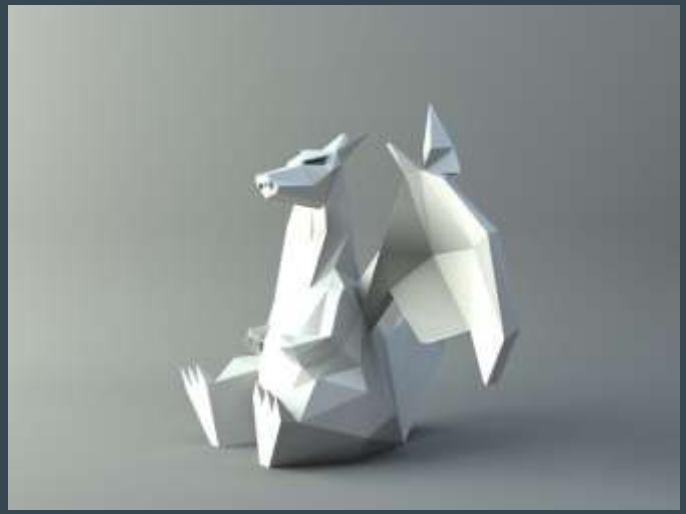
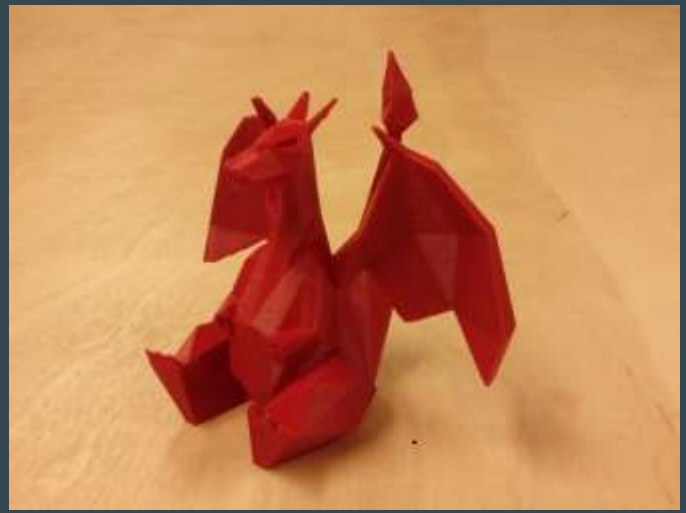
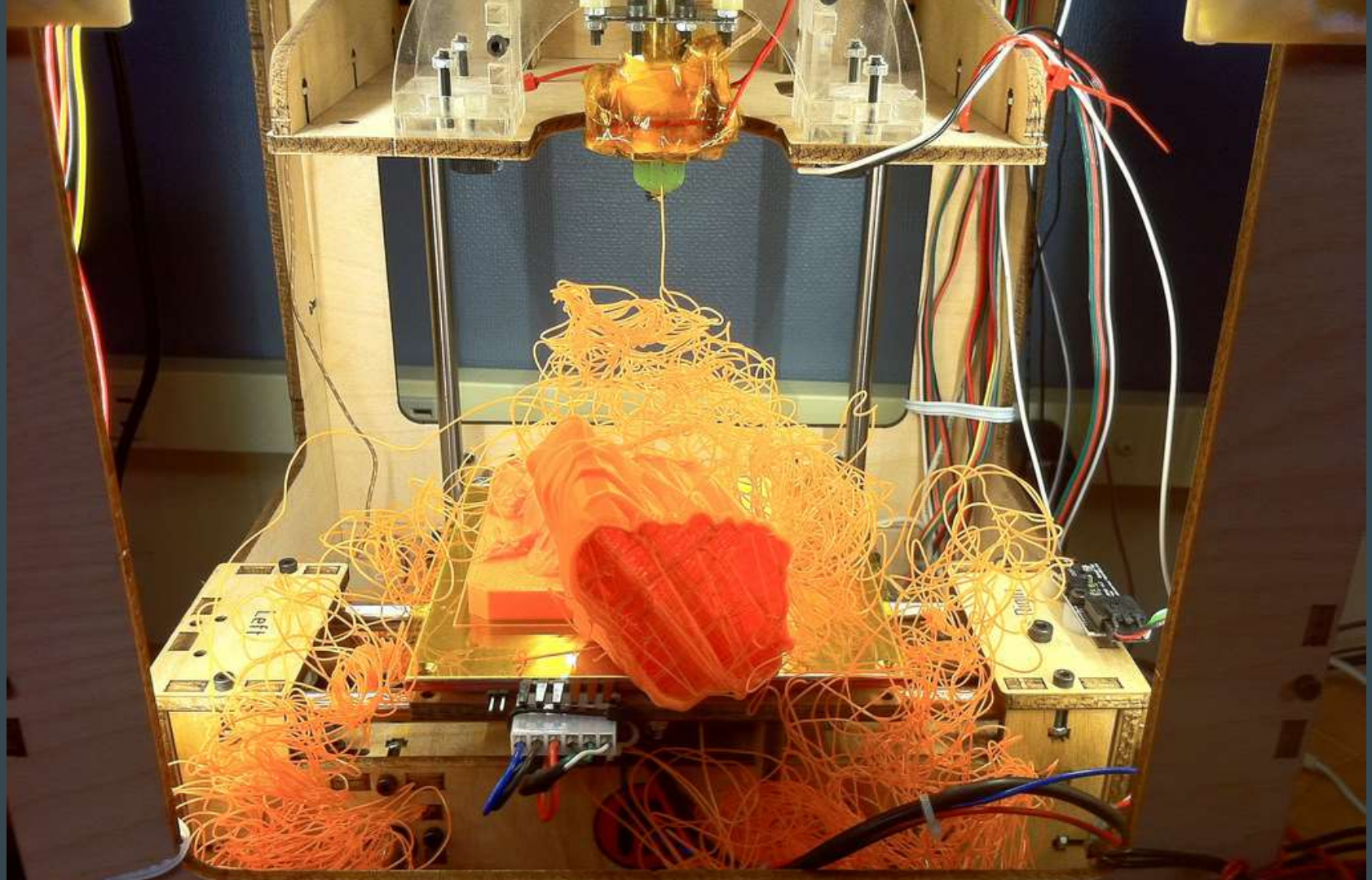
3D Printing Orientation and Guide

Student Innovation Idea Labs
ilab@cpp.edu

What is 3D Printing?



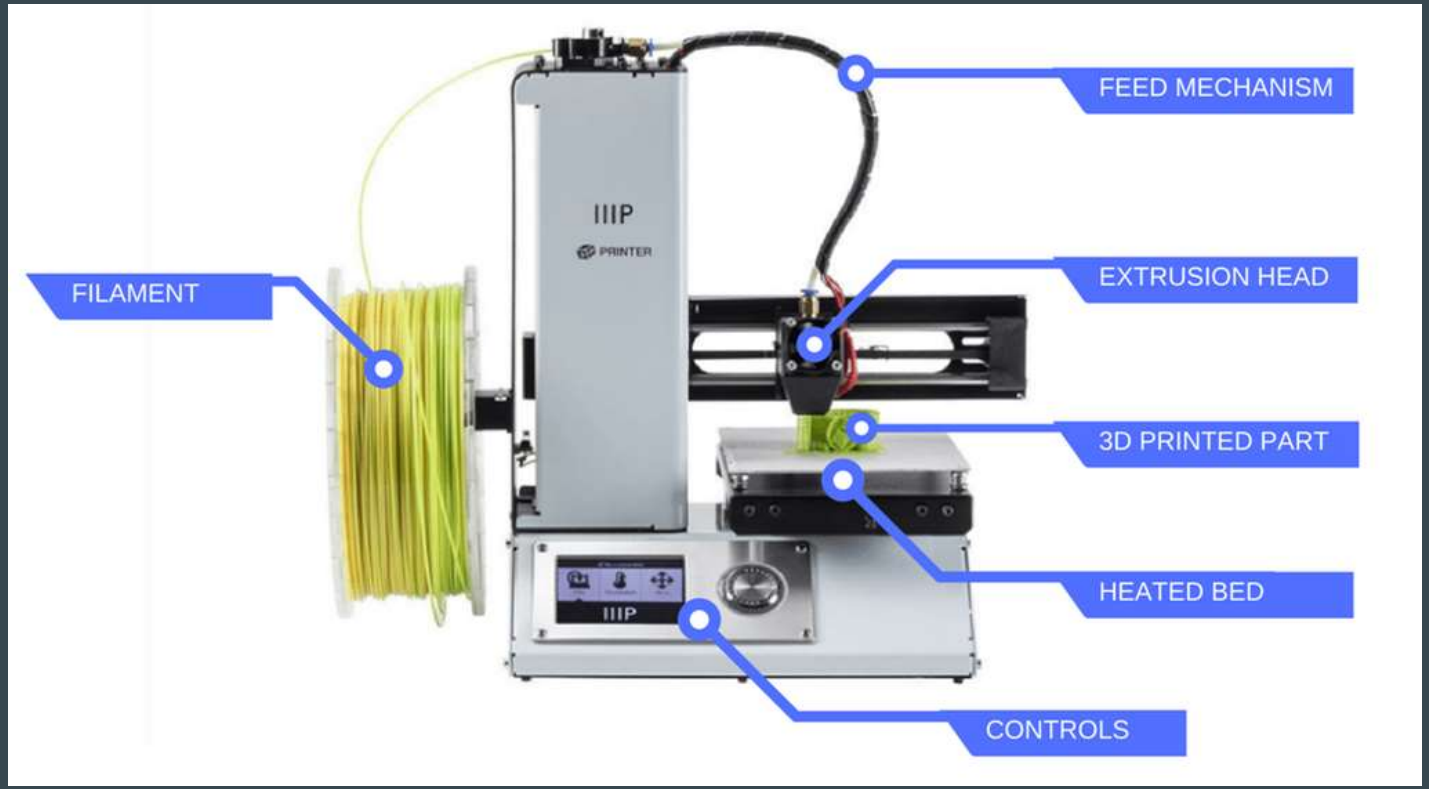



Left

Topics to Help You Succeed in 3D Printing

5. How FDM/FFF Printing Works
6. Types of Filament and Uses
7. Printable Type Files
8. Cura Slicing Software
9. Positioning of a Print
11. Support material
12. Layer Height
13. Bed Plate Adhesion
14. Infill
15. Wall Thickness
16. Tolerances
17. Hardware Issues
18. Cura

How FDM/FFF Printing Works



Types of Filament and Uses

	PLA (Polylactic Acid)	**ABS** (Acrylonitrile butadiene styrene)	**PETG** (Polyethylene Terephthalate “glycol-modified”)	**TPU** (Thermoplastic polyurethane)
Difficulty to use	Low	Medium	Low	Medium
Printers	Monoprice, Ultimaker, Craftbot	Ultimaker, Craftbot	Ultimaker, Craftbot	Craftbot
Pros	Easy to Use	Acetone Soluble, Heat Resistance (100 C)	Layers Weld Together Well	Flexible
Cons	Brittle, not Heat Resistant (50 C)	Warping/Shrinkage, Best in Enclosure	Lower Heat Resistance (80C)	Hard to print on Bowden Tubes
Temperature	180°C – 220°C	210°C – 250°C	220°C – 250°C	210°C – 230°C
Bed Temp	20°C – 60°C (Optional)	80°C – 110°C	50°C – 75°C	30°C – 60°C (Optional)

Printable Type Files

3D MODEL FILE

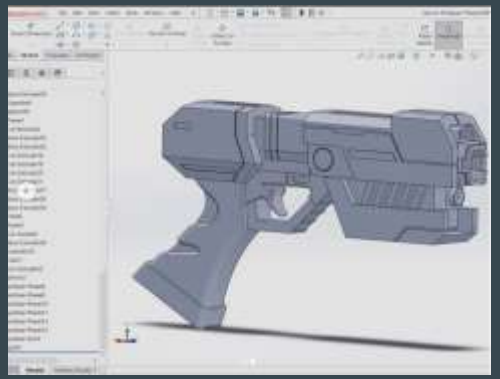
Ex: .SLDPRT
.DAE

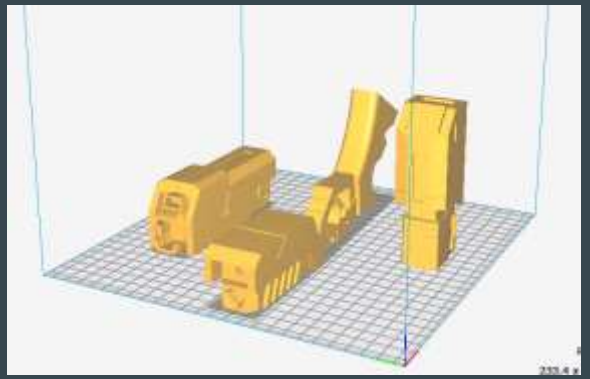
→

- .STL
- .OBJ
- .X3D
- .3MF

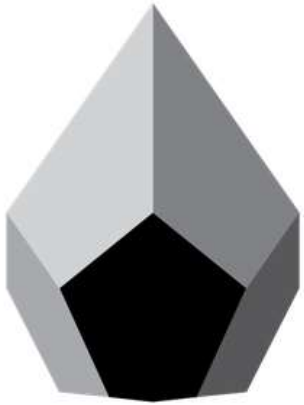
→

.GCODE





Cura Slicing Software



Design

The first thing you'll need is a 3D model. Ultimaker Cura works with STL, OBJ, X3D or 3MF file formats out-of-the-box.

Prepare

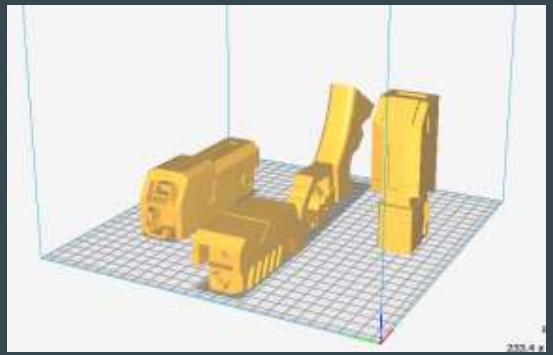
Within moments, Ultimaker Cura slices your model ready for print. You can preview it, scale it and adjust settings as you'd like.

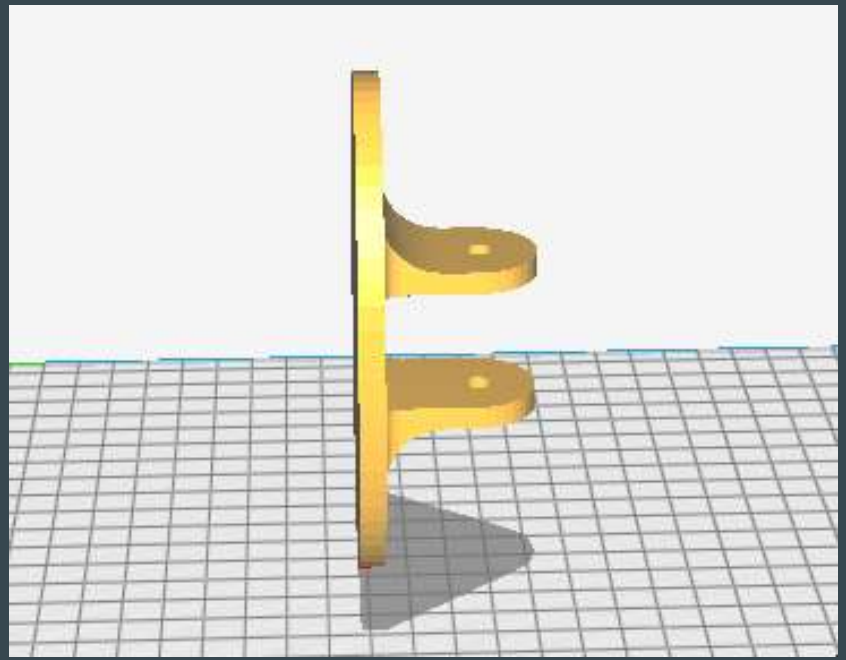
Print

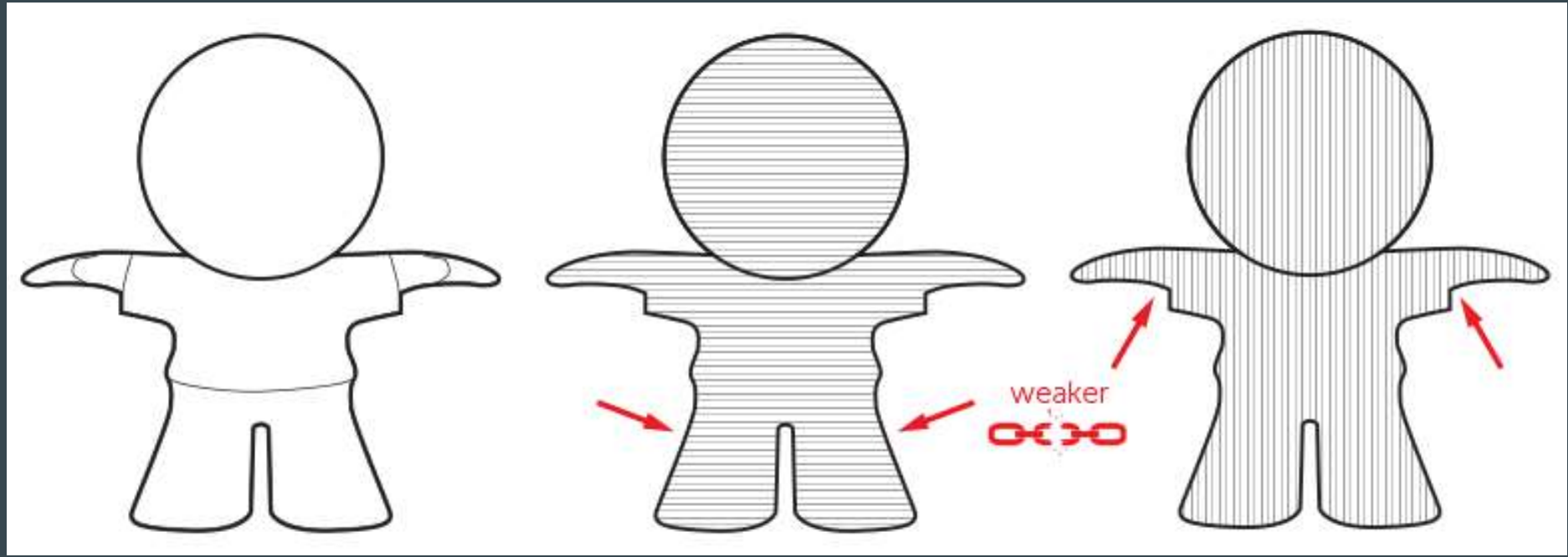
Send print jobs remotely using a network-enabled Ultimaker 3D printer, or save files to a USB stick or SD card.

Positioning of a Print

- Most Surface Area
- No Islands
- No Overhangs
- May Need to Split Part






weaker

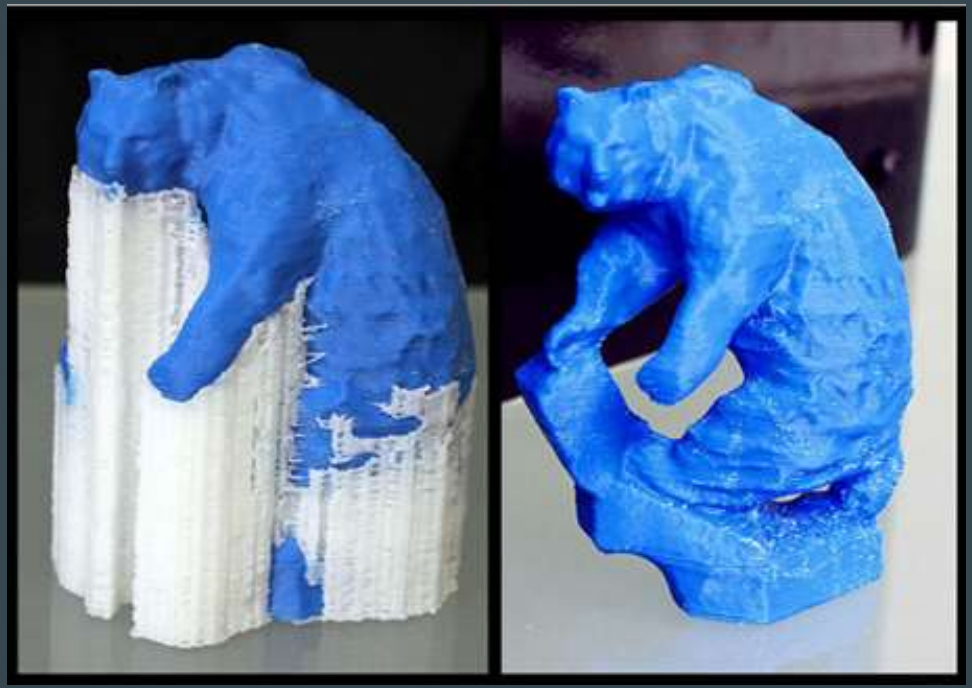
Support Material

Usage:

- No Ideal Positioning
- Floating parts
- Overhangs/Islands
- Organic parts

Cons:

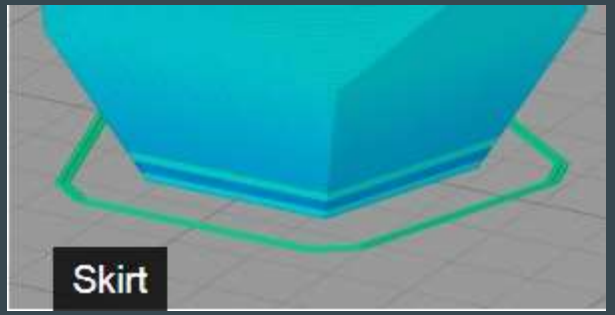
- Hard to Remove
- May Break Part
- Cosmetics



Layer Height/ Resolution

Note: I usually use .1 mm, .2 mm, or .3 mm

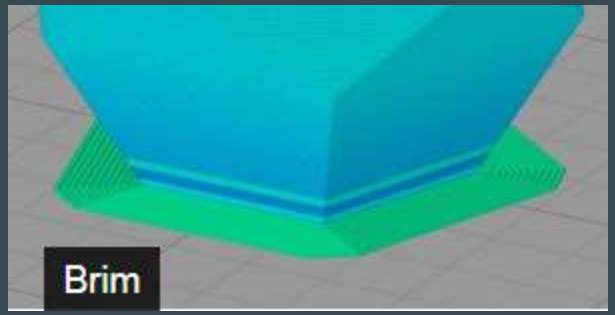
Bed Plate Adhesion



Uses:

- helps prime your extruder
- establish a smooth flow of filament

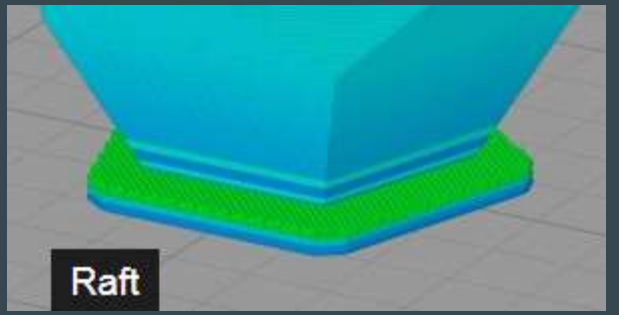
Filament use: Light
Print Speed: Fastest
Separation: N/A



Uses:

- holds down the edges of your part
- can prevent warping
- helps with bed adhesion

Filament use: Medium
Print Speed: Fast
Separation: Easy



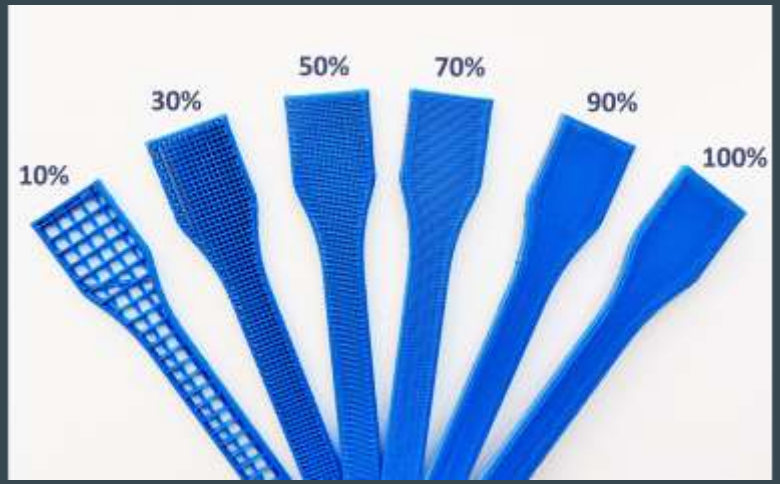
Uses: ABS Filament

- helps with warping and bed adhesion
- helps stabilize models with small footprints
- creates a strong foundation

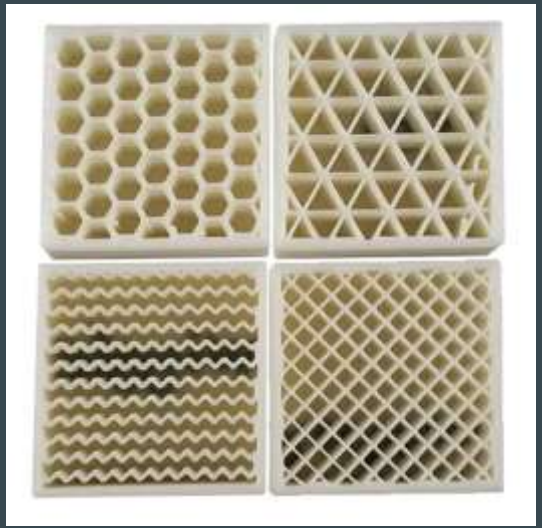
Filament use: High
Print Speed: Medium
Separation: Medium

Infill

- Save on plastic and materials
- Choose the Density of the Print
  - Higher percent takes more time
  - Higher percent = more strength



- Choose the Shape of the Infill
  - Grid- easiest infill patterns to print
  - Triangular - High strength in the direction of the shell.
  - Zig zag-Great to use for flexible materials.
  - Honeycomb-greater overall strength in all directions than a rectangular pattern



Wall Thickness

Most Printers: .4mm Nozzle => Multiples of .4 optimizes print time

.8mm

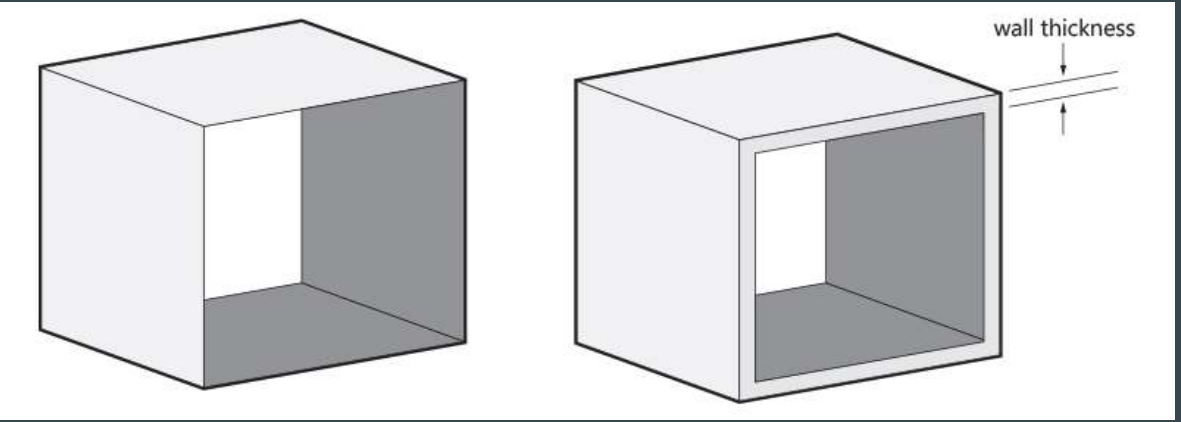
- Mediocre Strength

1.2mm

- Strong, Time Consuming

1.0mm

- Same time as 1.2mm, not as high of strength



Tolerances

- Printers are not 100% accurate to the model
- Parts with interferences/interactions
- Parts for exact dimensions
- Combined Parts
- Different for Every Printer
- ± .3 - .5 mm Safety

Hardware Issues

- Bed Leveling
- Clogged Extrusion Head
- Moisture in Filament
- Broken/ No Filament
- Electrical
- Cancel Button
- Problems with Preheat/Starting Print
- Micro SD cards
- Other Issues Guide

How to use Cura

- Set up printer
  - Other: Monoprice Select Mini (V2)
- Change Print Setup
  - Recommended to Custom
- Adjust all of the settings to fit your part
- Click Prepare to Slice your part
  - (if a time is already showing up than it may have auto-sliced)
- Plug in a MicroSD/SD card with the converter
- Click “Save to Removable Drive”
- Plug in the MicroSD or SD card into the 3D printer
- Press Print!

Test Your Knowledge!

1. Find a CAD file that you want to print
    a. If you do not have a modeled file you can find one on Thingiverse.com
2. Import the .STL file to Cura
3. Adjust Cura settings for ideal print
    a. Change your settings to the right 3D printer
    b. Change the print to ideal positioning
    c. Do you need any brims or support?
    d. Choose infill density
    e. Slice (make sure it's within the time limit)
4. Import the .gcode onto a sd card
5. Print!