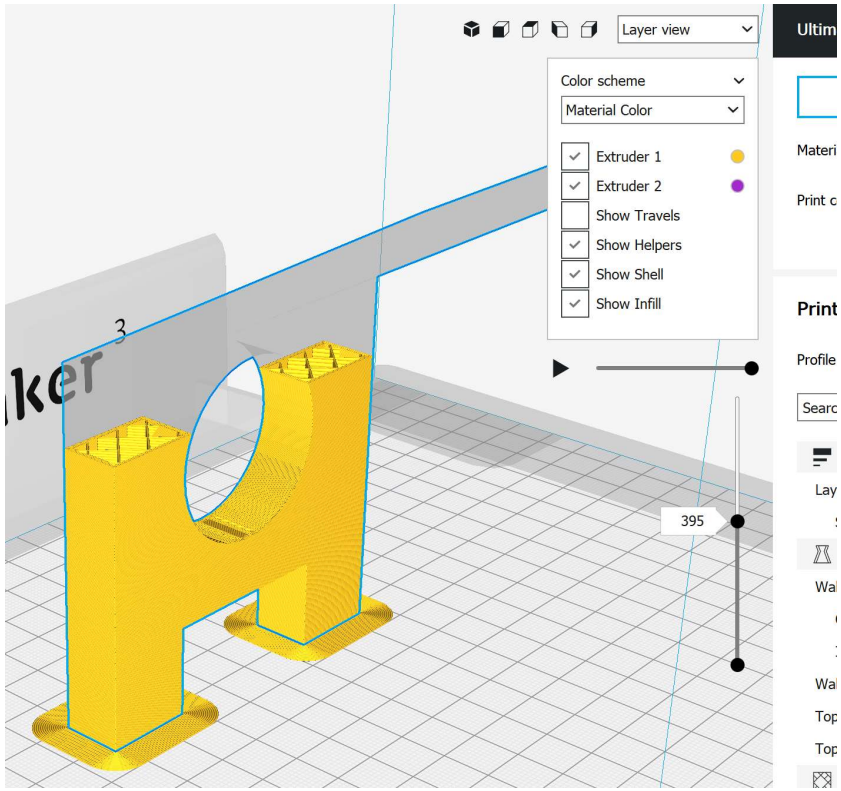
Support Blocker

- Using the **Support Blocker**, clicking on parts of the model will create a cubic area where **no supports will be placed**
- Clicking this area again will remove it



Layer View

Cross-sectional View of Part by Layer

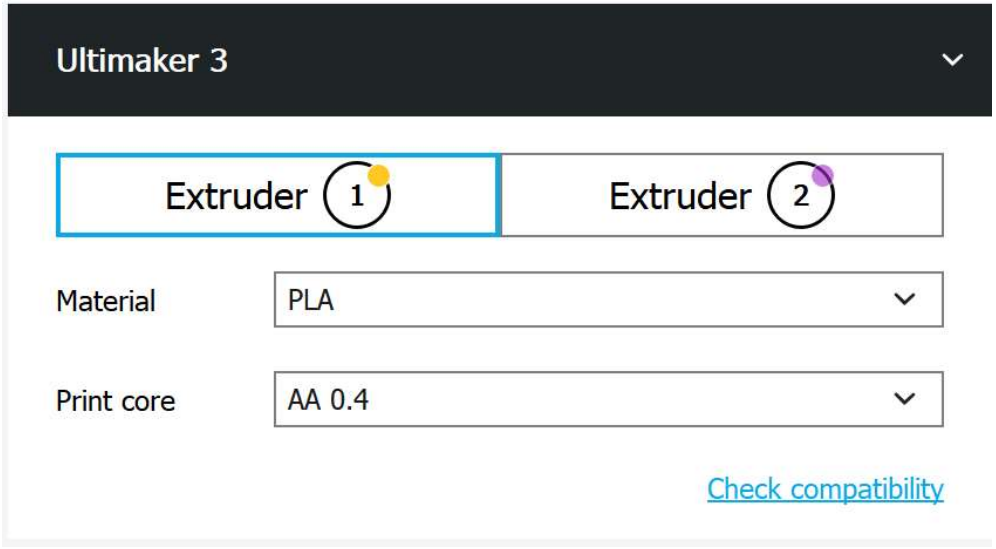


- Select this view in the drop down menu to the right of the preset view angles
- The vertical bar on the right can be moved to show each layer of the print

Configuration Settings Menu

Configuration Settings

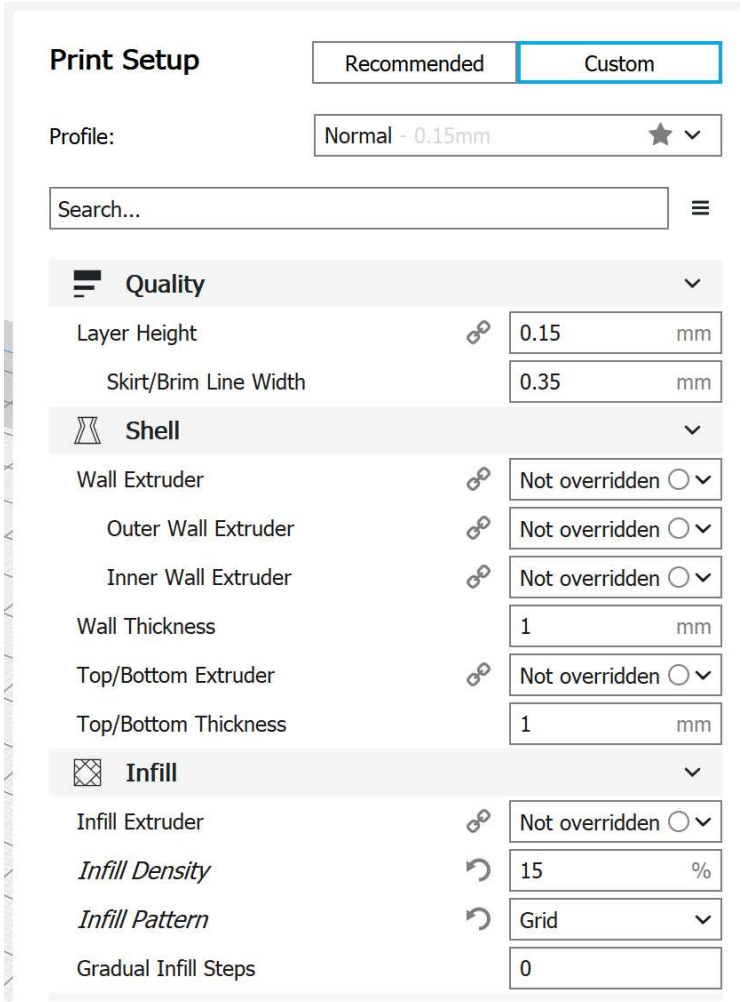
Printer & Extruder Selection



- The “self-serve” printer station has only Ultimaker 3 printers (default)
- The part on the virtual build plate should be printed with **Extruder 1**
- The material to be printed in can be selected, as well as the print core in the **Extruder 1** slot on the print head
  - Don’t change print core settings! PLA must be extruded with an **“AA”** core

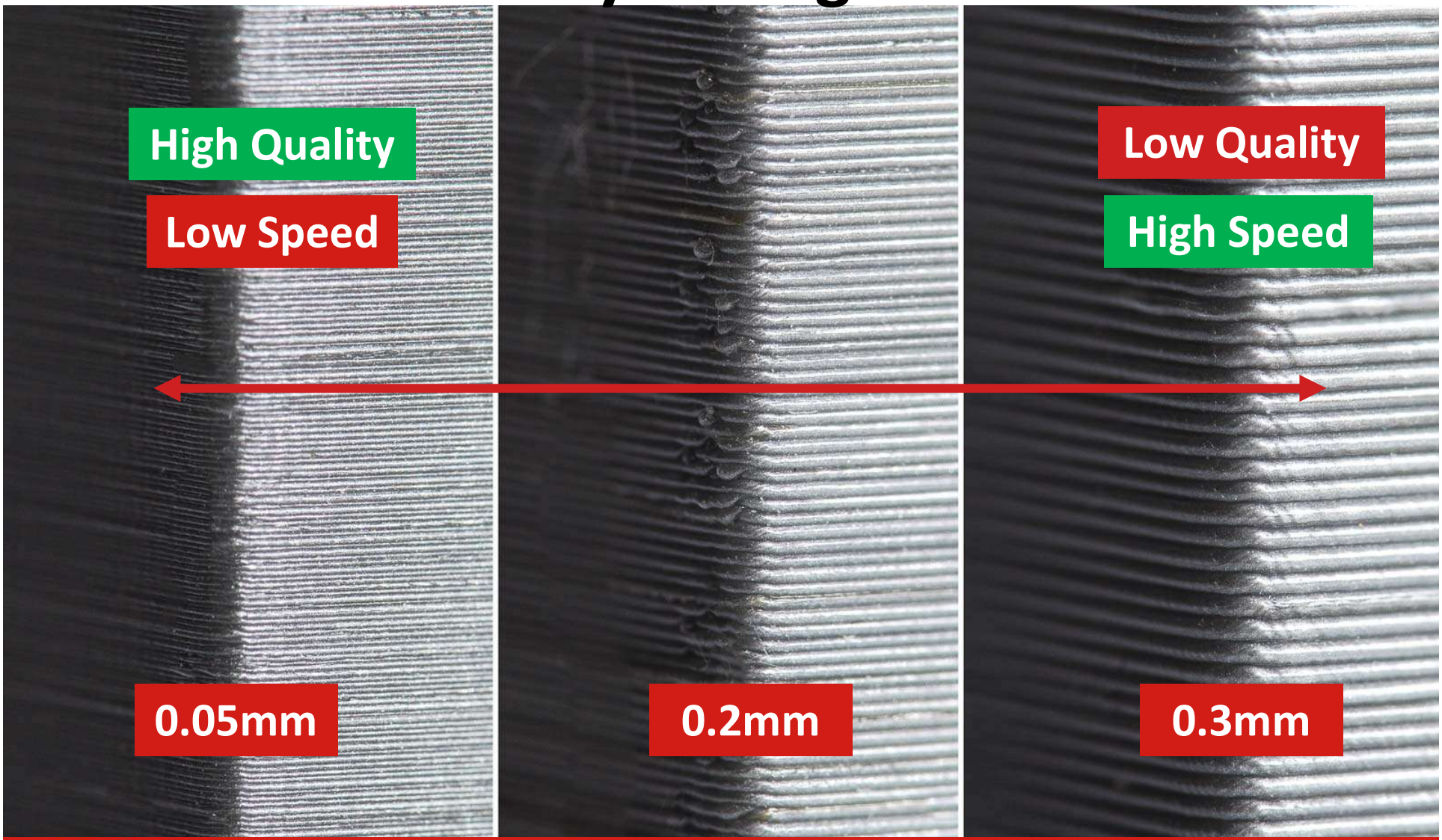
Configuration Settings

Print Setup

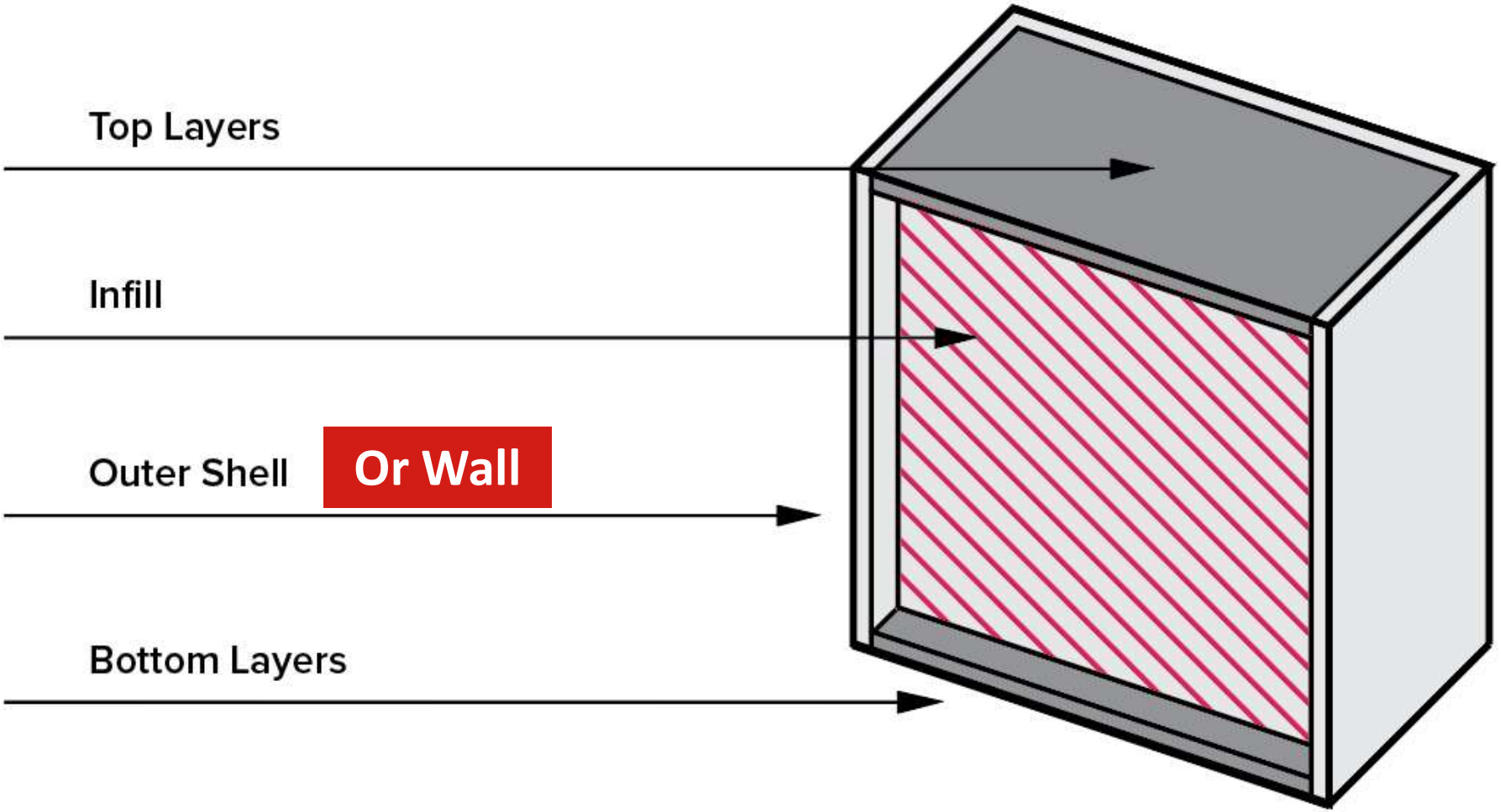


- In these drop-down menus, the print settings can be adjusted
- These settings are linked and influenced by a **Profile**
- **Profiles** are sorted by the layer height of each layer in the print…

Layer Height

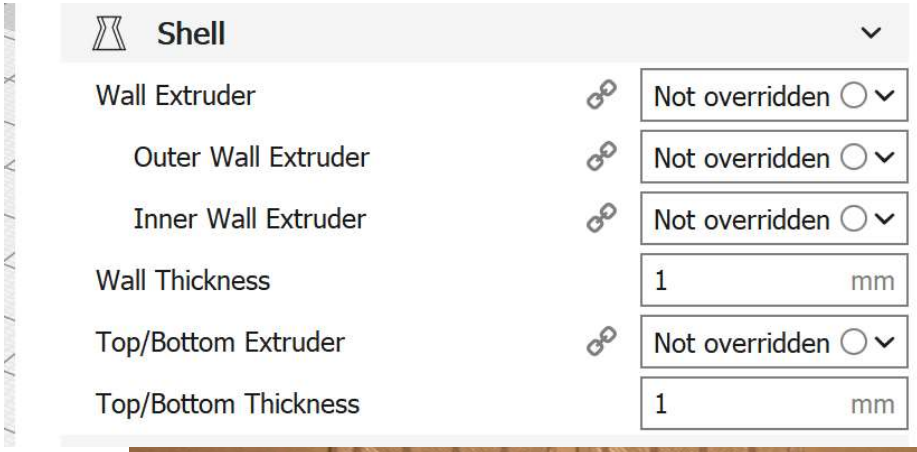


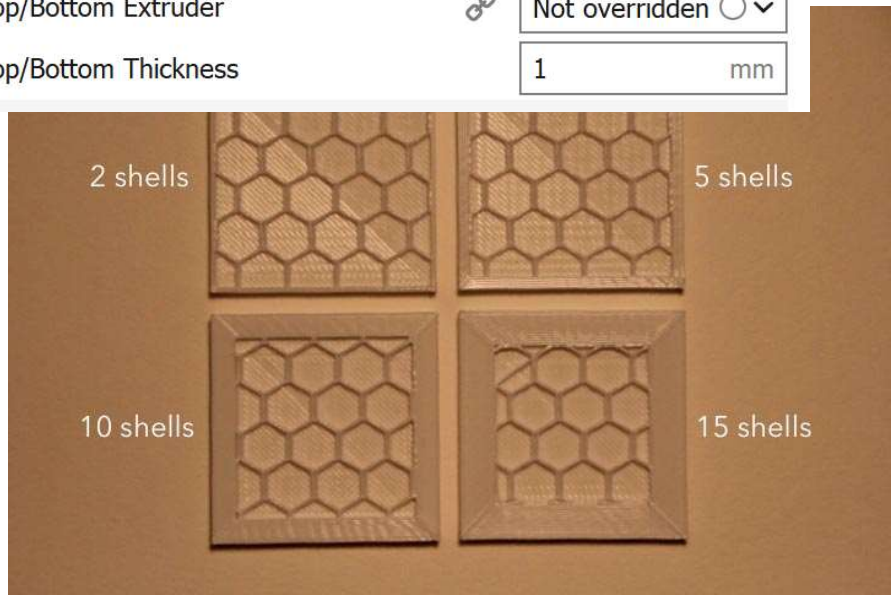
Anatomy of a 3D Print



Configuration Settings

Print Setup: Shell



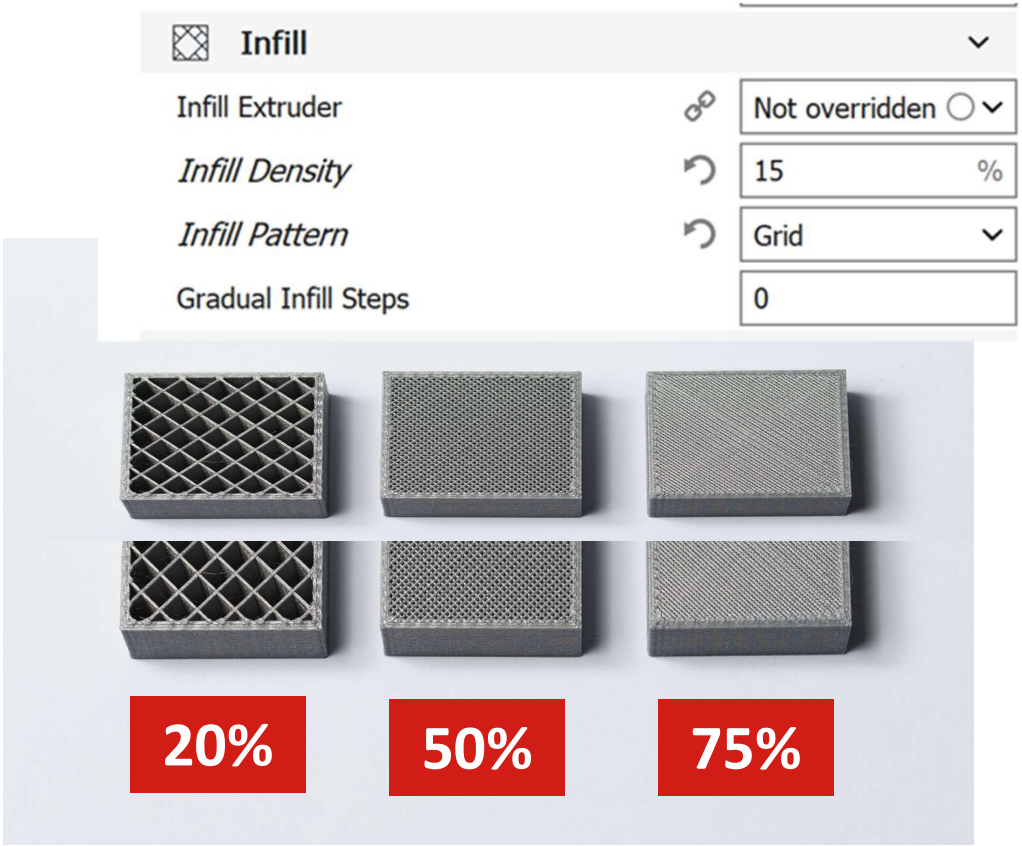


- The shell makes up the outside surface of the part, and has a defined thickness
- The **Wall Thickness** should always be a multiple of the printers nozzle diameter
- **Increasing shell thickness will increase strength of the part (but use more material)**
- **If you are drilling/tapping holes in the part, make sure the walls are thick enough**

Configuration Settings

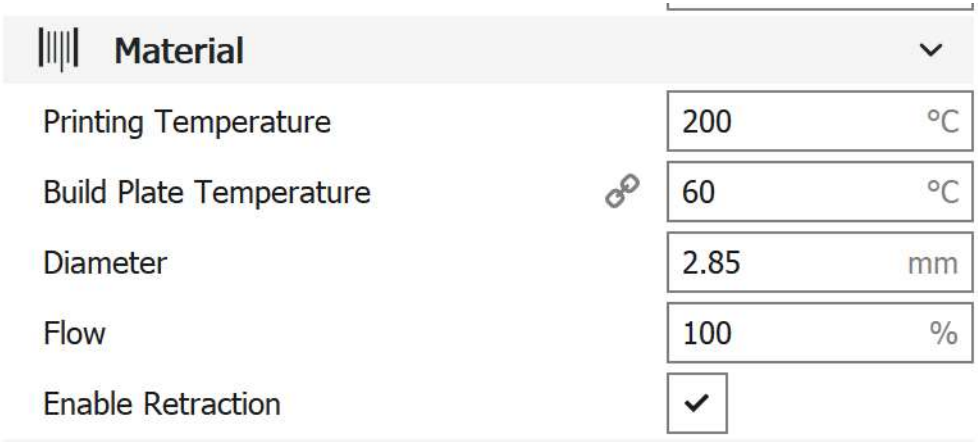
Print Setup: Infill

- **Infill** makes up the volume inside the shell of the part
- The **Infill Density** controls the overall density of the part, and thus its weight and strength
- **A higher infill density will take more time to print and use more material!**
- There are various **Infill Patterns** to explore, with different strength characteristics and build times!



Configuration Settings

Print Setup: Material Related

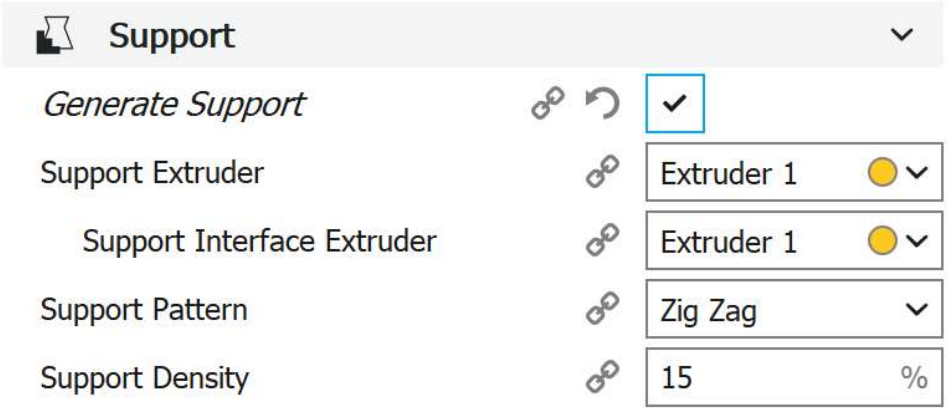


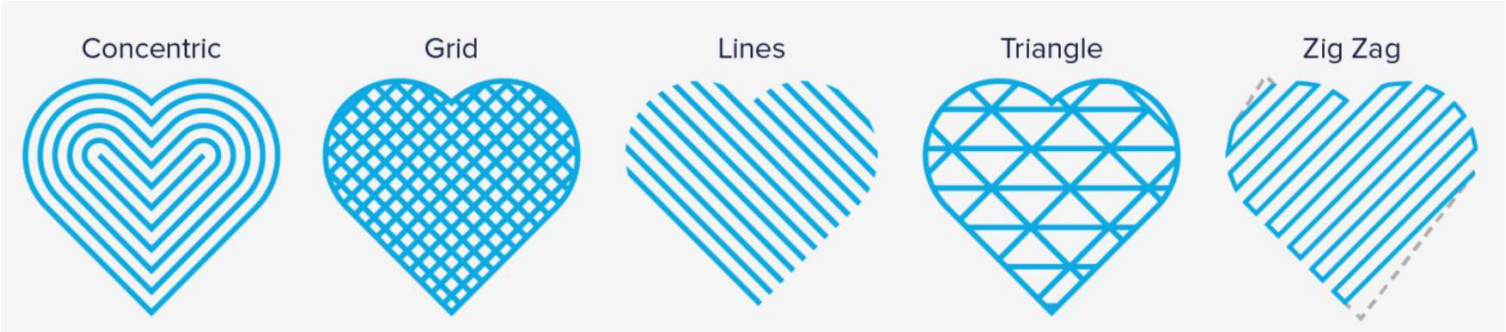
- The temperature that the nozzle reaches while printing can be adjusted
- The temperature that the build plate reaches while printing can also be adjusted here
- **These settings are linked to the print material and typically DO NOT need to be adjusted (but can be!)**

Configuration Settings

Print Setup: Support

- Supports will be generated if the box is checked
- The **Support Pattern** can be selected via drop down menu
  - (Zig Zag is quite effective)
- **Support Density** would control, in this case, how many zigs and how many zags are used to support the printed part

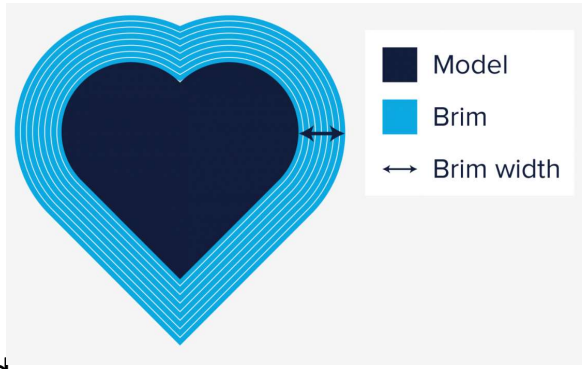


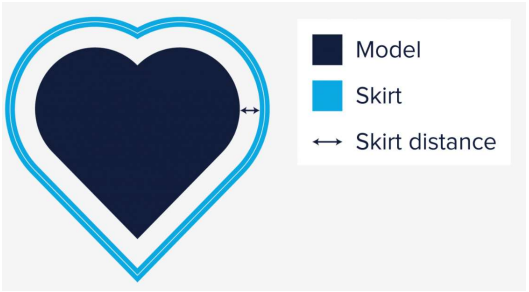


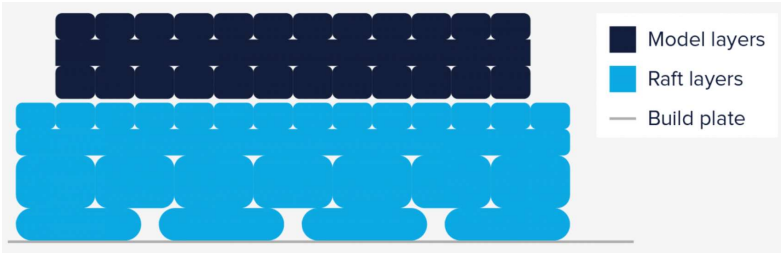
Configuration Settings

Print Setup: Build Plate Adhesion

- **Build Plate Adhesion** creates additional structures not present in the .STL file to assist the first printed layer's adhesion to the build plate
- **Brim, Raft and Skirt** are the types offered by Cura
- **Brim** prints a single layer surrounding the part's first layer to prevent warping on edges of part – should be used for longer prints that might warp
- **Raft** builds a thick interface structure between part and build plate, particularly useful for ABS prints
- **Skirt** prints a line around the part to prime the nozzle before a build – should be used for short prints with little risk of warping







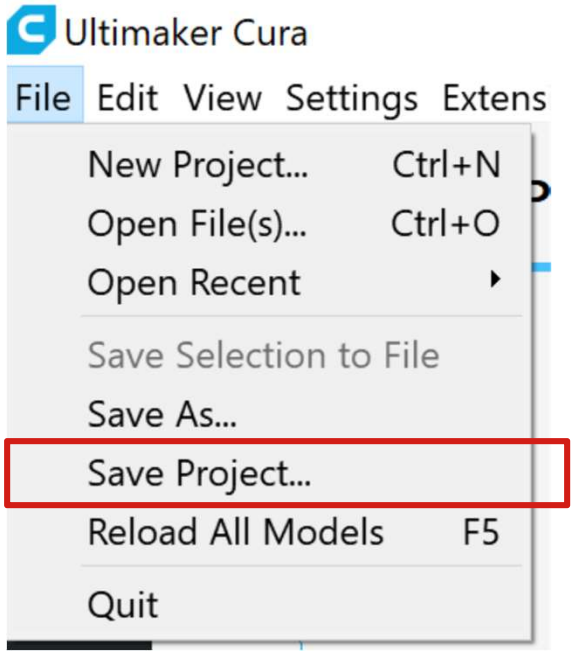
Sending File to USB

Final Steps

- The **estimated time of print** is displayed at the bottom of this menu, as well as the **length of solid filament to be used and the expected weight of the print**
- **These numbers are a result of your print settings!** Make changes accordingly!
- The file name and the dimensions of the bounding box of the print are also shown in the bottom left corner of the virtual build space
  - The file name can be changed. This will later show up on the Ultimaker 3's screen when selecting the print job
- When a USB flash drive is inserted, **Save to Removable Drive** appears. Click this to save your machine program to the external USB, then click "eject"

Don’t Forget to Save Your Project Too!

Final Steps



- Saving your project settings and not just the print job itself can be helpful if your print fails and you need to make changes or you would like to revisit the print at a later date

Filling Out a Router

Before You Start Your Print!

NJIT | Makerspace 3D Printing

Maker Information	
First Name	
Last Name	
UCID@njit.edu	

Print Job Start	
File Name	
Print Time (hh:mm)	
Print Weight (g)	
Print Material	
Filament Color	
Date (mm/dd/yy)	
Total Paid ($2/hr)	
Maker Approved	
Staff Approved	

Print Job Pickup	
Date (mm/dd/yy)	
Maker Signature	
Staff Approved	

Rate: $2.00/hr

- **You will need to fill out a Makerspace 3D Printing Router card!**
- This card will be attached to the printer to signify that you are an **authorized 3D printer** and your print is **paid for**
- Once you are done slicing on Cura, a Makerstaff staff member will mark down the **print time** associated with your print job and you will be charged accordingly
- Mount the Maker and Staff approved router card to the side of the 3D printer you are using
- Without this card attached to your printer, your print job **will be stopped!**

The Ultimaker 3 Printer & Starting a Print

NJIT | Makerspace

Preparing the Ultimaker 3 for a Print

Quick Checklist

1. Insert USB drive with exported print job on it into Ultimaker 3 printer
2. Remove solidified plastic residue from both nozzles on the printer
3. Make sure that the glass build plate is free from hairspray residue and fingerprints/oil (using I.P.A.)
4. Select the “Print” option from the Ultimaker 3 top-level menu using the jog wheel and select your print job, which should be at the top of the list
5. After the Ultimaker 3 has automatically levelled the build plate with both nozzles, it will pause to continue heating the build plate. **Spray the build plate with hairspray evenly for <u>less than 1 second</u>**

F.A.Q.

- **What if my print fails?**
  - The Makerspace will work with you to determine the cause of the print failure. If the print failing was a result of steep overhangs or long bridging sections that were unsupported (meaning your part wasn't designed for additive), then Makerspace staff will suggest areas for improvement. If the print failure was a result of the Ultimaker 3 printer, or another element outside of your control, the cost of the print will be refunded!

F.A.Q.

- **PLA or PETG?**

PLA

- Quick prototype prints
- Painting or gluing as a post-processing step
- The print requires support structures
- Environmentally friendly prints

PETG

- Durable, strong, slightly flexible prints
- Chemically and environmentally resistant prints
- High temperature tolerance required

- PLA is not as durable as PETG
- PLA supports are much easier to break off than PETG

- PETG has a temperature tolerance of about 75° C

F.A.Q.

- **How do I print with PETG filament?**
  - Let the Makerspace staff know you would like to use our engineering filament, PETG. Also make sure to **select the PETG material profile for nozzle 1** in Cura while slicing your print

F.A.Q.

- **What other filaments do you have that can be used?**

Ultimaker			
Filament Name	**Diameter**	**Color**	**Quantity (Spools)**
TPU 95 A	2.85mm	Red	1
		Black	1
		Blue	1
		White	1
PVA	2.85mm	Natural	8,5
PC (Polycarbonate)	2.85mm	Black	1
PP (Polypropylene)	2.85mm	Natural	2
PLA	2.85 mm	Black	16
		White	7
		Silver	1,5
	1.75 mm	Natural	1
PLA XTR	2.85 mm	White	1
		Maroom	1
PETG	2.85 mm	Translucent Blue	2
	3 mm	Translucent Red	1
		Clear	2
		Silver	2
PCTG	1.75		1
PCTG +	2.85mm		1
		Silver	1

These materials can only be used via a Quote Request in the 3D print lab!

Markforged X7		
Markforged X7	**Spool length**	**Quantity**
Onyx	800 cm^3	4
Kevlar	150 cm^3	2.5
Carbon Fiber	150 cm^3	3
HSHT Fiberglasses	150 cm^3	2.5
Fiberglasses	150 cm^3	2

Formlab		
Formlab Resin	**Color**	**Quantity**
Photopolymer Resin	Black	2
Flexible Photopolymer	Black	2

F.A.Q.

- **How do I remove my supports and brim structures?**
  - Carefully! For supports, these can typically be broken off by hand by gripping the supports and slowly bending them from the part until they snap off. Work your way around the part slowly. For stubborn supports, use a pair of pliers from the tool chests.
  - For brim structures, these can also be removed by hand but have a tendency to leave a few layers still attached to the part. **CAREFULLY** use a utility knife or precision knife from the tool chests to slice off leftover layers
    - **Always cut away from your body!**

F.A.Q.

- **I came to pick up my print and it isn't in the printer…where is it??**
  - After your print is completed, if you are not here to remove it from the build plate, a Makerspace staff member will remove the print for you, place it in a red plastic bin with your router card and store it at the front desk for you to pick up. Please present your student ID when you pick up your print from the front desk!

Available Training Sessions

After this training, consider attending…

- **Intro to CO_2 Laser Cutting**


We look forward
to seeing your creations!
NJIT | Makerspace