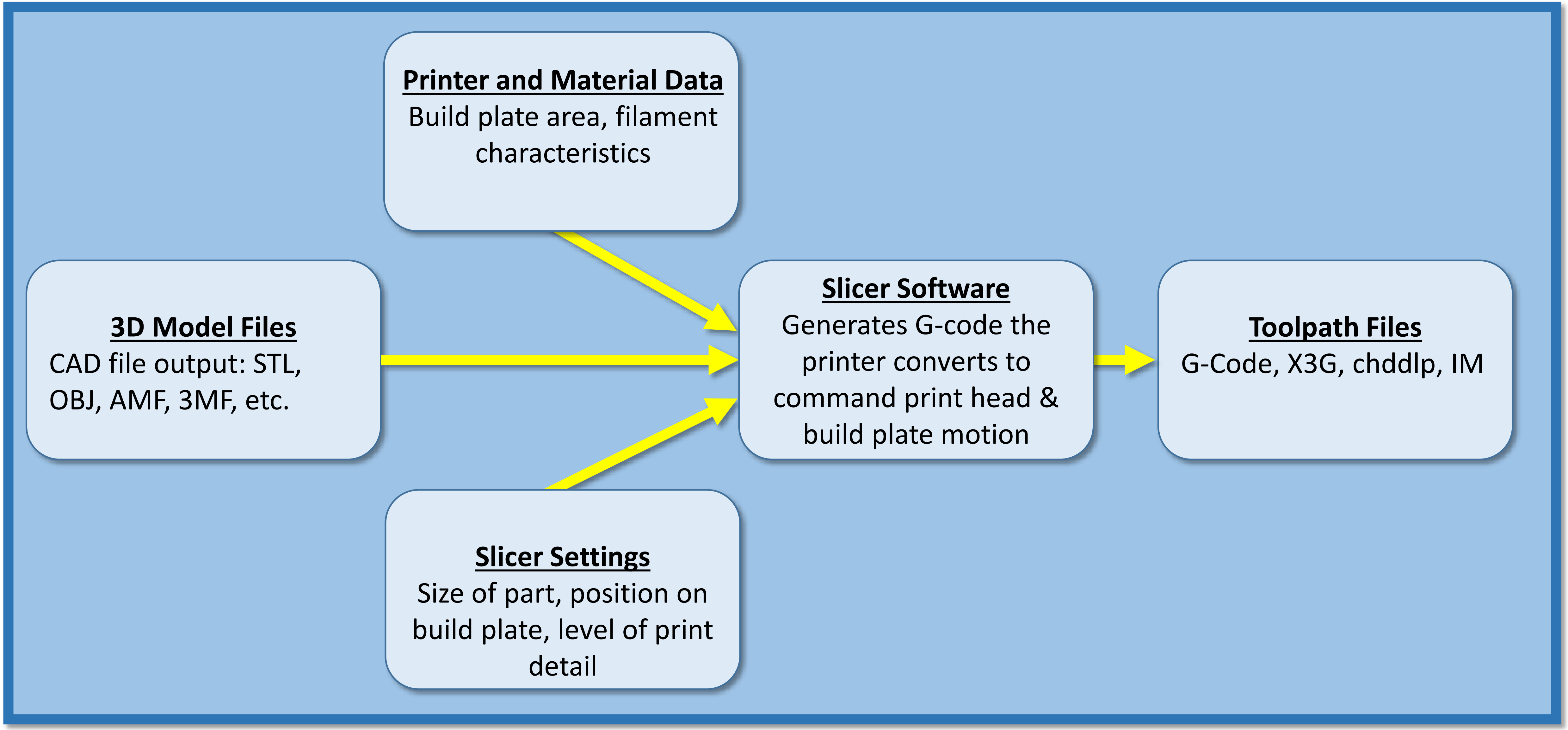
Introduction to 3D Printing

Prepared by C.C. Clarke

What is 3D Printing?

- Also known as additive manufacturing, or rapid prototyping, 3D printing is not new. Photopolymer printing (light-sensitive liquid) was patented in 1980.
- A recent uptick in the 3D printer marketplace caused by patent expiration has led to a multitude of new companies offering 3D printers, giving the false impression this is a “new” technology.
- Fused Deposition Modeling (FDM) was patented in 1989. It uses successive layers of melted plastic to create 3-dimensional objects. After each layer is printed in X and Y, the build platform lowers one layer thickness in Z to prepare for the next layer.

Simplified 3D Printing Workflow



Benefits of 3D Printing

- 3D printing is ideal for creating physical parts during the design-phase due to fast turnaround, cheap materials cost, and reduced labor costs - no machinist required!
- Up-scaled 3D printed objects are well-suited for concept demonstrations and training aids – especially when the real parts are micro-miniature and too small to see without magnification.
- Unforeseen interference issues can often be discovered and resolved prior to machining metal, saving time and cost.
- Complex assemblies can be joined with adhesive or thermally pressed-in threaded inserts.
- Precision fixtures can be 3D printed when high loading is not a factor.
- Tight tolerances allow parts to be press-fit.

Step 1: Modeling

- The process starts with a 3D model. It is produced using CAD or dedicated modeling software. The 3D model is a digital description of an object’s surface shape and dimensions.

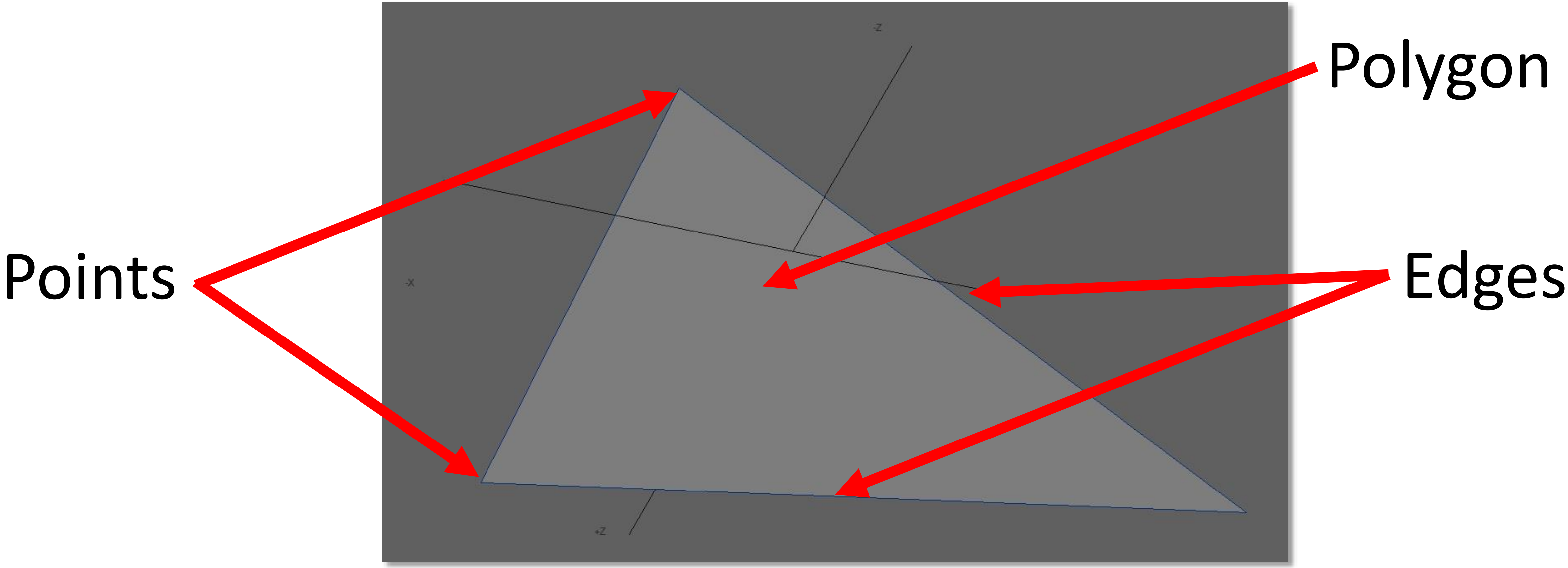
- There are many different file formats used to describe 3D models, but to be useful for 3D printing and to simplify file sharing, the files need to be open-source or neutral formats rather than proprietary formats specific to a particular software app.

Step 1: Modeling

- To be printed, polygons defining the object's shape are made of triangles.
- The method of creating a polygonal surface varies but sometimes does so at the expense of being precise. For example, a CAD program might describe an exact sphere of a given radius centered on stated coordinates, but the equivalent mesh would be only an approximation of this shape.
- The most popular 3D model file is the STL format, which uses a mesh of triangles to define surfaces.
- Thousands of STL files can be found on sharing sites such as Thingverse, MyminiFactory, and Cults. Increasingly, these sites support other 3D model formats.

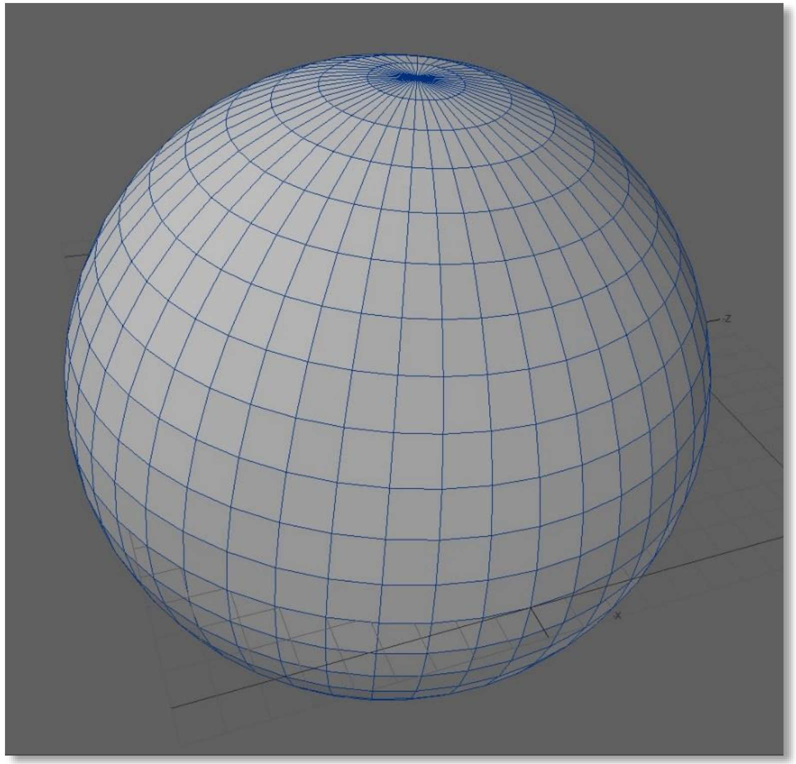
Step 1: Modeling – Anatomy of a Polygon

- Polygons are the building blocks of all 3D objects.
- Their shapes are defined by Points (vertexes), and Lines (edges).

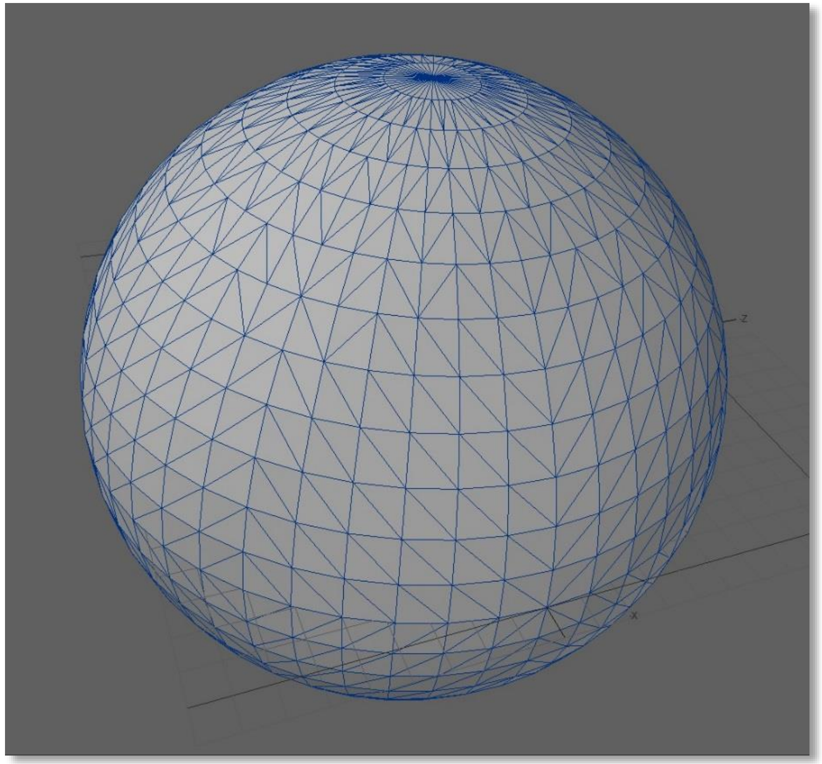


Step 1: Modeling – Polygonal Equivalents

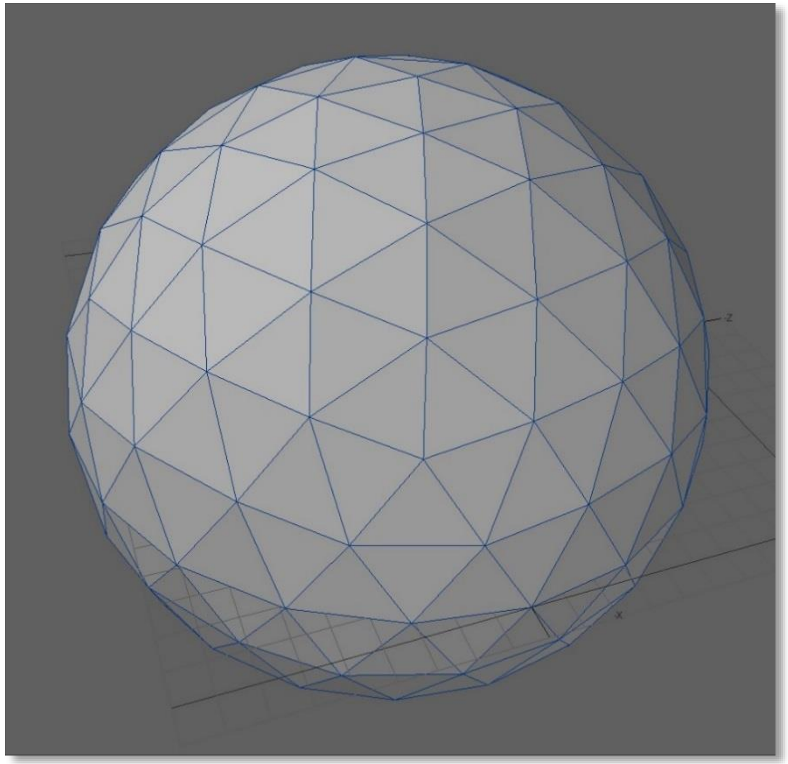
Each sphere is the same size, but the polygonal type defines the quality of printed detail.



Quads – Four-Sided Polygons



Triangles – Three-Sided Polygons
(Standard format for slicing)



Tessellated – Three-Sided Polygons
(Less surface detail)

Step 2: Slicing – How Layers are Defined

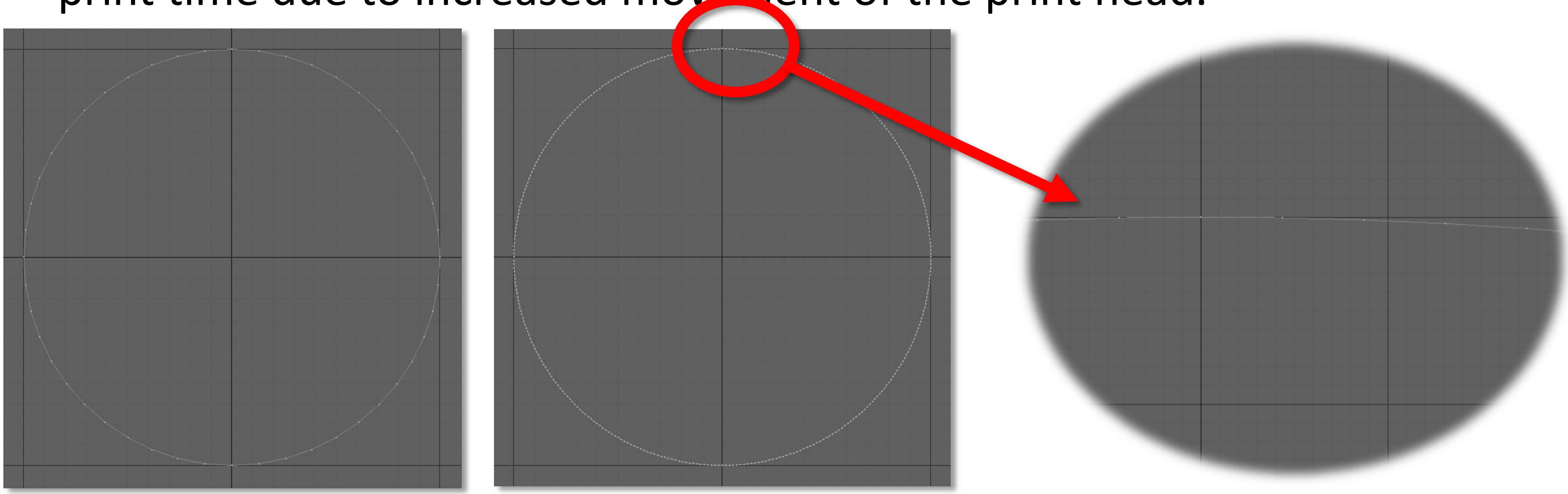
- The next step in the workflow converts the 3D model into something printable using slicer software. The slicer requires information about the characteristics of the printer, (size, speed, and method of printing) filament properties, and other user-selected settings, (quality, supports, and scale, among others).
- ideaMaker, Cura, Simplify3D, and Octoprint are popular slicers. We use ideaMaker, written by Raise3D.
- After configuration, the software processes the information and the output is a set of instructions for the 3D printer to produce a physical item. The instructions are written in G-code.
- All of the G-code toolpath files are specific to the machines and materials they were generated for. They may not work on other machines and may even damage them.

Step 2: Slicing – Material and Print Time

- A 3D printer prints only line segments. A curve is interpreted as a series of interconnected straight lines.
- The total number of line segments in each layer determines how much filament is used.
- The feed and retraction speeds for the total length of filament determine how long the print will take to complete.
- Filament quantity, cost, and estimated printing time is based on line segment calculations performed by the slicer.

Step 2: Slicing – Detail vs Print Time

- In this example, a circle's circumference is made of a series of connected line segments. The greater the number of line segments, the smoother the curve for circles of the same diameter. More line segments increase the print time due to increased movement of the print head.



48-Sided Circle

360-Sided Circle (Same Size)

Line Segment Close-up

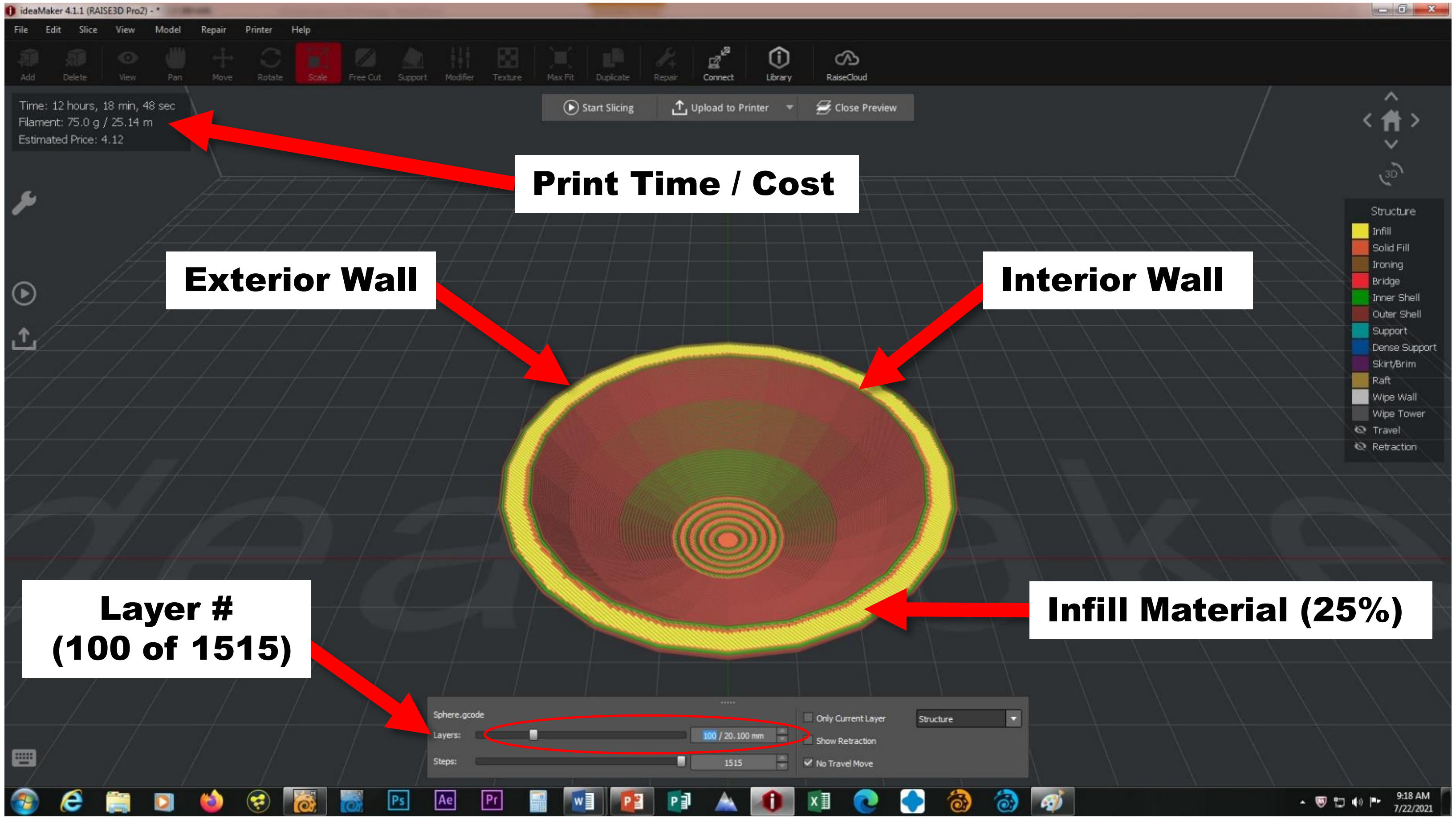
Step 2: Slicing – Detail Considerations

- It is essential to understand the relationship between the slicer and 3D model when it comes to the quality of the finished part.
- A model with more faceting (large space between line segments) produces less detail and noticeable flattening on curved surfaces.
- The slicer only works with the 3D model it is given. It cannot change the 3D model to “smooth” it out.
- A slicer can determine the surface finish (as opposed to detail) of a part.
- Printers can lose fine detail if the filament feed speed is too fast. (More on this to follow.)

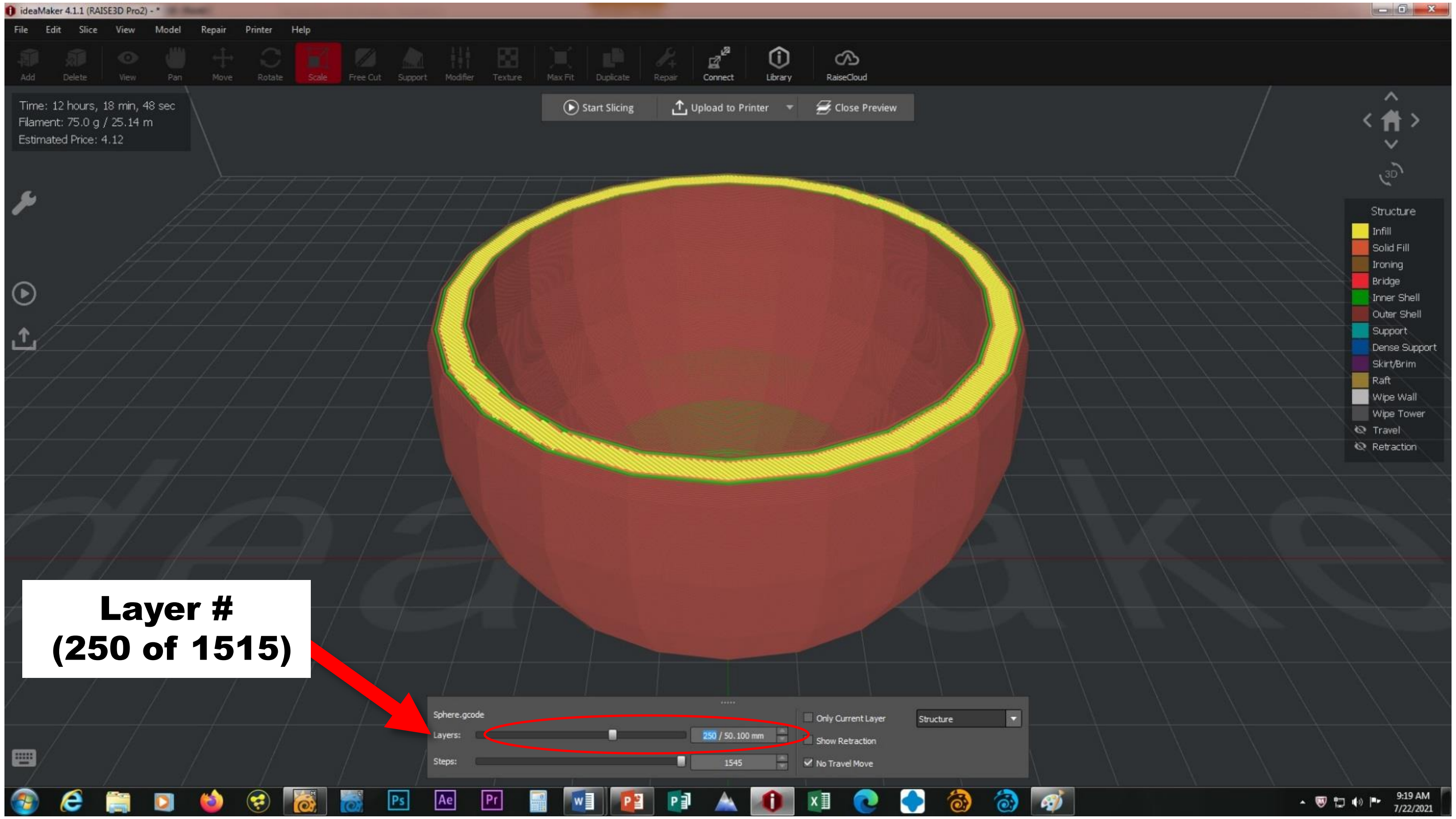
Step 2: Slicing – Terminology

- Infill: The amount of material between the interior and exterior walls. A solid cube would have 100% infill. Infill is the biggest contributor to print time. 15% is a good compromise between strength and print time. When part strength is required, 100% infill is recommended.
- Supports: An unsupported layer requires something to adhere to or it will sag. If the angle of the part is greater than 30°, it is advisable to use support structures or print quality will suffer --or the print may fail.
- Raft: A flat base printed between the build surface and the printed object used to promote adhesion, add support, and prevent warping of high-temp filament. The layer height is greater, (rather than squished) so the part will be easier to remove from the raft. Rafts leave a rougher surface finish.
- Brim – An raft that uses less material to print when a full-sized raft isn't required. It is attached to the exterior edge of an object and prints much faster.

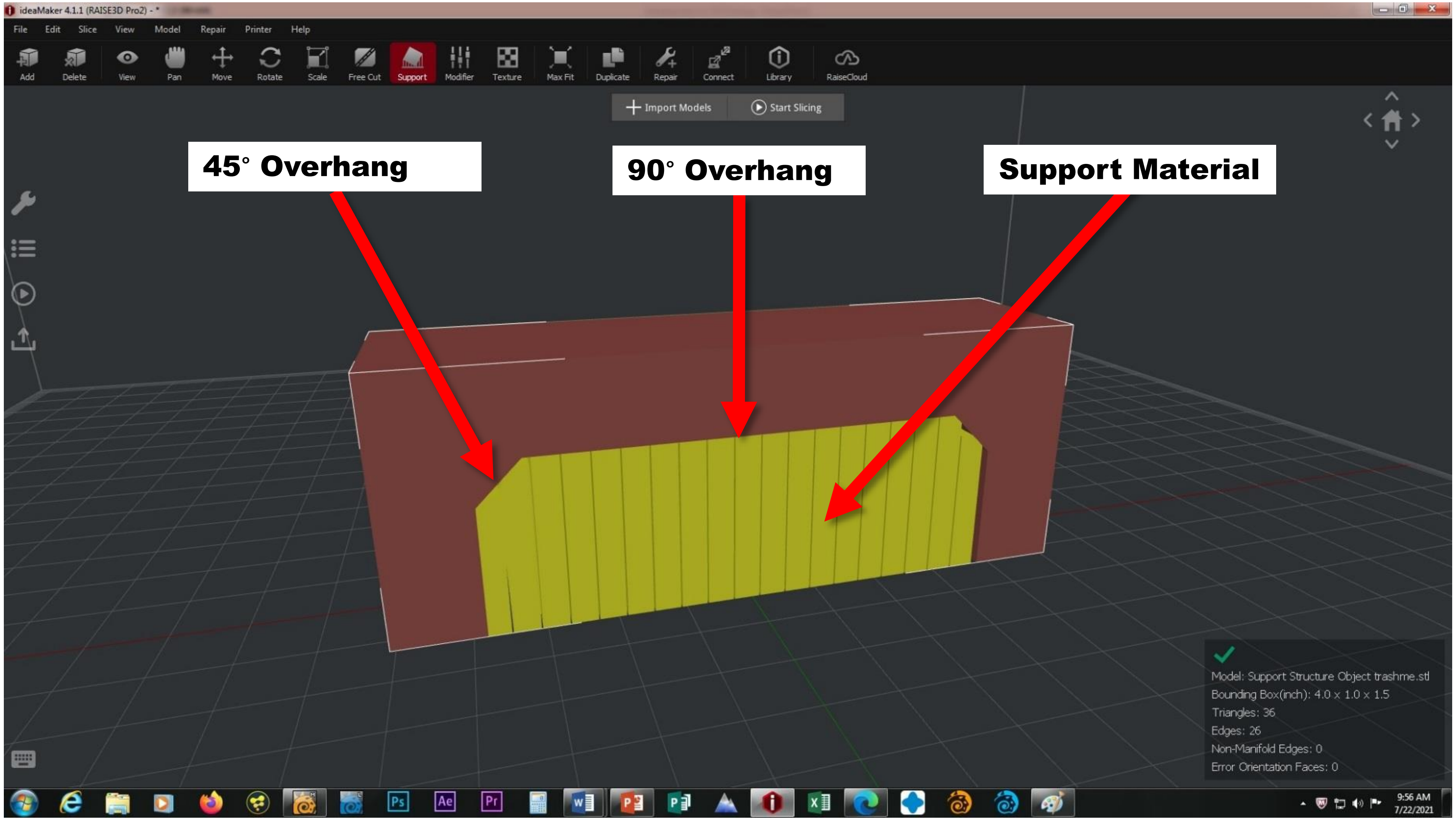
Step 2: Slicing with ideaMaker



Step 2: Slicing with ideaMaker - Preview

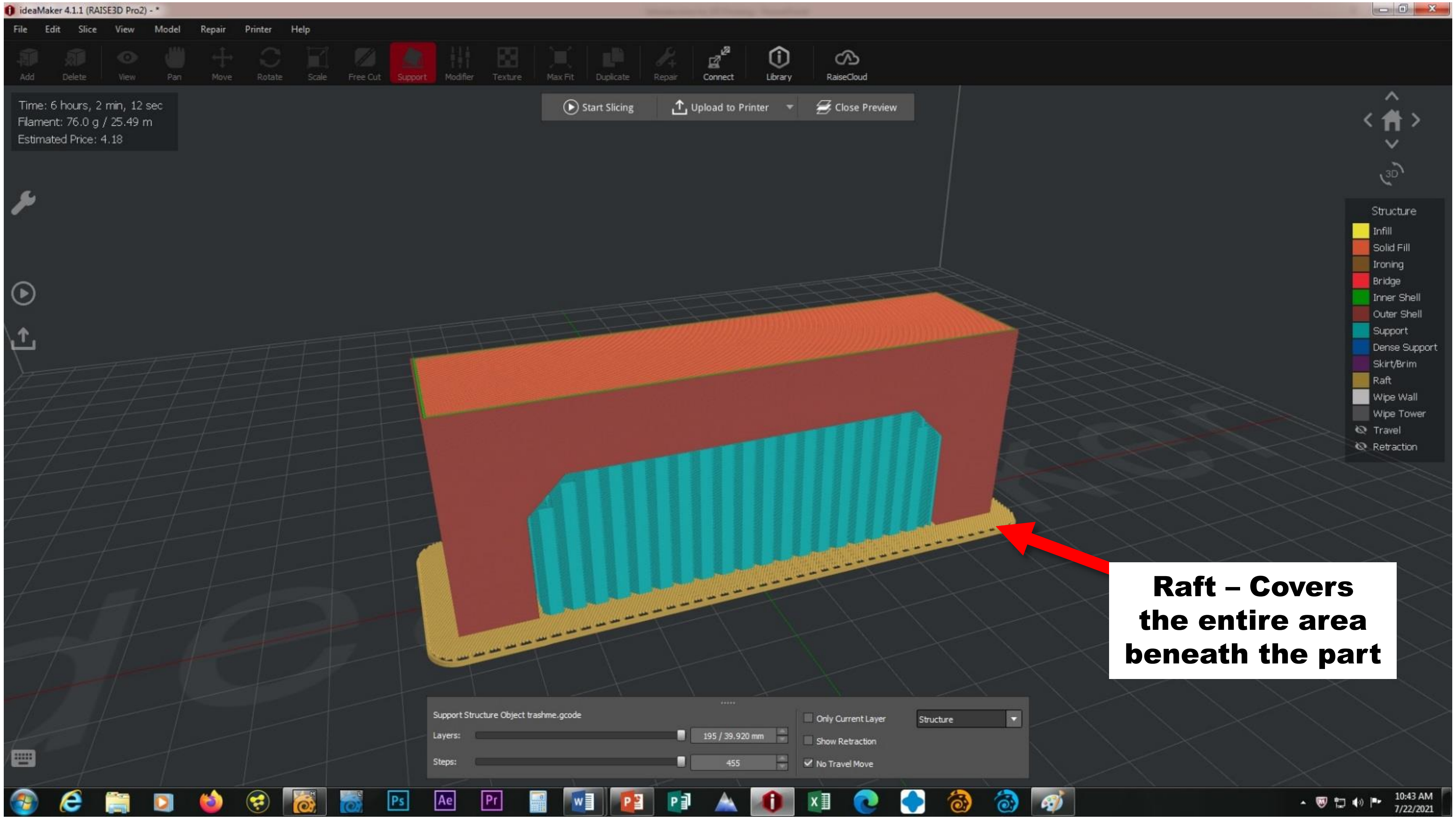


Step 2: Slicing with ideaMaker– Support Structure

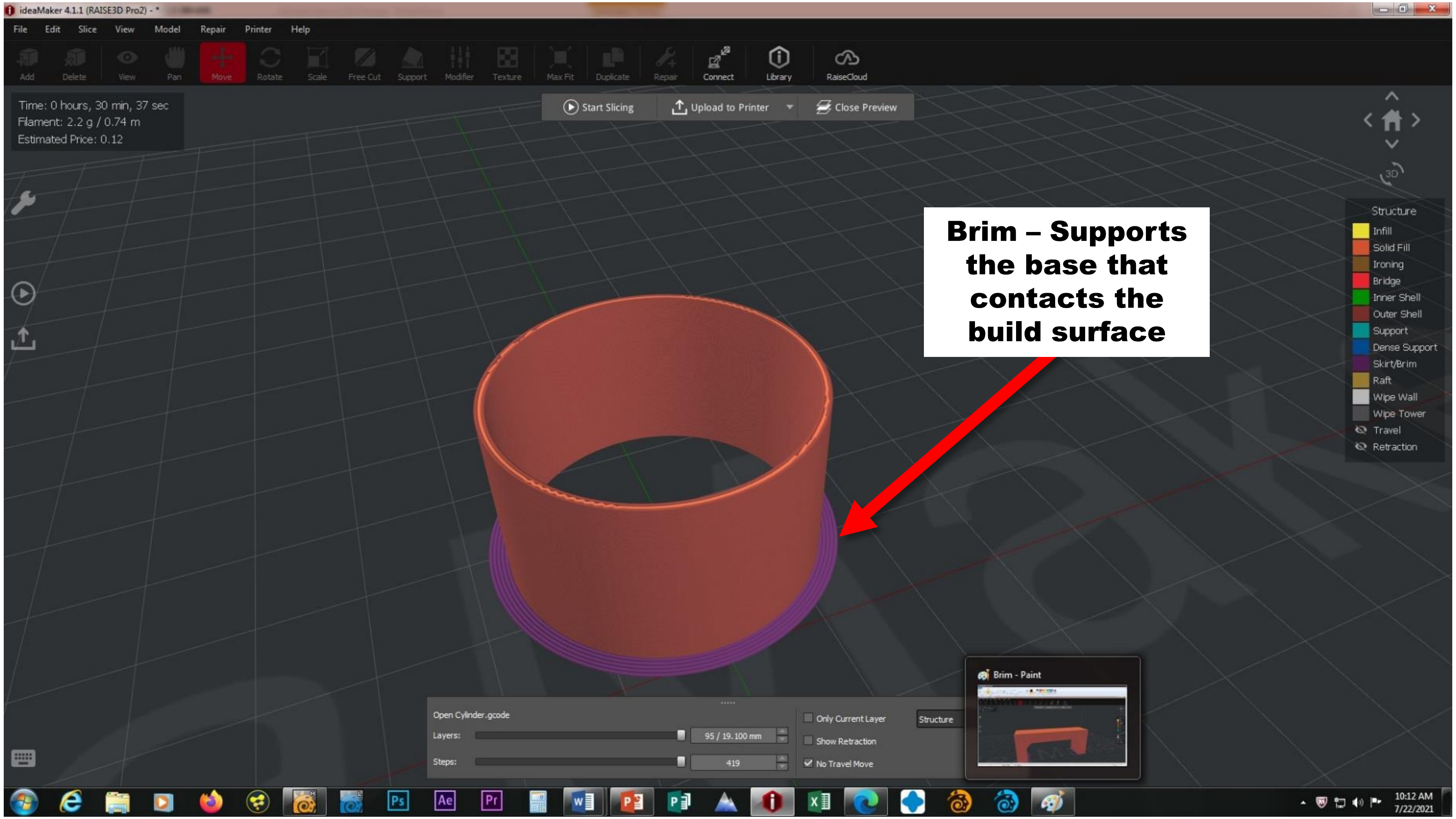


Without supports added, this part would fail to print properly.

Step 2: Slicing with ideaMaker – Raft & Supports

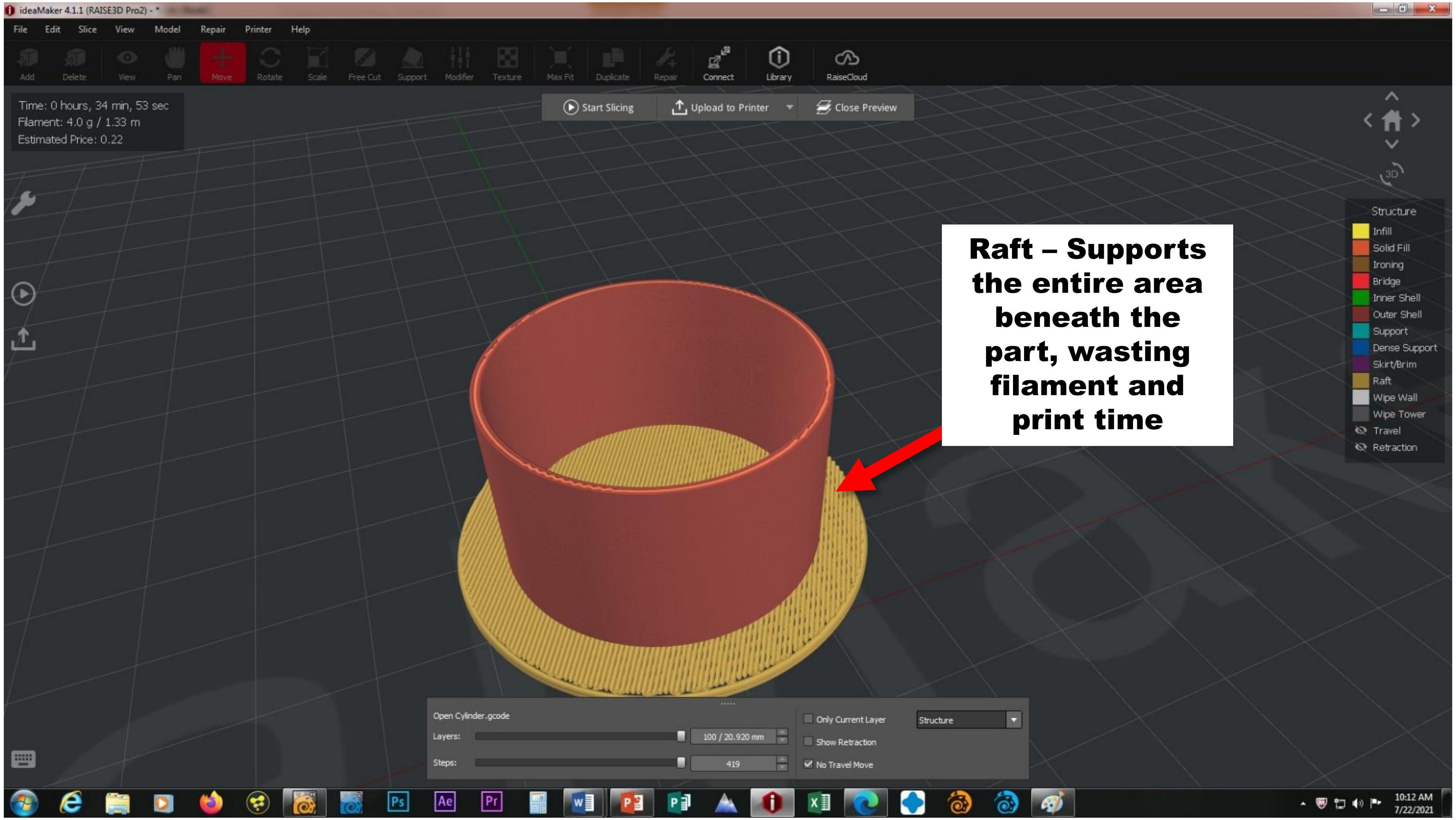


Step 2: Slicing with ideaMaker – Brims



Brim – Supports the base that contacts the build surface

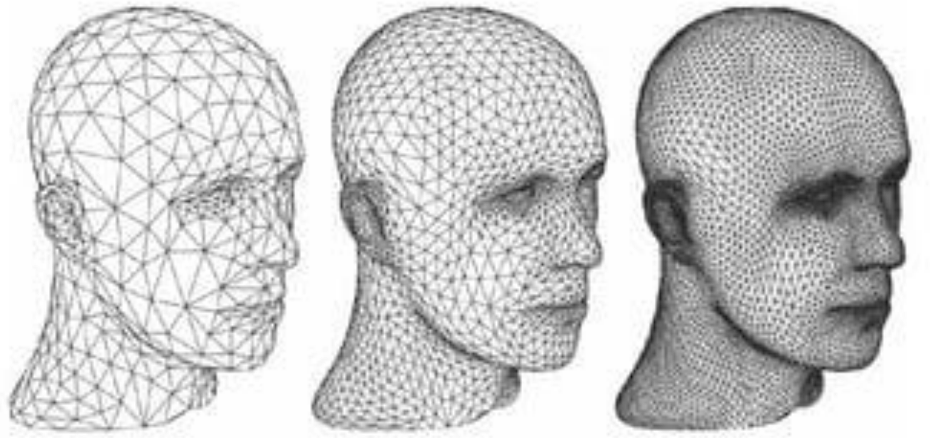
Step 2: Slicing with ideaMaker – Rafts vs Brims



Choosing the correct support type impacts print time!

STL : The De-facto File Sharing Format

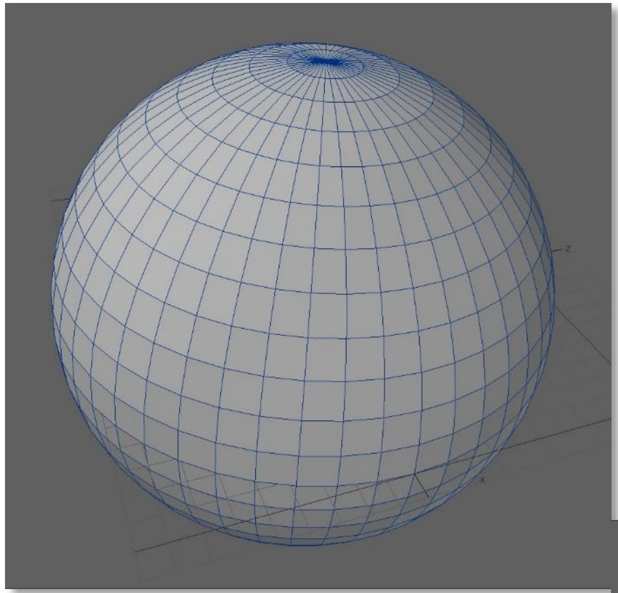
- STL stands for stereo-lithography, one of the earliest forms of 3D printing.
- The STL format was invented in 1987 by 3D Systems, a company founded by 3D printing pioneer Chuck Hull.
- STL files describe 3D objects as a mesh of tessellated triangular polygons, each defined by the coordinates of its vertices.
- For this reason, STL is often referred to as “Standard Triangle Language” or “Standard Tessellation Language”.
- A complex model with curved shapes can be made up of tens of thousands of triangles. Adding more triangles improves the resolution of the shape but creates larger STL files and takes longer to print.



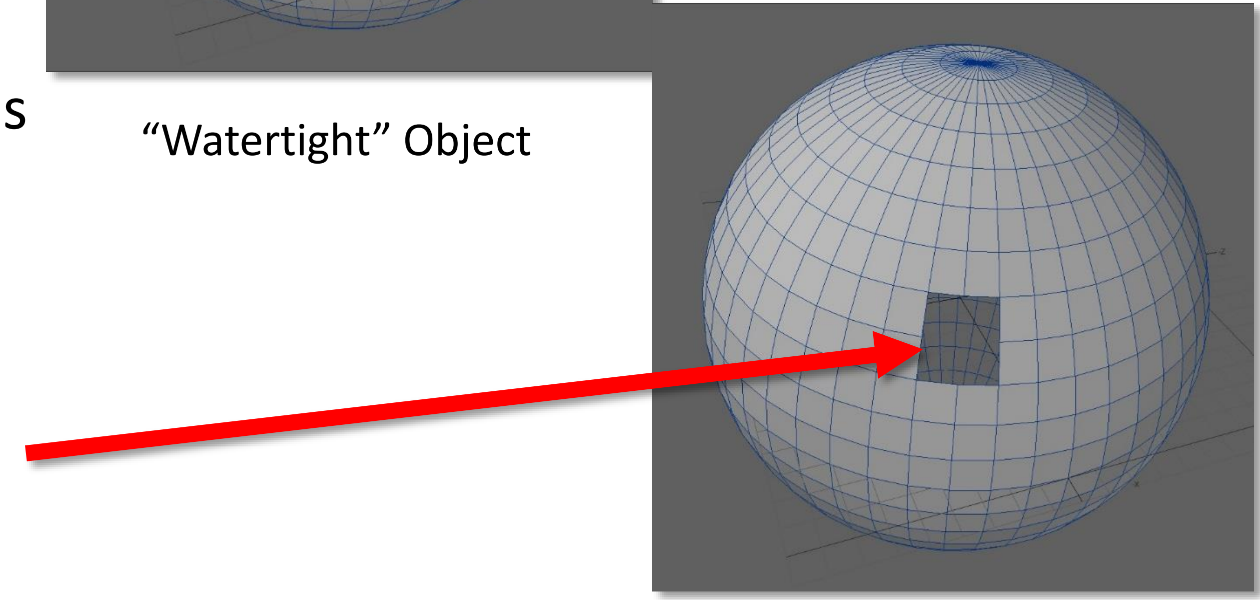
STL : The De-facto File Sharing Format

- In addition to the file size issue, the STL format has several major drawbacks. It does not enforce checks on geometry or explicitly link one triangle with another. The resulting "triangle soup" can be slow to process and error-prone.

- STL files with holes or gaps (exterior surfaces are not "watertight") cannot be sliced / printed properly because the geometry cannot exist physically.
- The missing polygons allow the observer to "see through" the other side. This cannot happen in the real world. A slicer cannot repair this.



"Watertight" Object



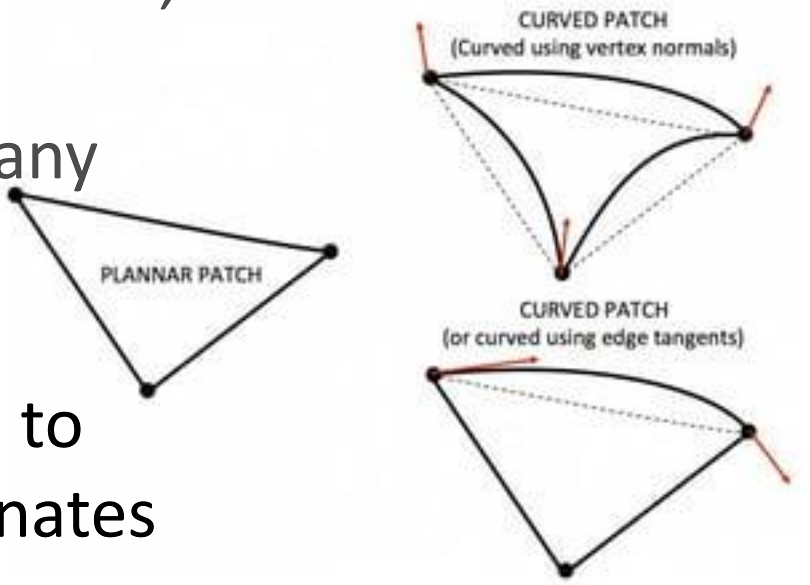
Object with Missing Polygons

OBJ: The 2nd Most Popular Format

- The OBJ format, also known as Wavefront Object, was first defined in the 1990s by Wavefront Technologies and tried to improve on the STL format in several significant ways.
- Like STL, OBJ is an open-source and neutral file standard with widespread software support and availability of existing models.
- OBJ files usually have improved model resolution and therefore higher surface quality over STL files.
- OBJ files provide no scale information. This must be defined in the slicer.

AMF: Dubbed, "STL 2.0"

- AMF, short for "Additive Manufacturing File", was introduced in 2011 and positioned as "STL 2.0". It's an XML-based format designed to be faster, less error-prone, and capable of storing color, material, and texture information (without using external files), among many other attributes.

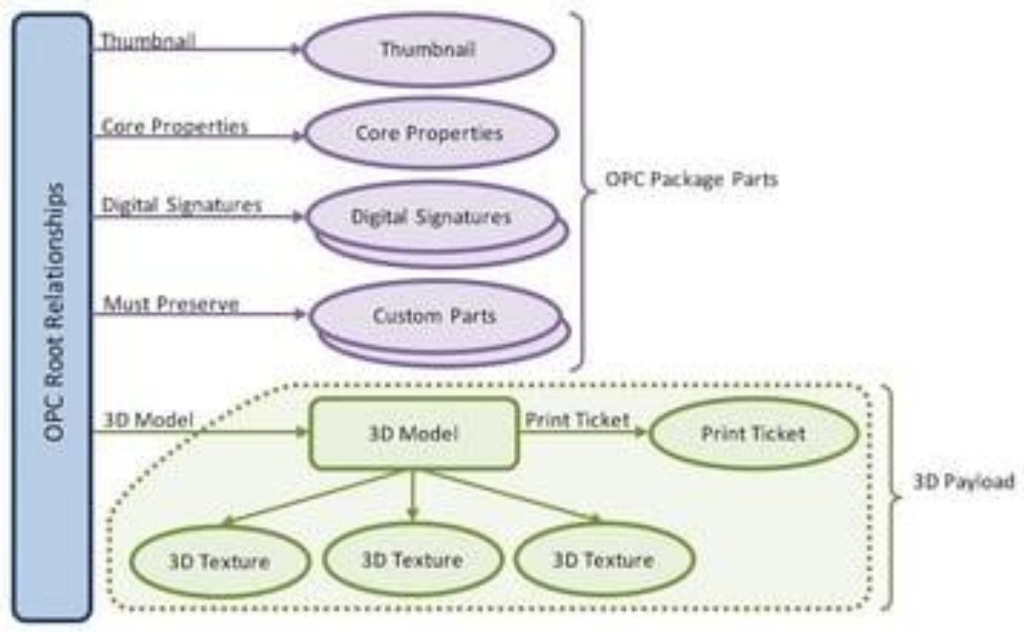


- The format uses curved triangles that must conform to strict rules of topology and linkage; this nearly eliminates a host of model issues that plague STL in particular.

- Adoption, however, has been slow, and its future is uncertain.

3MF: A Potential New Standard

- Microsoft established the 3MF Manufacturing Consortium) in 2015.
- Open standard with strong industry backing.
- Can incorporate security and encryption for parts of models that are IP sensitive.
- Is regarded by some as too complex for simple applications, meaning that adoption by entry-level CAD programs is relatively slow.



G-Code – Too Useful to Die

- As mentioned previously, the output of the slicer is a file containing toolpath information to instruct the printer (a tool) what to do. The most common toolpath file format is G-code.
- G-code is even older than STL but has stood the test of time better. Development work started in the MIT Servomechanisms Laboratory in the late 1950s, and the first standardized version was agreed to in the 1960s.
- G-code is used to run many types of Computer Numerically-Controlled (CnC) equipment and comes in various dialects to support different machine types.

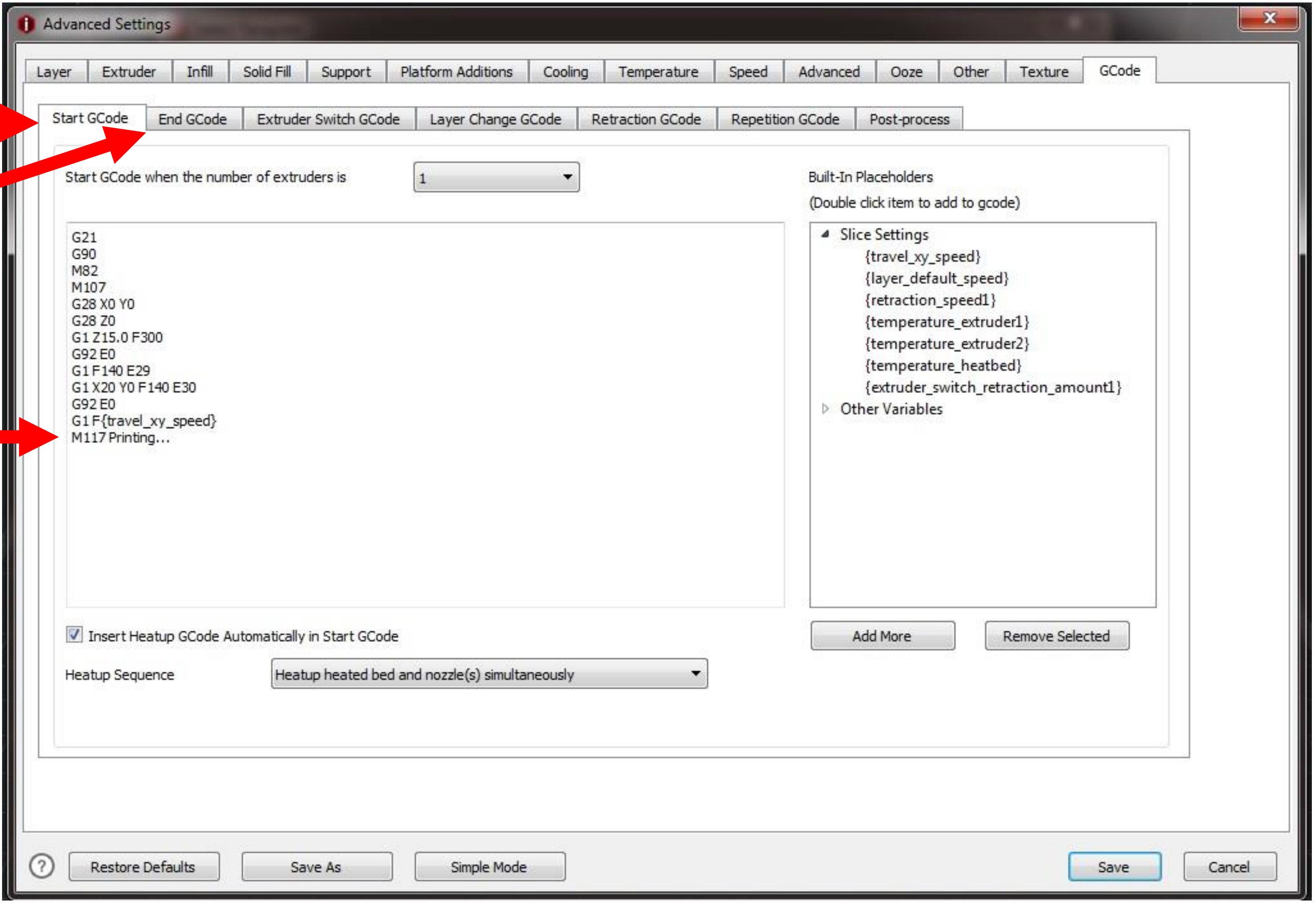
G-Code

- For FDM 3D printing, the most significant factor has been the emergence of standard firmware (Marlin, for example).
- Raise3D printers use a proprietary variation of Marlin.
- The firmware inside a 3D printer reads the G-code file and executes each command in order, sending control signals to stepper motors, fans, heater blocks, and other printer components.

G-Code – ideaMaker Example

- 2 Main Groups of G-code are used in ideaMaker : Start and End Printing Commands

- Start G-Code: Prepares the 3D printer to print from a known state.
- End G-Code: Returns the printer to a known state after completion of the print.
- The code, “M117” executes the converted STL (part) file.
- G-Code begins with a letter followed by a number.



Filament

- A continuous length of plastic wound on a plastic spool, melted in successive layers to form a part.
- Spools of filament typically weigh 1 kg.
- Filament diameter is commonly 1.75 mm.
- 100-140m per spool.
- Filament diameter swells with humidity and becomes brittle.
- Brittle filament can fracture while feeding.
- “Wet” filament, steams in the nozzle, leaving contamination and the potential for clogging.

Dual-Spool Configuration on Raise3D Pro Series Printers

Filament - Types

- Plastic filament comes in a wide variety of materials and colors.

- The majority of filament we use is PLA, (Polylactic Acid) which smells like popcorn, (derived from corn oil.) It will eventually degrade when exposed to moisture.
- PLA has a low melting point and is environmentally safe to work with. PLA warps easily when exposed to heat – especially in direct sunlight.

- ABS (**Acrylonitrile Butadiene Styrene**) should be used with an approved hood to ventilate the toxic fumes.
- ABS has a higher print temperature, is more impact-resistant, and is less affected by UV rays, so it is ideal for parts that will be used outdoors or high-heat applications.
- ABS requires an enclosure to contain heat to minimize warping during printing.

Parts of a 3D Printer – The Build Plate

- The Build Plate is the movable “bed” or base on which the part is printed. It moves down one layer height after each “slice” is printed to prepare for the next layer. The replaceable top surface the part adheres to is called the build surface. It costs approx. $70 a sheet and can be easily damaged by metal scraping tools when removing a firmly-adhered part.

- The aftermarket steel, flexible build plate/surface shown on the left requires no tools to remove the part and greatly extends the life of the Build Surface. It is secured in place to the stock metal Build Plate (on the right) with magnets.

The Extruder

- The Extruder consists of a stepper motor that precisely pulls the filament from the spool into the top and feeds it downwards into the hot end where the filament melts before exiting the nozzle.

- A tension screw (located on the side) adjusts the amount of force required to move the filament by pushing it against a concave gear inside. Too little tension and the filament will skip. Too much tension and the filament will bind, grind, and eventually stop.

- During pauses between layers or gaps in the part on the same layer, the extruder "retracts" or pulls the filament upwards in small increments to prevent it from leaving the nozzle and damaging the finish of the part. (Called "drooling".)

- Feed and retraction speed are the main extruder settings adjusted within the slicer.

- A sensor on the top pauses the print if the filament runs out.

A typical BondTech dual-extruder (note the offset stepper motors) used in the Raise3D Pro series.

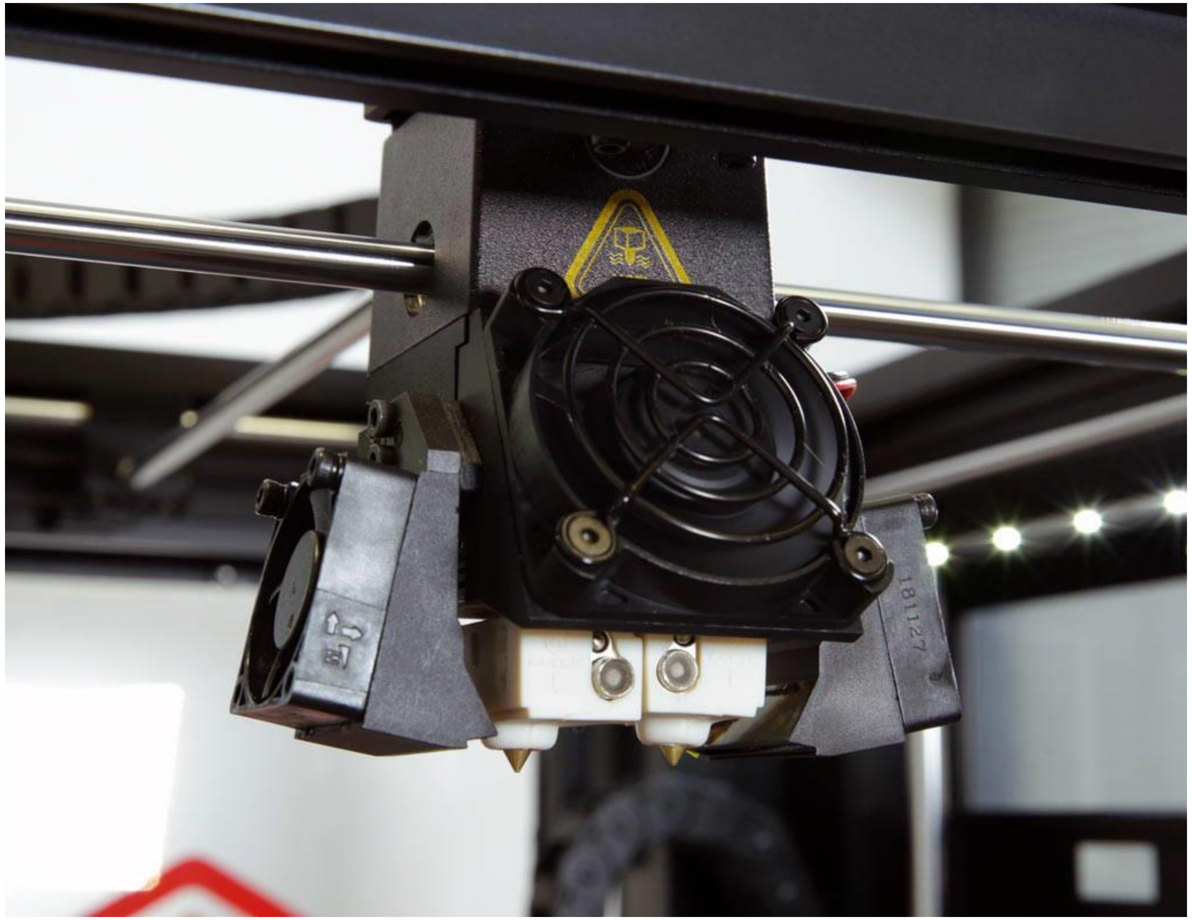
The Hot End – The Working End of the Printer

- Raise3D printers use dual extruder / hot ends for printing two types of filament.

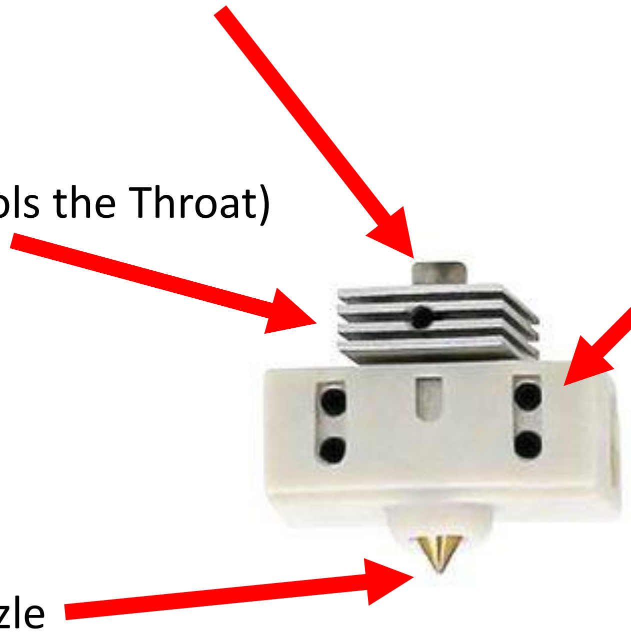
- A nozzle fan at each side cools the melted filament right after being deposited.

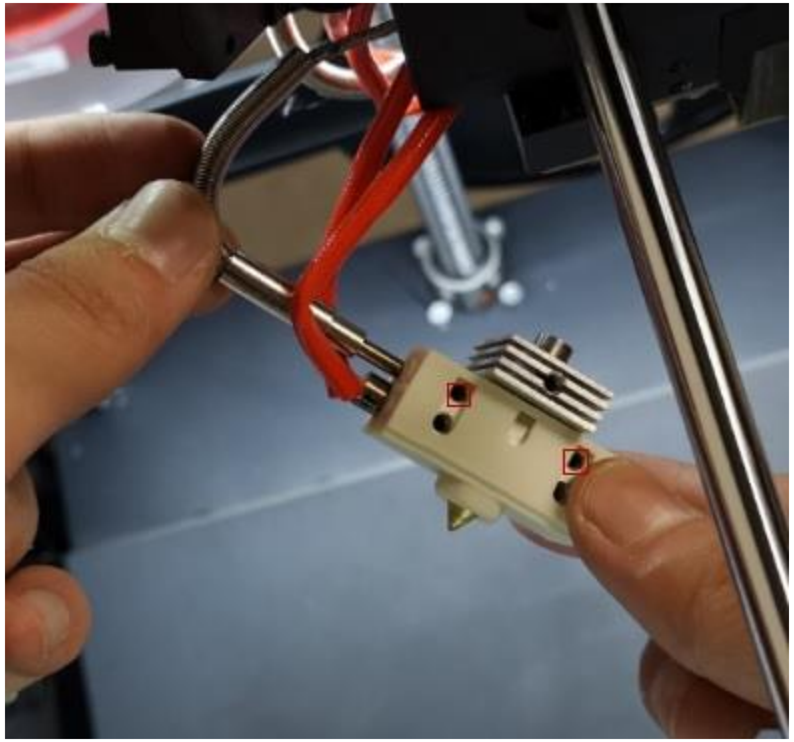
- A larger, center fan cools the heat sinks on top of the hot end where the filament enters to prevent softening of the filament as it is extruded.

- Without cooling, the softened filament cannot be pushed downward into the nozzle chamber and will result in a clogged path, also known as "air printing" where the print head and build plate continue to move but no filament exits the nozzle.



The Hot End – The Working End of the Printer



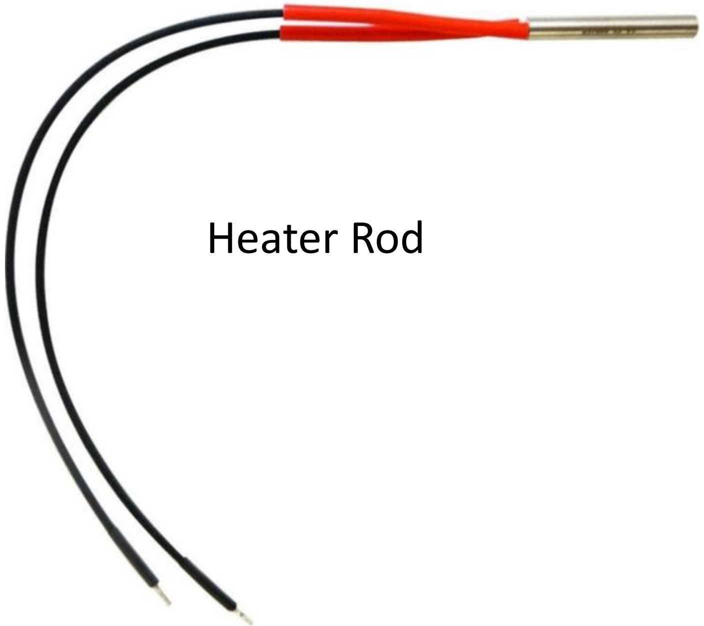


Hot end with Heater Rod installed and Thermocouple being adjusted

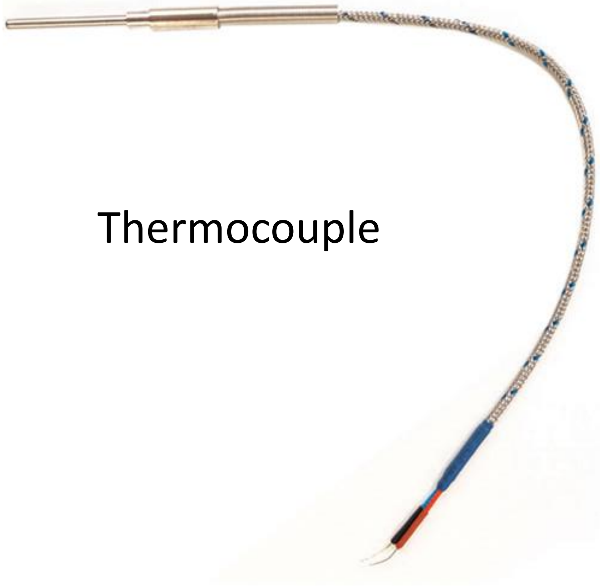
- A heat block with a nozzle and throat installed under a silicon insulating cover. (Heater rod and thermocouple omitted.)

The Hot End – The Working End of the Printer

- The Heater Rod heats the aluminum block to the desired temperature required to transform the filament from a solid to liquid state.
- The Thermocouple measures the heater block temperature and provides feedback to the Control board to maintain the set temperature.



Heater Rod



Thermocouple

The Nozzle – The "Tip of the Spear"

- The nozzle directs the liquid plastic onto the build surface.
- Nozzles have different bore (hole) sizes which determine the diameter of the extruded filament.
- Larger bores print faster, (more filament exits the bore) but print less detail due to thicker layers.
- The size of the bore is usually stamped on the side of the nozzle.
- Typical nozzle bore size is .4mm.
- Hardened nozzles are required for abrasive filaments.

Printer Calibration for Best Quality Prints

- Two settings must be completely understood to consistently obtain high-quality prints:

1. Layer Height – Initial height of the nozzle from the build surface. This is manually adjusted prior to printing and should only be changed if the build surface is replaced, mechanical changes are performed on the print head, or print quality degrades.

2. Layer height is generally ½ the bore diameter.

3. Print Speed – How fast the hot bead of plastic is applied to the build surface.

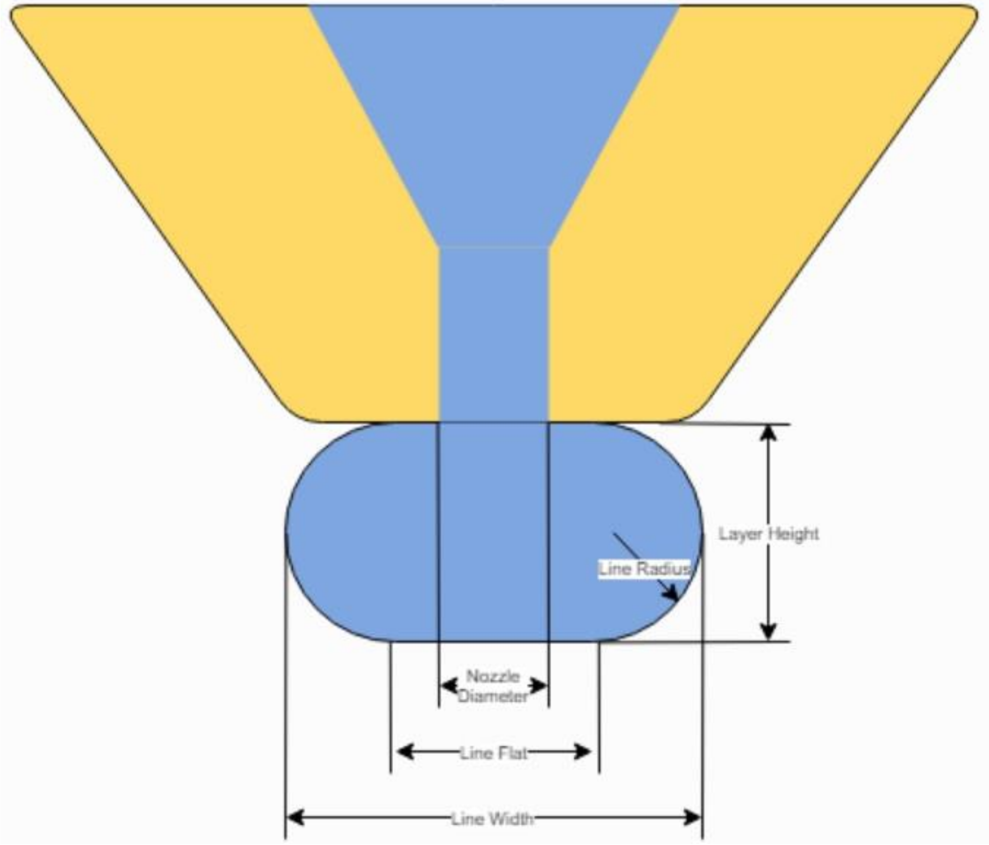
Layer Height

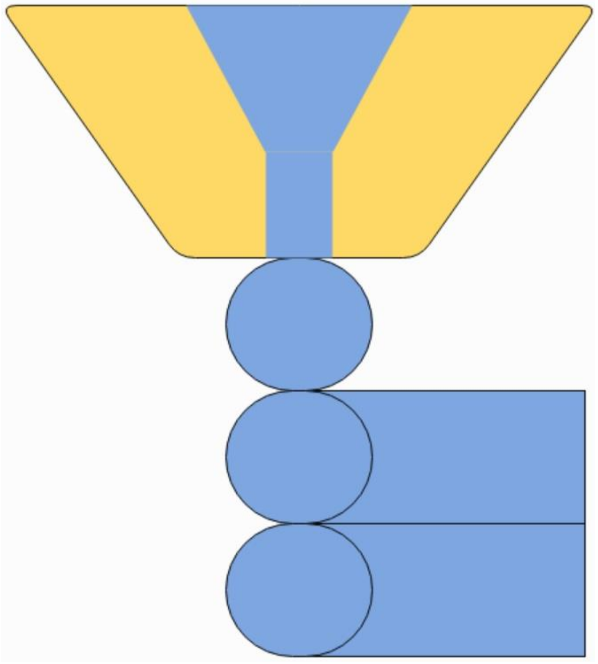
- The nozzle-to-build surface gap is determined by adjusting the build plate when the Z axis (up /down direction) is homed.

- The gap is a function of the nozzle bore and filament feed speed.

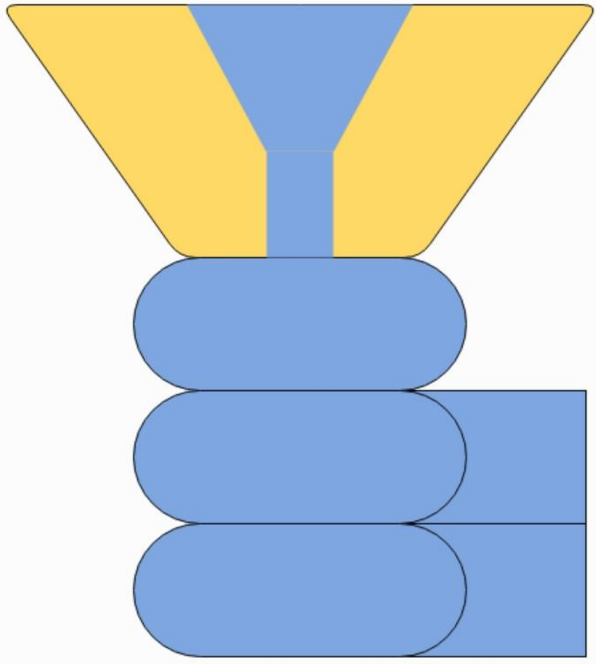
- The ideal filament bead is not round, but oval due to "squishing" from the end of the nozzle. The flat area provides more surface for the next layer to adhere to, resulting in a stronger print.

Nozzle Height to Bore Ratio





Layer height is equal to nozzle width resulting in under-extrusion.



Layer height is greater than nozzle width resulting in over-extrusion.

Diagnosing Common Print Failures

- Symptom: Air Printing (the printer gantry and build plate move, but no plastic exits the nozzle):

1. Filament binding on the spool.
2. Debris accumulation on the extruder hob gear.
3. Incorrect adjustment of extruder hob gear tension on the filament.
4. Nozzle blockage.
5. Heat Creep.
6. Filament embrittlement caused by moisture absorbsion (old filament).

Heat Creep Explanation

- Normally, the extruder gear pushes a solid strand of filament downwards into a small pool of melted plastic within the heater block, forcing it out of the nozzle.
- Heat creep is caused by the migration of heat from the heater block upwards into the filament path through the metal throat tube which has a heat sink attached. This softens the filament much higher than necessary, causing the pool of liquid filament to grow and move into the throat tube.
- The larger section of soft filament cannot be pushed out of the nozzle and eventually cools at the top of the throat tube, blocking the filament path.
- Once the filament stops flowing, the extruder hob gear continues to try to push the filament down and eventually abrades the outside of the filament enough that the gear spins freely. This results in a clicking sound.

Heat Creep Recovery Overview

- To clear a blocked filament path, the filament is cut in sections to allow the (clogged) hot end to be lowered and repaired.
- When the hot end is free from the extruder, the heater rod and thermocouple are removed to allow securing the hot end in a vise to remove the nozzle (held tight by the cooled filament in the heater block) using an 8 mm wrench.
- The heat break in the throat is very thin and will shear with little force applied. It is important to remember the throat is normally held to the block by the friction of the nozzle against its chamfered end. Once the nozzle is removed, the throat is held in place by melted filament binding the throat threads.
- Heat the block to 215C on a hot plate; (or heat gun.) Holding the heater block with a heat-resistant glove, unscrew the throat from the block manually using pliers.   If excessive force is required, ensure the block is hot or risk destroying the throat tube and having to remove the broken section from the heater block.

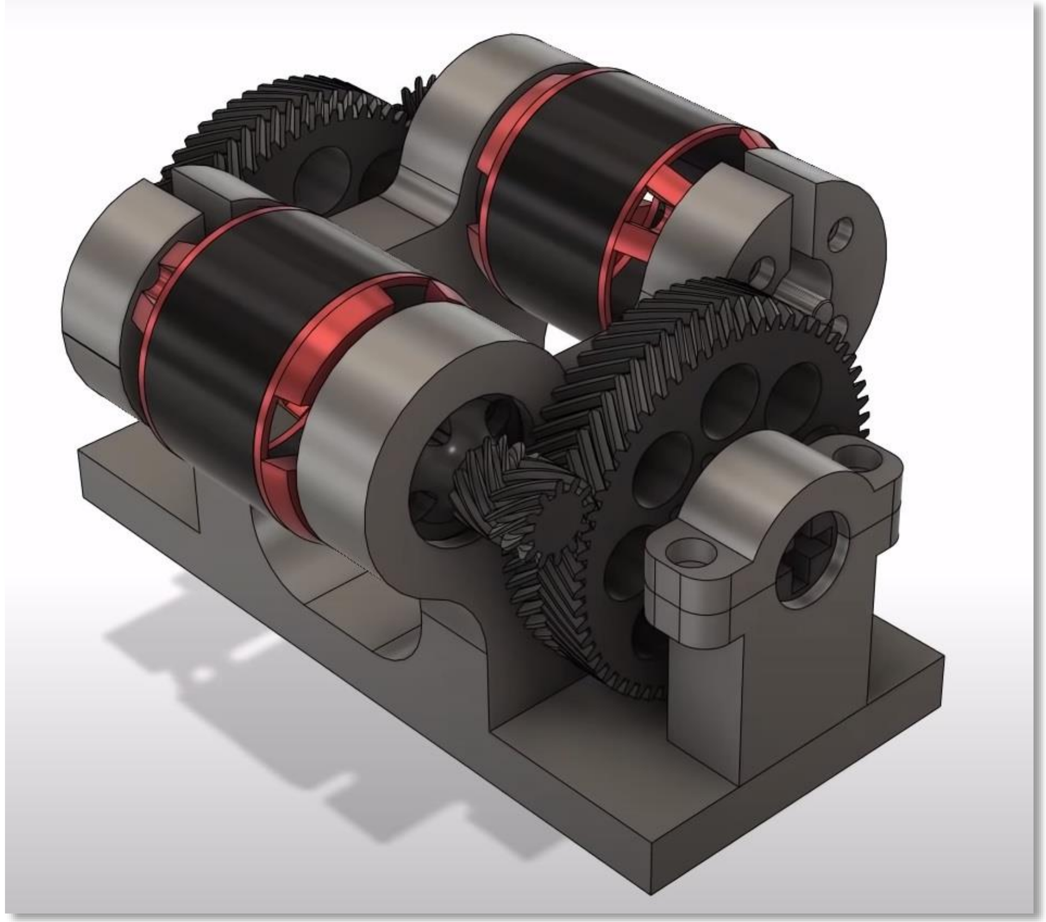
Heat Creep Recovery

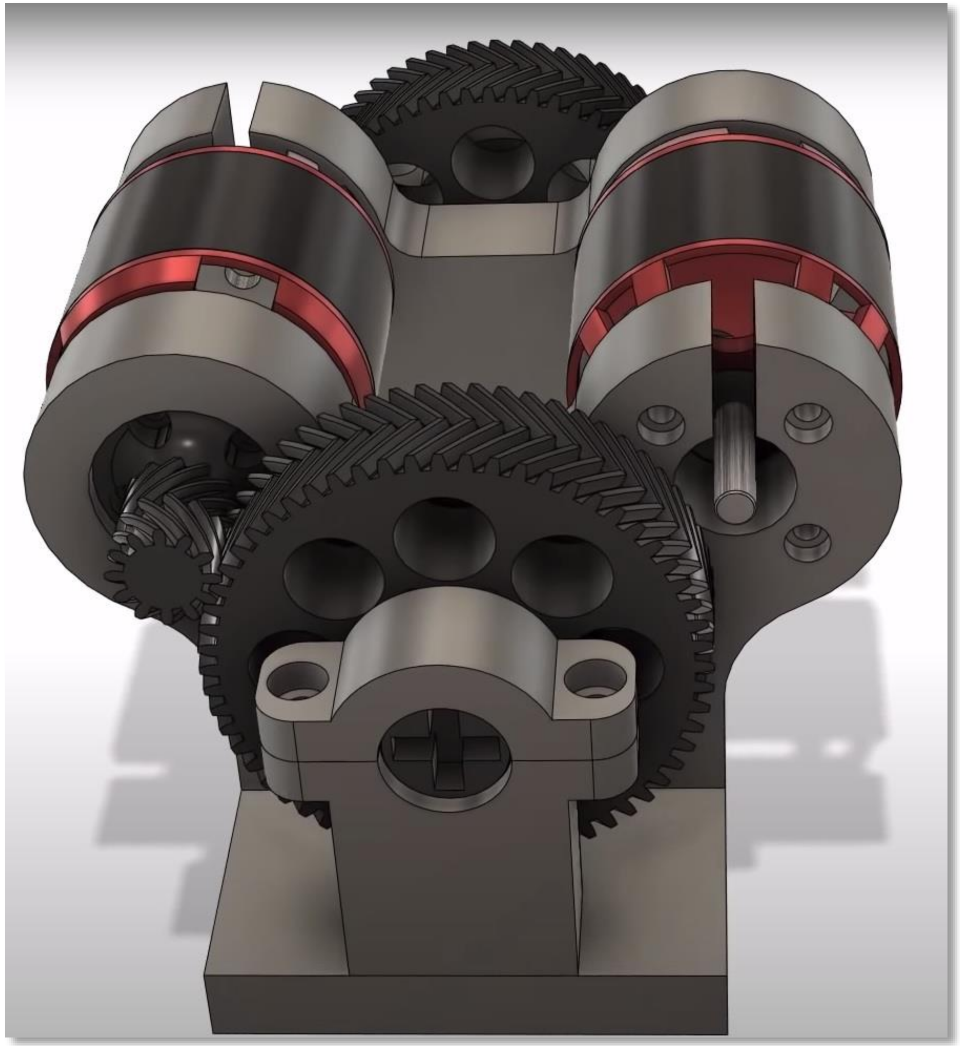
- Open the extruder tension cover and verify the hob gear has abraded the filament.
- A normal unload command will not work since the hob gear cannot grip the filament.
- Initiate the unload command to heat the hot end and attempt to manually pull the filament out of the top of the extruder. Do not use excessive force to avoid bending the gantry rods.
- If this fails, (and it usually does, but it’s the easiest way to get the filament free so it’s worth a shot) cancel the Unload command and lower the build plate 100mm to allow easier access to the hot end.

Heat Creep Recovery

- Ensure the heater block is cool to the touch.
- Cut the filament 6” above the top of the extruder.
- Loosen the extruder tension adjustment screw.
- Remove the two 3mm fasteners that secure the filament runout sensor housing.
- Lift the filament runout sensor housing and cut the filament just below the housing.
- Pull the filament out of the housing from the top and reinstall the housing.
- Remove the 30mm fan and air deflector from the affected heater block.
- Loosen the single 3 mm collet clamp screw that secures the hot end (via the throat tube) to the bottom of the extruder.

Printing Gears



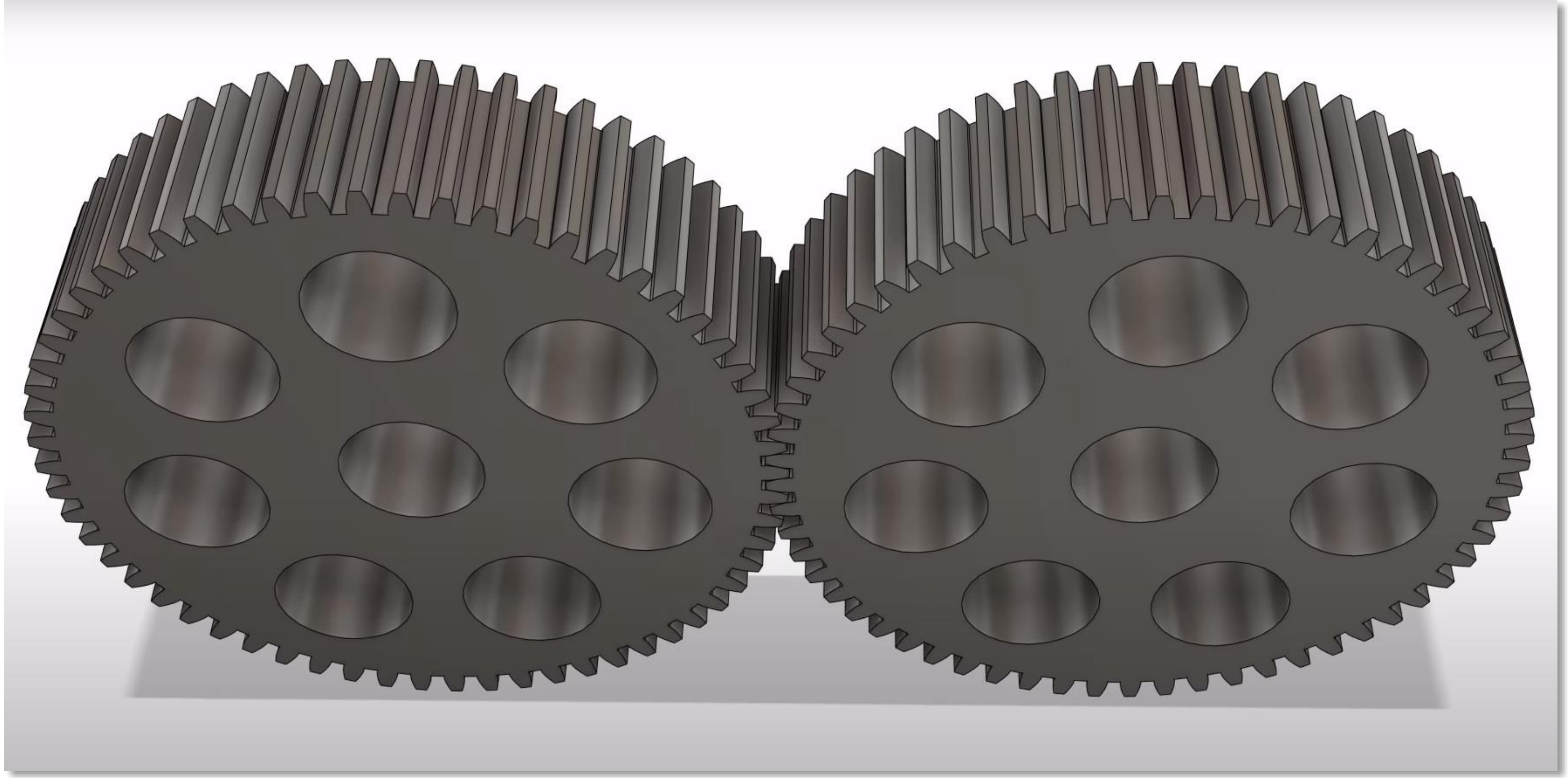


3D-Printed Gear Considerations

(L-R: Spur, Helical, Herringbone Gears)

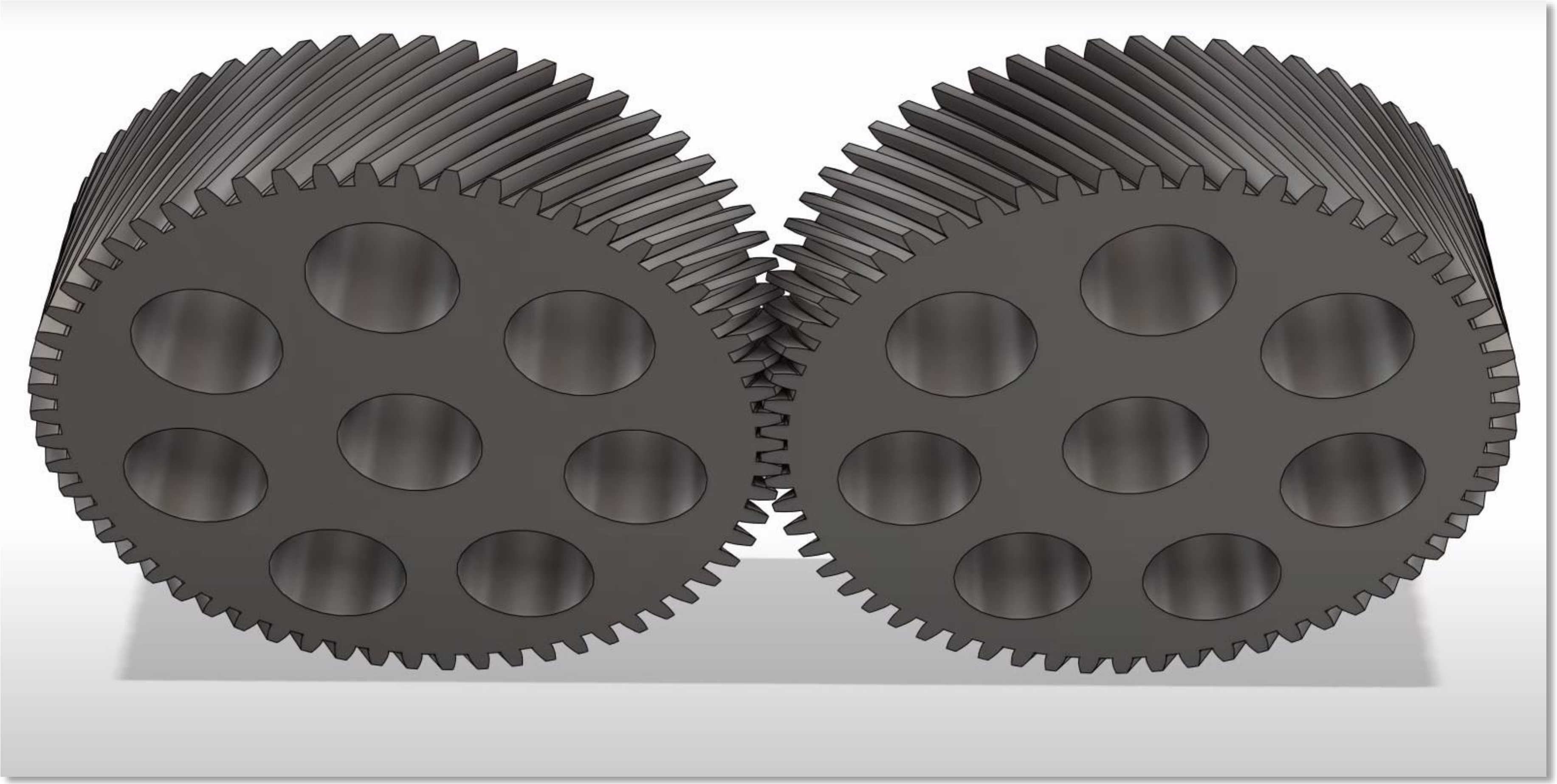
PLA is sufficient for low-power gears, but Polycarbonate printed gears are much stronger.

Printing Gears – Spur Gear



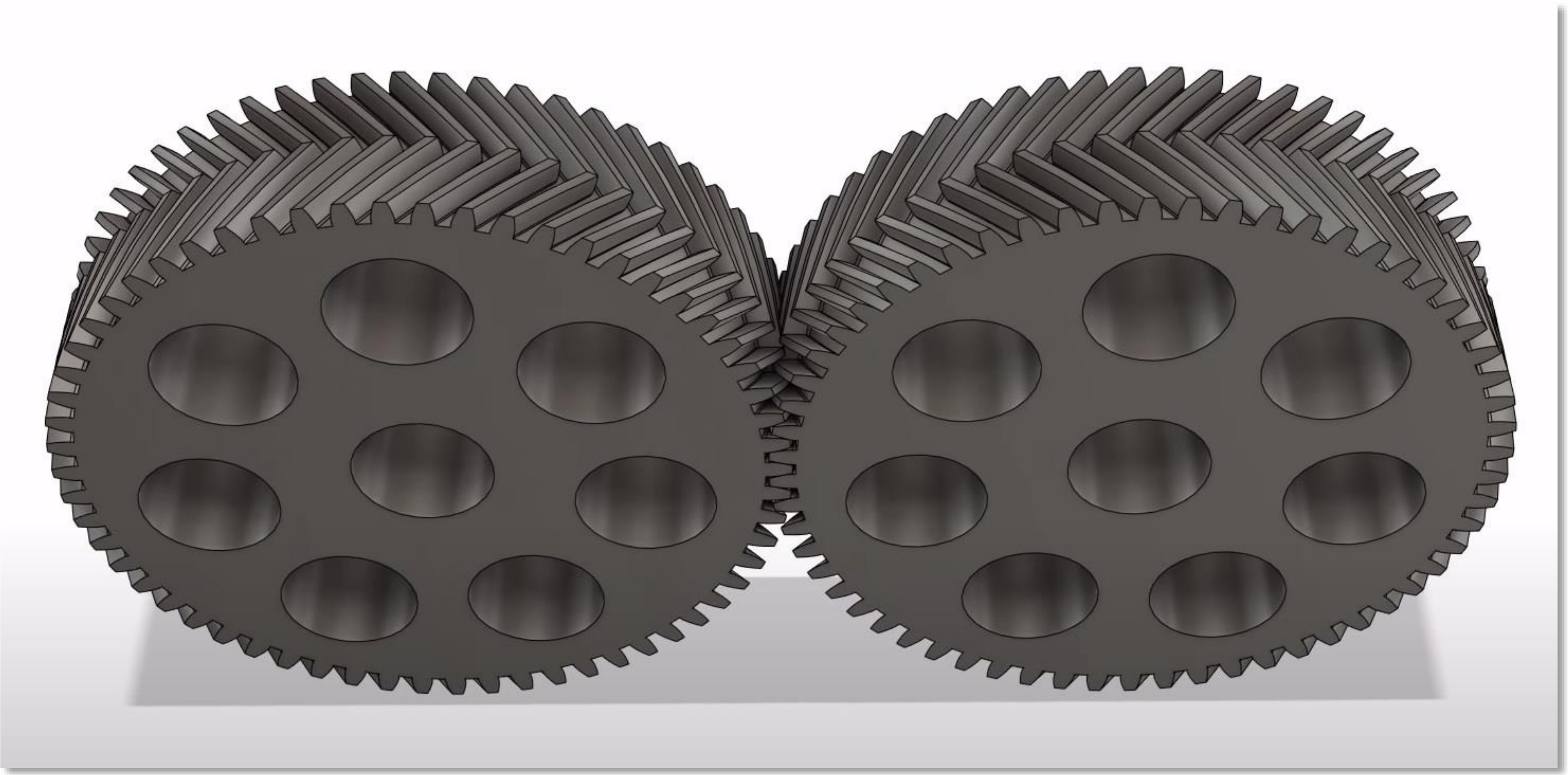
(Efficient, Optimized for Low-Speed, Low-Power Applications)

Printing Gears – Helical Gears



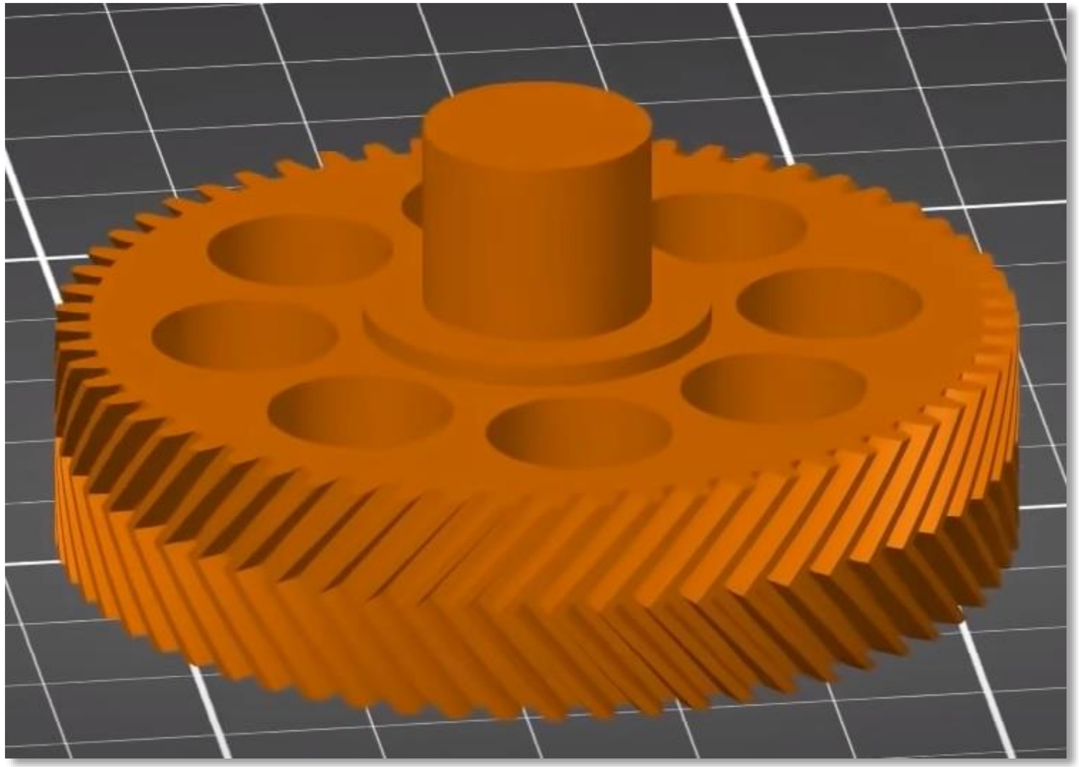
Better Load Distribution on Teeth Across Printed Layers

Printing Gears – Herringbone Gears



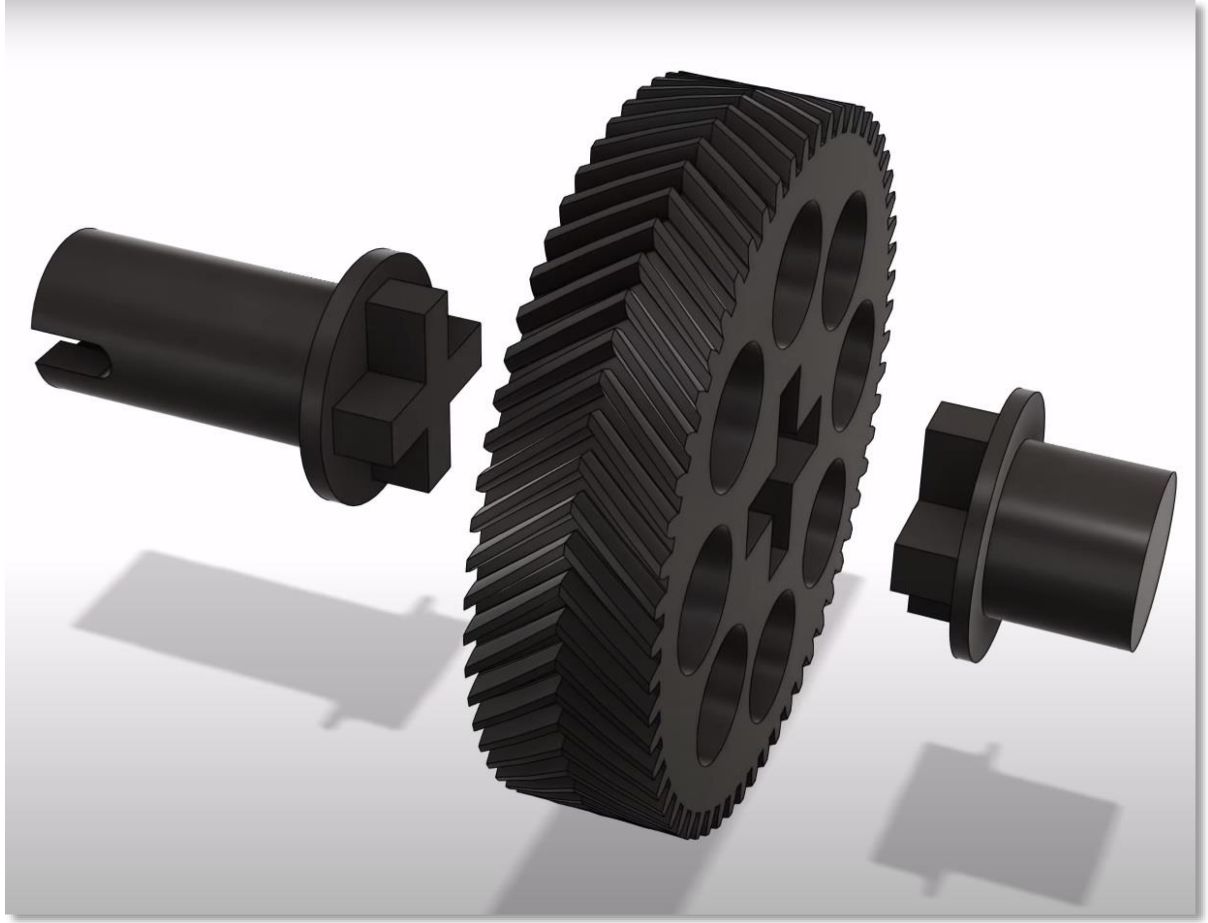
Herringbone Gears Offer Good Load Distribution and V-shape Reduces Axial Play

Printing Gears – Optimal Alignment



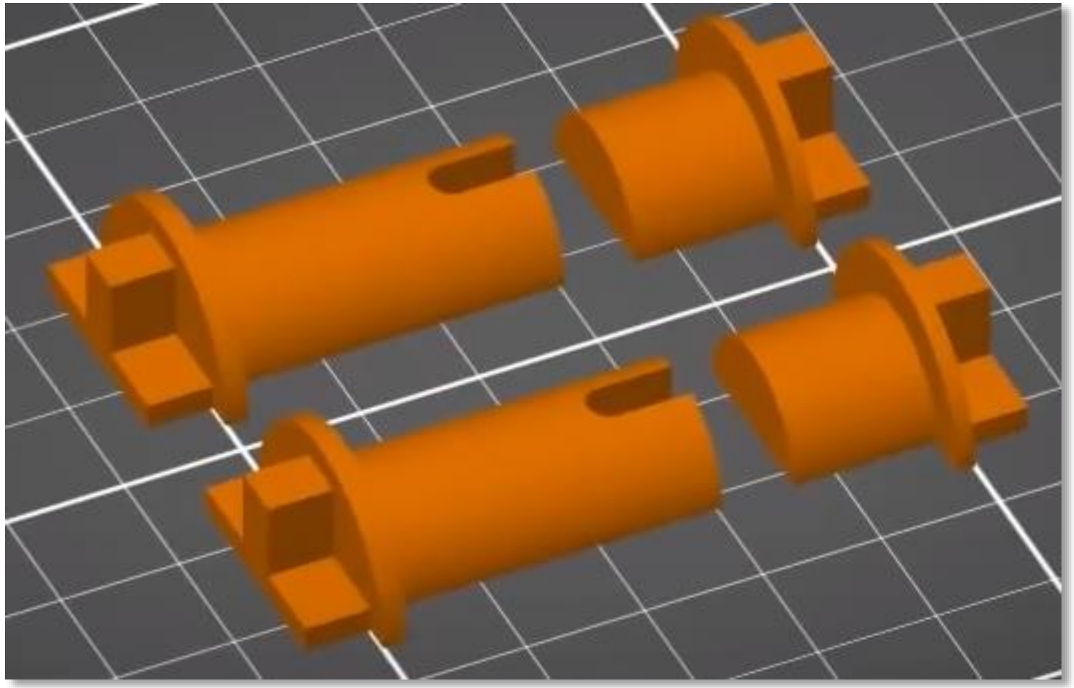
Orienting the teeth perpendicular to the load increases strength, but using an integrated shaft increases the likelihood of a single layer yielding under load.

Printing Gears – Sharing the Load



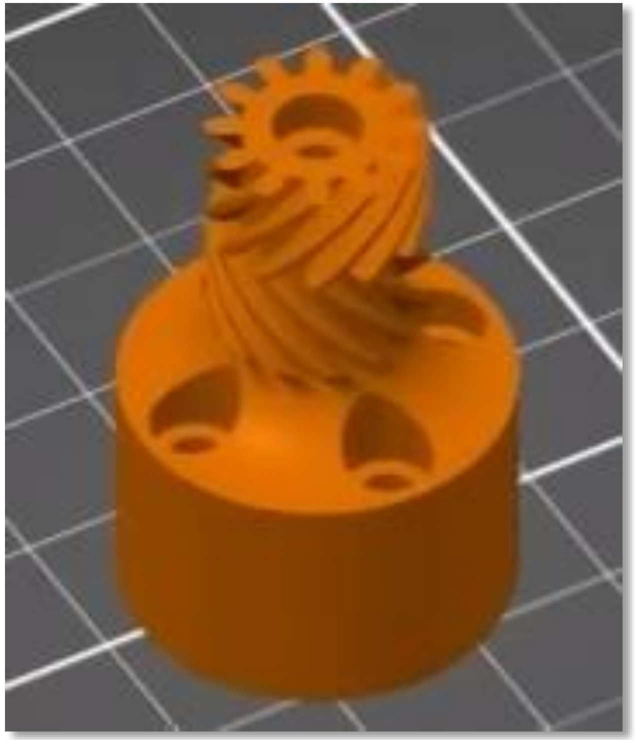
To optimize printed strength, the axels are printed separately in a different orientation.

Printing Gears – Axel Printing



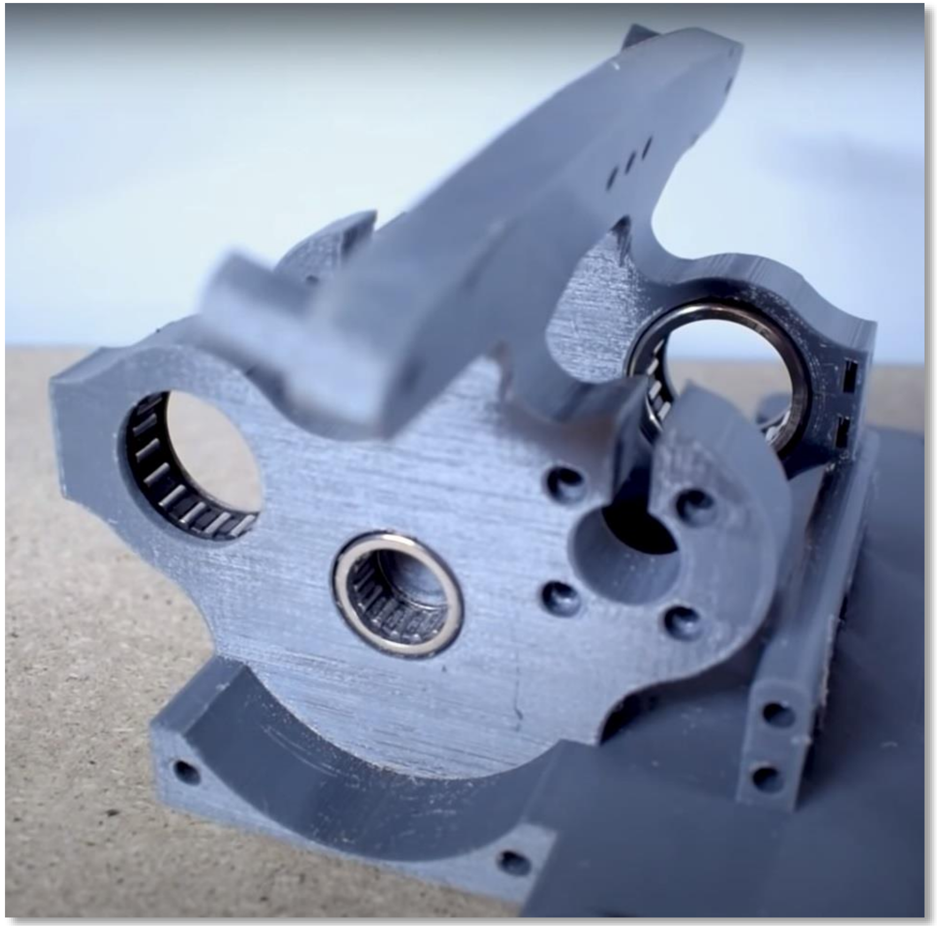
Splitting the axels opposite the loading force and subsequent bonding provides a stronger part.

Printing Gears – Pinion Strengthening



To strengthen the pinion gear, a fastener can be installed in the shaft.

Printing Gears - Bearings



Roller Bearings are Preferred for High Reliability