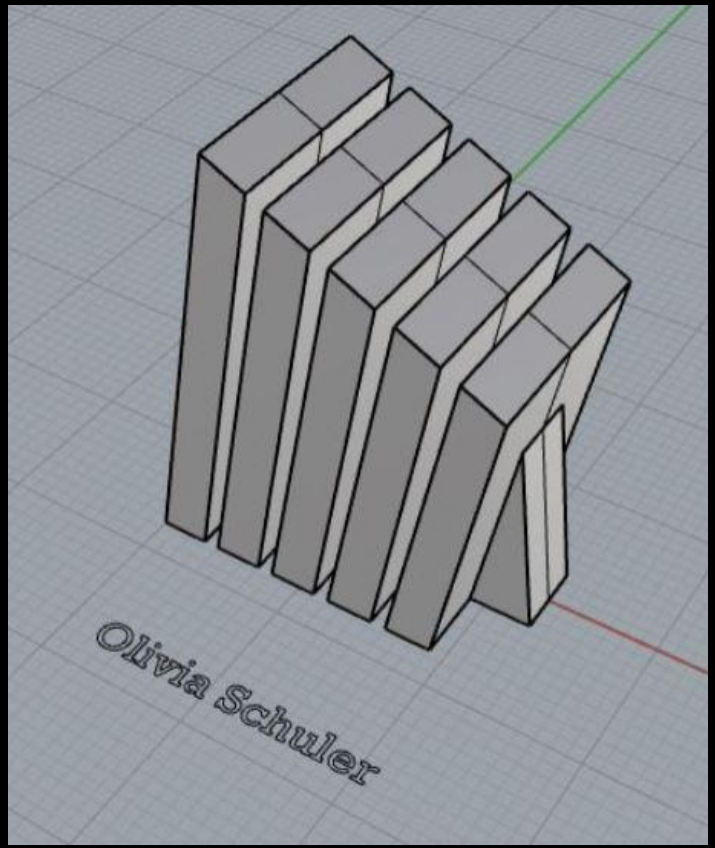
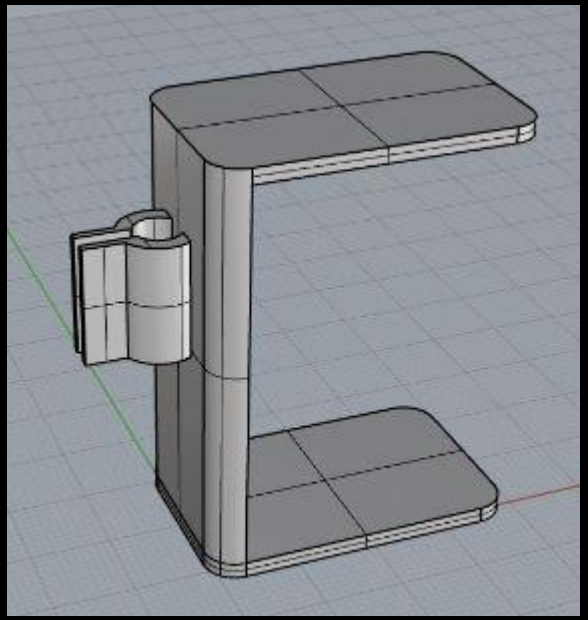
Product Design

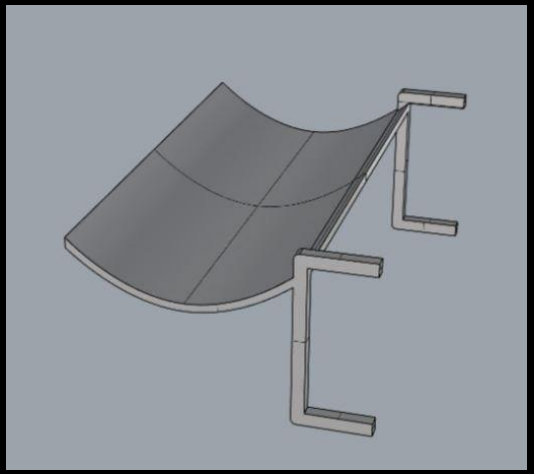
OIDD 415/515
MEAM 415
IPD 515

- Intro 3D Printing
- Technologies
- Materials
- Designing for 3D Printing

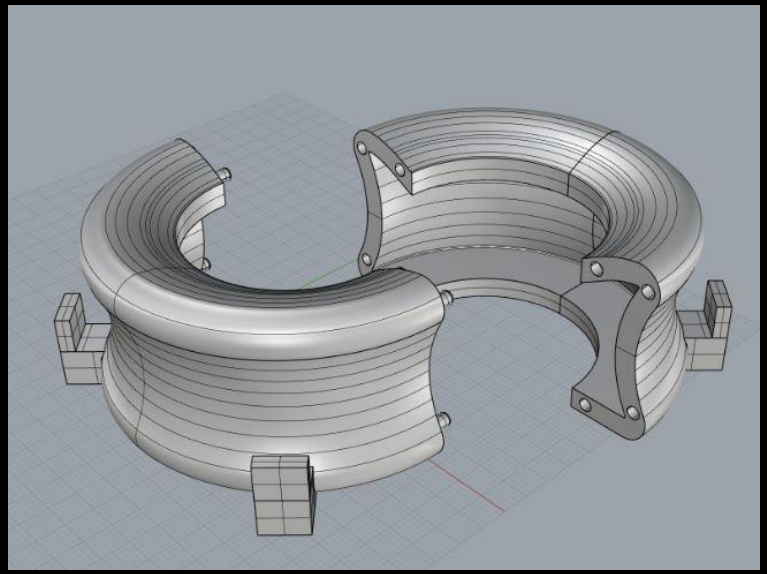
Great work!

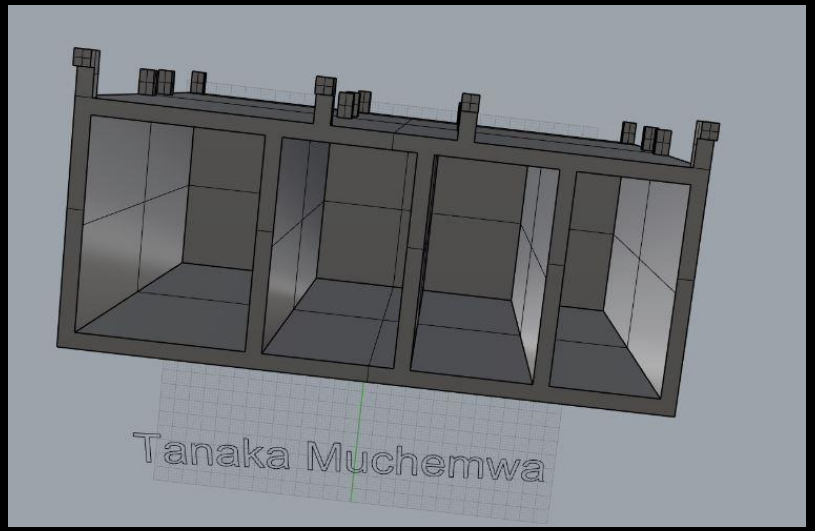


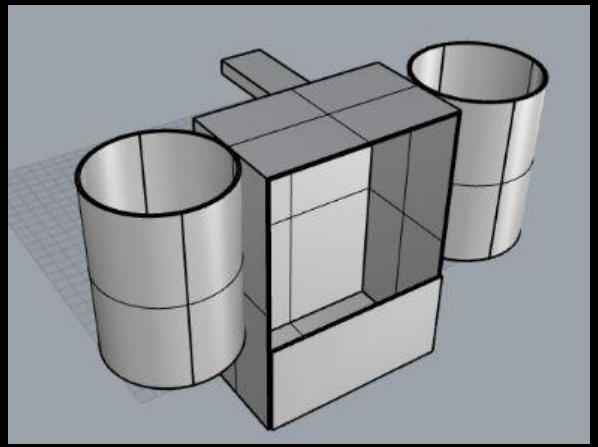




Great work!

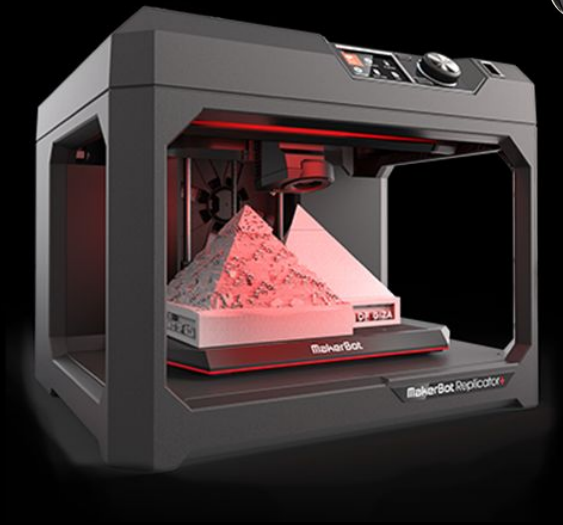






What is 3D Printing?

3D printing is one type of process of producing a physical object from a digital file. The digital file is sliced into 2D cross sections which are then made physically, one on top of another, by the 3D printer. Think of it as building your 3D model layer by layer.



- 3D printing = additive
- CNC = subtractive

3D Printing - Fused Deposition Modeling

- Cold plastic on a spool is fed into a hot extruder - like a robotic hot glue gun
- Extruder draws a layer of your object, moves up a small amount, and draws the next layer

Pros:

- Cheap
- Fast printing
- Accessible

Cons:

- Low resolution/quality - build lines
- Waste of support material
- Toxicity

3D Printing - Fused Deposition Modeling

- Cold plastic on a spool is fed into a hot extruder - like a robotic hot glue gun
- Extruder draws a layer of your object, moves up a small amount, and draws the next layer

Pros:

- Cheap
- Fast printing
- Accessible

Cons:

- Low resolution/quality - build lines
- Waste of support material
- Toxicity

3D Printing - Fused Deposition Modeling

3D Printing - Fused Deposition Modeling

3D Printing - Fused Deposition Modeling

Materials - Plastics

- ABS - legos
- PLA - most common
- PET(g)
- TPU (flexible)
- PLA composites
- Carbon fiber
- Nylon

3D Printing - Selective Laser Sintering

Layers of powder are applied to a base. After each layer a laser sinters (fuses) the cross section of your object. The bed drops slightly, another layer of powder is applied, and the laser sinters the next layer to the one below it.

Pros

- Support material is reusable
- Affordable, even for large parts
- Can achieve very thin walls
- Moveable parts are possible
- Beautiful and flexible material
- Can be dyed

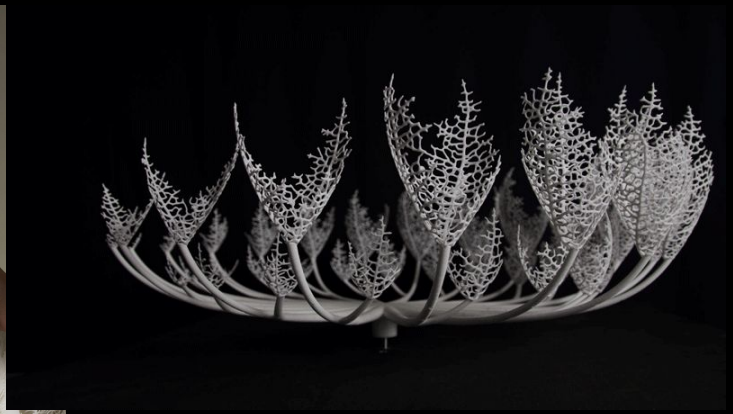
Cons

- Industrial printing only - Shapeways

3D Printing - Selective Laser Sintering

Ashley Marcovitz

Iris Van Herpen



Nervous System

3D Printing - Selective Laser Sintering

Nervous System

3D Printing - Stereolithography

SLA - photosensitive resin plus light (laser) - laser cures a cross section of your piece, layer by layer, fusing to the one below

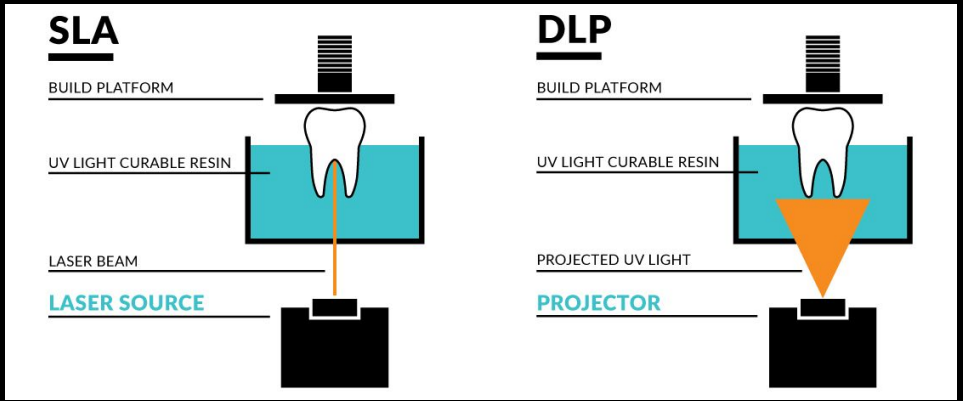
Pros:

- High level of detail and accuracy
- Can achieve truly clear prints

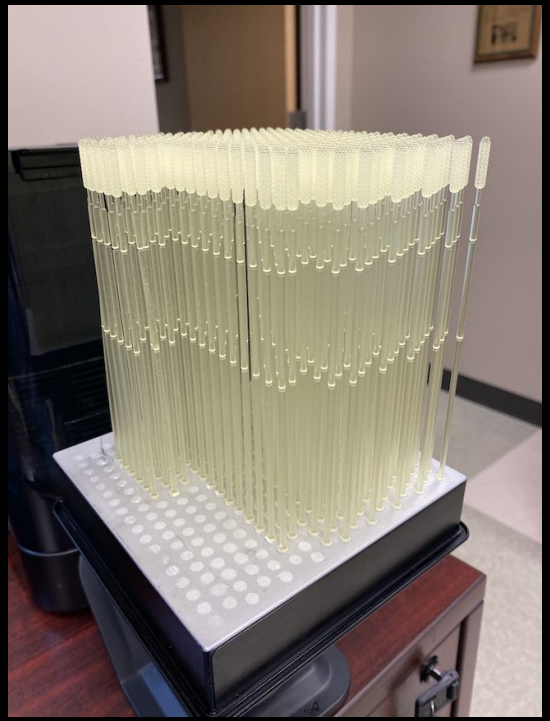
Cons:

- Expensive
- Supports can leave marks (FormLabs)

Digital Light Processing (DLP) - uses a flash of light to cure the resin; faster and gaining popularity



3D Printing - Stereolithography



Adidas 4D Shoes: One of the first mass-produced products utilizing 3D printing technology. The sole is produced using the DLP process from the company Carbon 3D.

Product Design

OIDD 415/515
MEAM 415
IPD 515

- Intro 3D Printing
- Technologies
- Materials
- **Designing for 3D Printing**

Designing for 3D printing

- **No matter what software you design in, there are best practices for designing for 3D printing**
- The following slides discuss things to keep in mind specifically for printing using Fused Deposition Modeling (FDM) printer, like the Stratasys F120 (used in the Studios) or a common hobby printer like models by Prusa, Ender or MakerBot

Slicing

How does my file get to the 3D printer?

- Your .stl file is put through a “slicing” software, in which the operator can orient your object for optimal printing and choose print settings like infill, resolution, speed, temperature, and placement of support material
- The slicing software will “slice” your object into vertical contours called layers - each layer is generally 0.1 - 0.3mm thick
    - Like a sliced loaf of bread stacked vertically
    - These are the ridges you see on your print
- These “instructions” are sent to the printer. They tell the printer what to print layer by layer. (gcode)
- Common slicing software: GrabCAD, Simplify3D, Cura

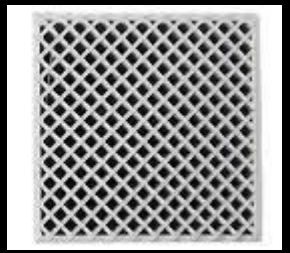
Infill

The structure inside your print - FDM only

- Infill is like a scaffolding built inside of your 3D print to support it during printing and give it strength as a final object
- Infill can range from your object being totally empty (0% infill) to totally solid (100%) infill but generally you'll print with an infill of 10% - 35%
    - Low infill (around 5%) is good for objects that don't need a lot of final strength - such as items that are decorative
    - High infill (35% or higher) is for objects that need to be strong or withstand force - such as items that will bear weight
    - Studios will print at a low infill unless otherwise requested

5% infill

10% infill

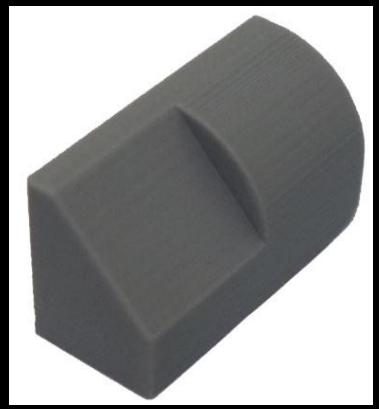


35% infill

Resolution

The height of your layers

- Resolution is the layer quality, or level of detail at which your part will be printed.
- The resolution of a part is the **exact height** of each layer of plastic extruded. The thinner your layer height, the smoother and more geometrically accurate the part will be to the CAD model.
- Resolution is chosen during “slicing”

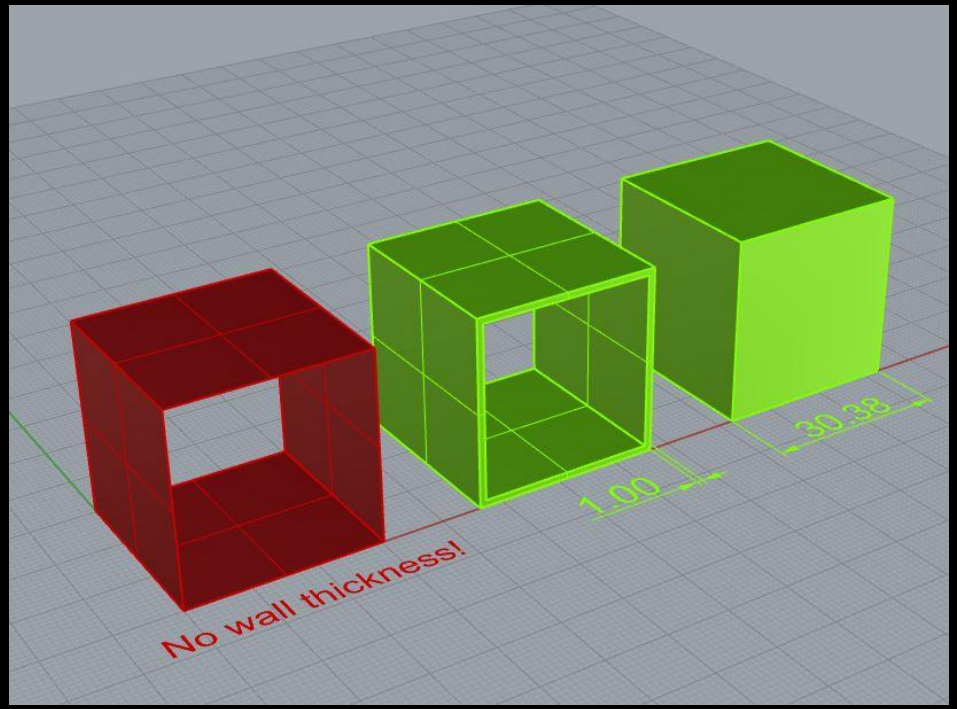


Left to right:
0.3mm layer height
0.2mm layer height
0.1mm layer height

Designing for 3D printing

2. Account for the minimal wall thickness.

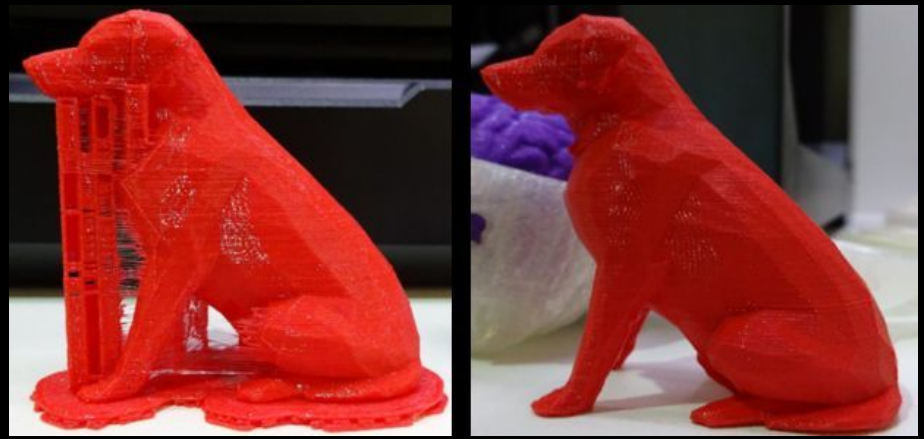
- In 3D Printing, **wall thickness** refers to the distance between one surface of your model and the opposite surface (think OffsetSrf)
- FDM printers can only print so thin - otherwise your part will not be structural or won't print at all.
- The minimum wall thickness for FDM printers is usually 1mm.
- If your object is a large solid, like a cube, the thickness of the whole cube is the wall.



Designing for 3D printing

1. Minimize unnecessary overhangs

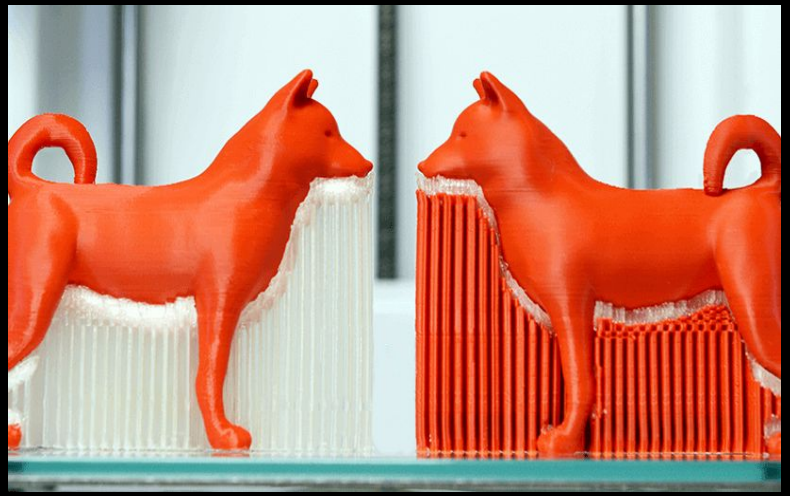
- **FDM Printers can't print "into thin air".** If there are overhangs, the slicing software will generate **support material AUTOMATICALLY.**
- **Supports** are structures that hold up your part's overhangs during printing so it doesn't sag or cause the print to fail. The support material is then broken off (or dissolved) when your part is completed.
- With traditional FDM printing, supports will leave an undesirable rough surface on your print. Dissolvable support does not leave marks but it can take some time to remove.



A 3D Printed dog before and after support material is removed. His snout is a large overhang.

Designing for 3D printing

1. **minimize unnecessary overhangs** (either by design or orientation)



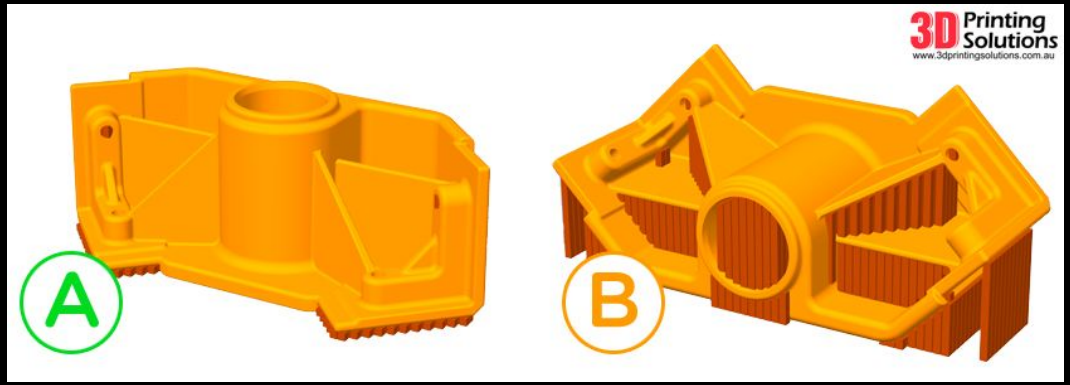
Left: Dog on the left has dissolvable support, dog on the right has more typical support made of build material

Right: The Y is at a high enough angle not to need supports.

Print Orientation

1. minimize unnecessary overhangs (either by design or orientation)

- **In addition to design choices, you can also minimize support material use by what orientation your part is printed.**
- You don't necessarily have to model your object in the orientation it will be printed in Rhino (you can orient it later in the 3D printing slicing software if needed).
- However, the orientation your part will be printed in is something you should think about while modeling.
- Including a flat surface at least the size of your thumbnail indicates to the printer operator to place that surface on the print bed.



Print A uses a lot less supports than Print B, just by changing the orientation.

Printing “In the Z Axis”

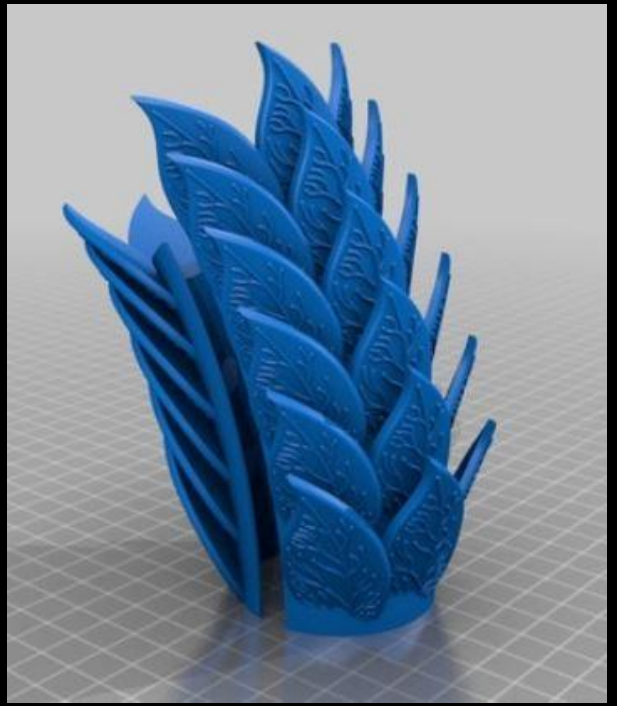
Better quality can sometimes be achieved by printing objects vertically

- The best print quality is achieved by printing upright on the build platform.
- You can design your object with a flat surface that can be placed on the plate
- Less applicable when printing in Studios, but if you print on most other FDM printers this is a great thing to do!

This object prints better straight up and down. When printed flat it loses accuracy and has more visible build lines.

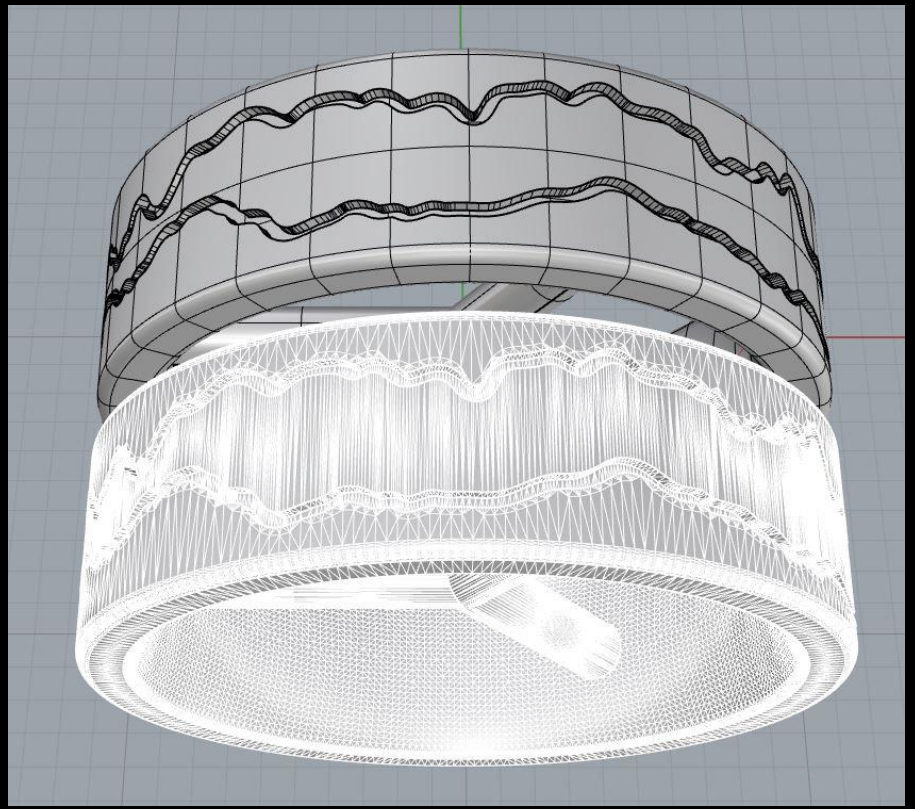
Designing for 3D Printing

1. **minimize unnecessary overhangs** (either by design or orientation)



File Types for 3D Printing

- Slicing softwares can’t read Rhino files (.3dm)
- You must export a .stl file
  - .stl converts your object to a “mesh” made of triangles
  - A universal file type that can be read by any CAD software
  - Not editable*
- Follow instructions in One Part Challenge for exporting to .stl



*Not in a way that is easy or generally useful

One Part Work Time

- Choose Small Objects MM template
- SHADE your Perspective viewport!!!!!!!!!
- Remember to work at the correct scale!
- Measure with calipers!
- Minimum wall thickness of 1mm!
- Your object must be no larger than 75mm cubed, and no smaller than 25mm cubed (1-3”)
- Your object must have a function
- Incrementally save frequently using command **SAVEAS**
- Check for naked and non-manifold edges before exporting, using command **ShowEdges**
- All solids should be combined using **BooleanUnion** (don’t just Group everything)
- **Instructions on exporting to .stl are in Canvas!**
- **Files are due week of 3/28!**

Homework

- Lecture: Global Sourcing
- Visualization III
- CageEdit and Rebuild video
- Advanced Rhino Tools video

- Start purchasing any supplies to make your Alpha Prototype! We will work on these in class week of 3/28!!