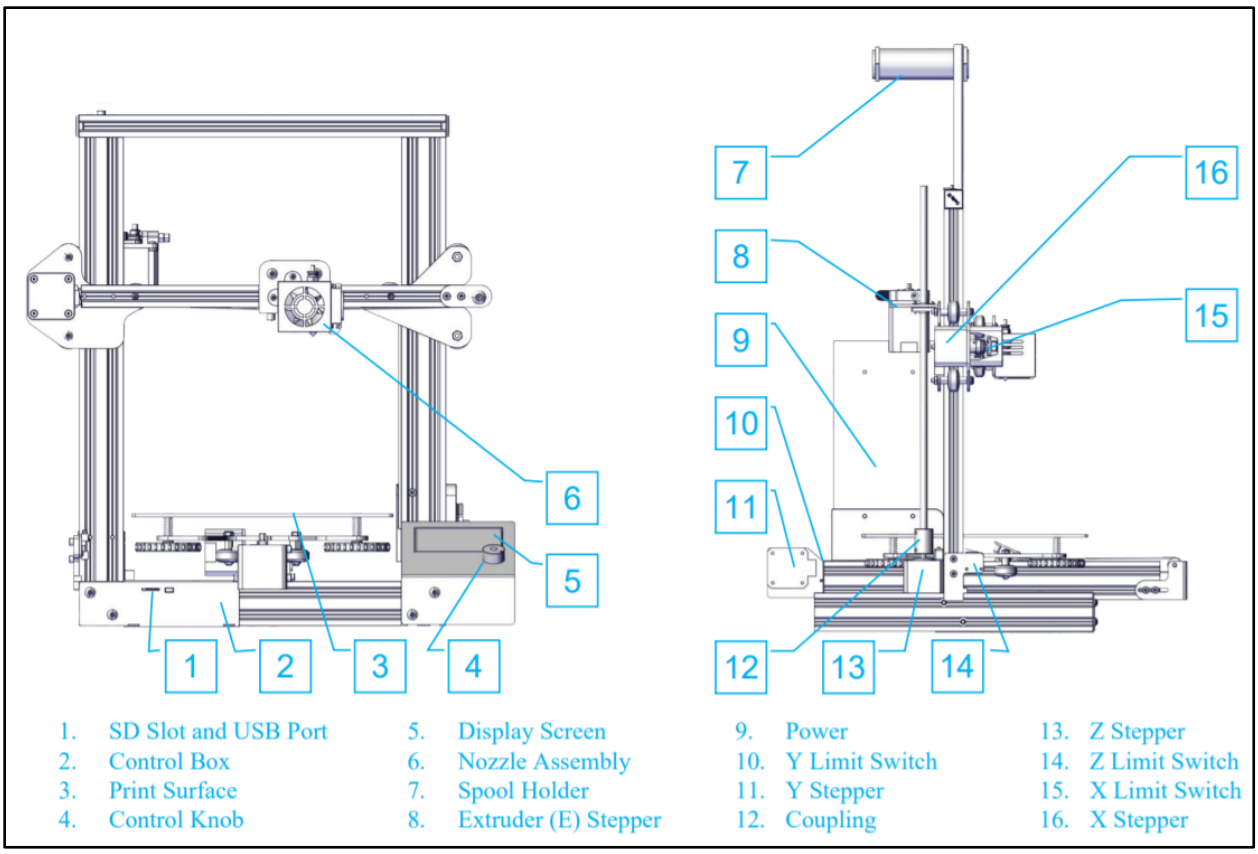
Table of Contents

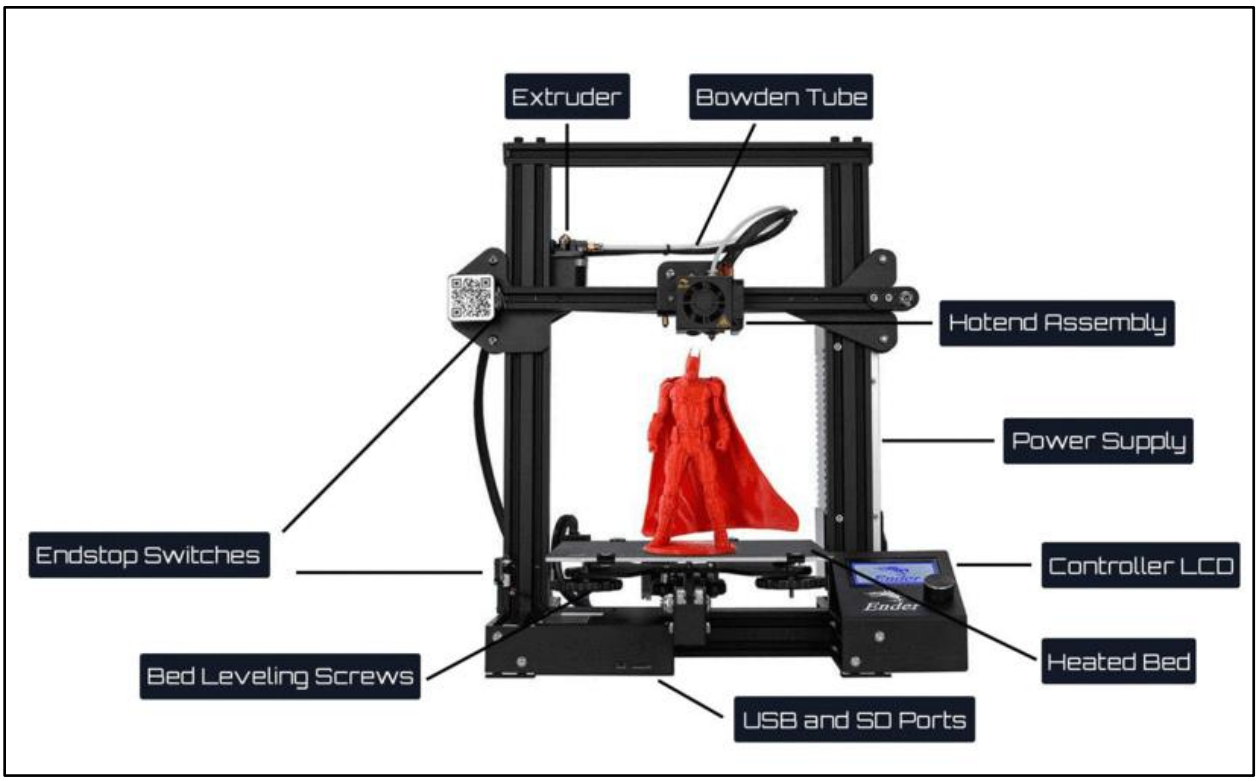
Basic Science and Terminology 2
Cura Settings 6
Layer Height 6
Retraction 6
Infill Percentage 6
Supports 6
Adhesion 6
Filament Settings 7
PLA 7
PETG 7
ABS 8
TPU 8
Stuff to Acquire to Improve Your Prints 9
Stuff to Buy in Order to Make Your Life Easier 9
Printer Upgrades 9
Programs 9
Design Knowledge 10
Tolerances: 10
Trouble Shooting 11
Bed not level 11
Under Extrusion 13
Z-Seam 14
Warping 15
Stringing 16
Fun Things to Print 17
Lithophanes 17
Maps 17
Miscellaneous Stuff 17
Websites 18

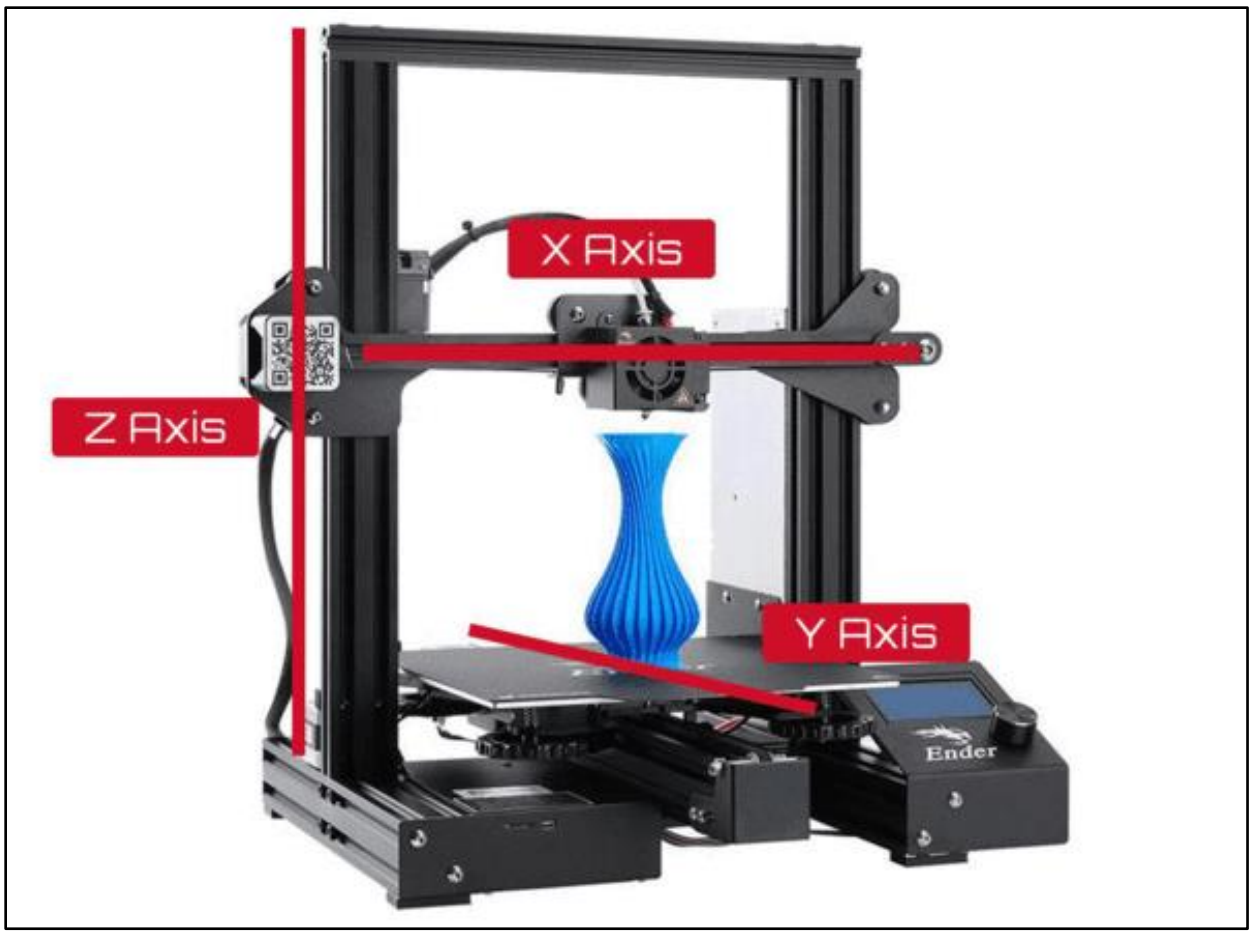
Basic Science and Terminology

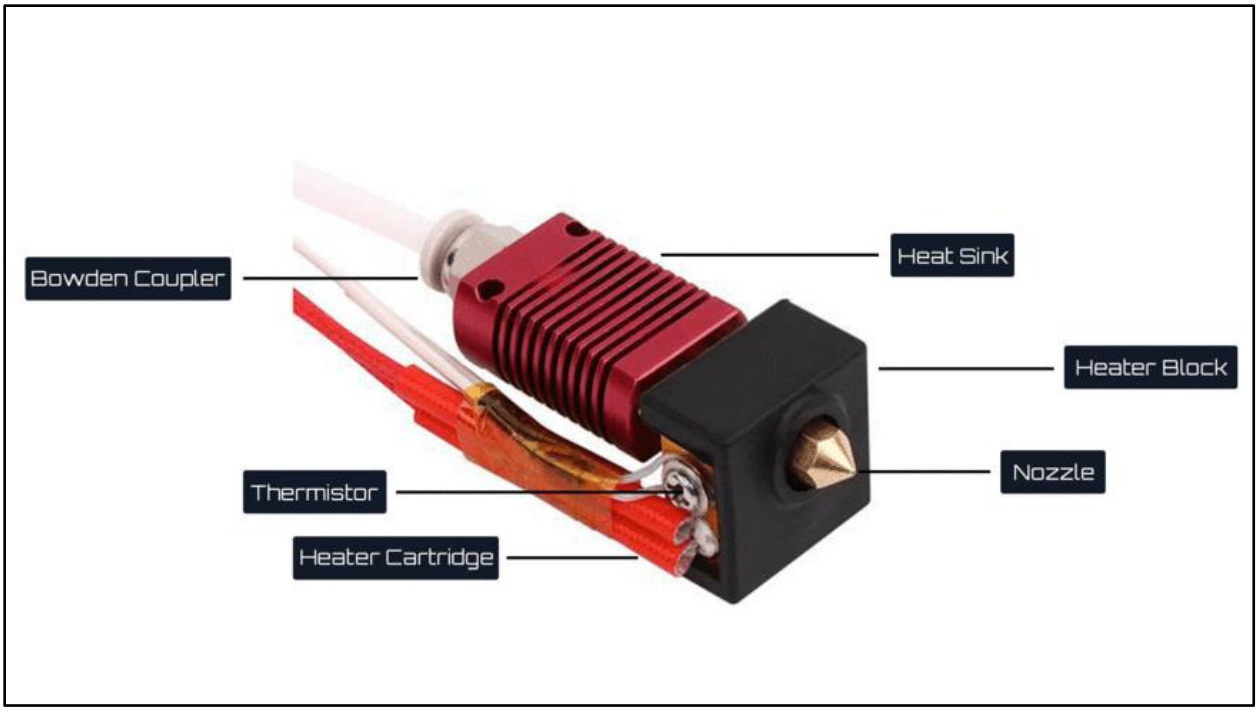
- **ABS:** Acrylonitrile Butadiene Styrene, a common thermoplastic known for its durability and resistance to heat
- **CAD:** Computer Aided Design, used to create models for 3D Printing
- **Extruder**: A motor with gears that pushes filament towards the printer's hot end.
- **Extrusion:** The action of plastic being pushed out of the nozzle
- **FDM (FFF) Printing:** FDM uses thermoplastics (meltable plastic filaments) to create an object layer by layer, starting from the build plate and building the model from the ground up.
- **G-Code:** Language that communicates with the printer and tells it what to do. G-code can be described as commands for a machine that instructs it to perform actions like moving components, extruding filament, and so on. While most makers don't directly interact with G-code, every time you set a temperature, position, or start a print, you're sending G-code commands to your printer.
- **Hot End:** Consists of a throat, heat break, heat block, and more depending on the hot end. The hot end uses a heating cartridge and a thermistor to heat the filament, which travels through the hot end's throat.
- **Infill:** Increases or decreases the amount of your model that is solid plastic within the slicer
- **Layer Height:** How thick each layer of your printed model is (adjusted in slicer)
- **Nozzle:** The small piece of hardware that screws into the hot end and is where the filament goes through last. The nozzle reaches high temperatures by absorbing the heat produced by the hot end. Once the filament hits the nozzle, the melted material is pressed through the small hole at the nozzle's tip and extruded.
- **Overhang:** Part of a 3D model that extends out without support under it
- **Over Extrusion:** Too much plastic is being extruded
- **PLA:** Polylactic Acid, arguably the most popular thermoplastic used in 3D Printing, known for its affordability, will warp in heat
- **Retraction:** The extruder reversing the direction of the filament (i.e. pulling it away from the hot end). Typically, this is done in short bursts between consecutive instances of extrusion. Among other things, it's a great way to prevent stringing.
- **Slicer:** Slices model layer by layer to create a G-code file for 3D printing. Slicer is a program that turns a 3D design into a list of commands that a printer uses to print a model. Without a slicer, your 3D printer can't print; it needs instructions in the form of G-code
- **STL:** Stereolithography, a form of 3D file most commonly used in 3D printing
- **Under Extrusion:** Not enough plastic is being extruded



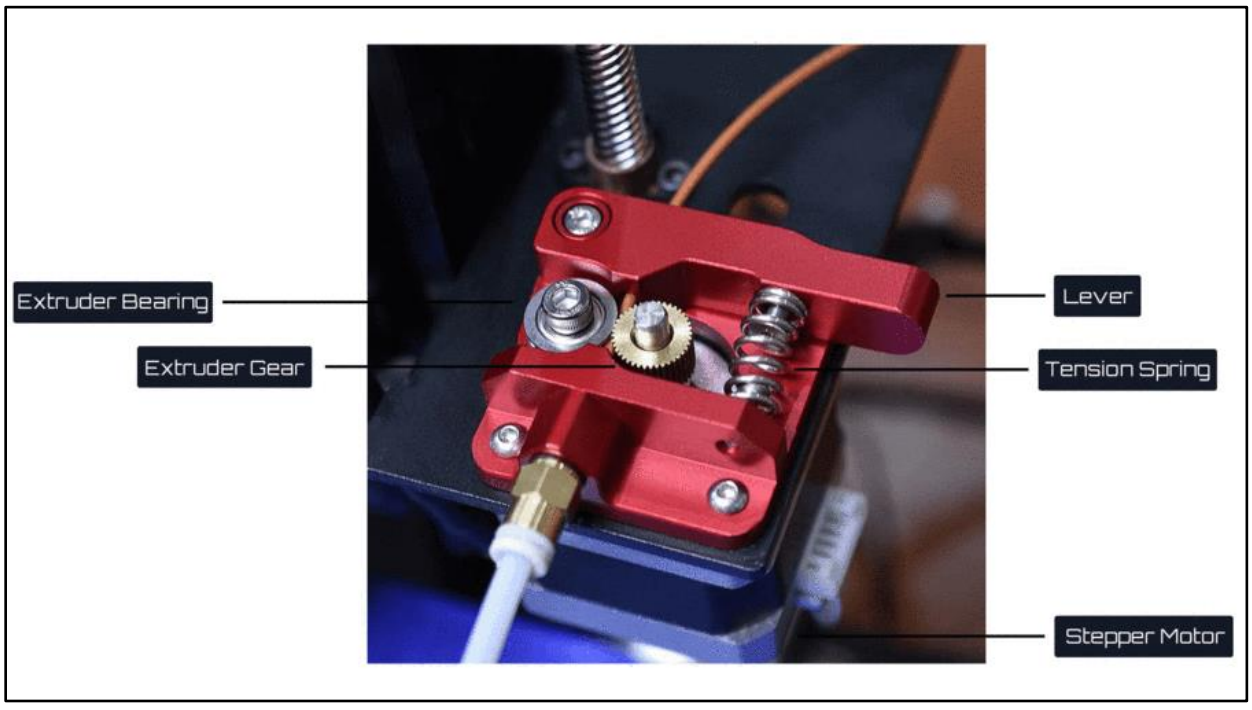
1. SD Slot and USB Port
2. Control Box
3. Print Surface
4. Control Knob
5. Display Screen
6. Nozzle Assembly
7. Spool Holder
8. Extruder (E) Stepper
9. Power
10. Y Limit Switch
11. Y Stepper
12. Coupling
13. Z Stepper
14. Z Limit Switch
15. X Limit Switch
16. X Stepper




X Axis
Z Axis
Y Axis
Ender


Bowden Coupler
Heat Sink
Heater Block
Nozzle
Thermistor
Heater Cartridge

Teflon tube
Heat sink
Pneumatic fittings
Fan cover
Insulation silicone case
Thermistor
Heatbreak
Fan
Brass nozzle
(standard 0.4mm)
Heated aluminum block


Extruder Bearing
Extruder Gear
Lever
Tension Spring
Stepper Motor

Cura Settings

https://all3dp.com/2/ender-3-cura-settings-best-ender-3-cura-profile/

Layer Height

- "Resolution" of a print
- 0.12mm = Fine detail and slow
    - Use for things that require a lot of detail
- 0.28mm = Course detail and not as slow
    - Use for things that don't require a lot of detail

Retraction

- Used to prevent stringing
- Raise if there is too much filament oozing out of the nozzle, leaving your print with a bunch of strings or clumps on the outer edges
- Lower if nozzle gets clogged or the prints take too long
- Retraction distance: Amount of filament the extruder will retract from the nozzle
    - It's good to start with a setting of 5 mm and adjust it up or down by 1 mm until you have it just right. The goal is to get the lowest possible number while reducing as much stringing as you can. A good range is 3 to 7 millimeters. Too low of a number won't reduce stringing, too high and you could cause damage to your filament and increase your print time.
- Retraction Retract Speed: The speed in mm/s the extruder pulls the filament out of the Bowden tube and the nozzle.
- Combing Mode:

Infill Percentage

- How dense a print is
- 100% = Thin or small objects only and lithophanes
- 50% = Objects that will require a lot of strength
    - You should never need to be between 51% and 99% because it is unnecessary
- 21% - 49% = Use best judgement on how much structural integrity you think you'll need
- < 20% = Decorations

Supports

- Increase how much material is used but also allow you to print parts with overhangs.
- Tree support is recommended most of the time
- Traditional Support is recommended

Adhesion

- **Skirt:** A detached perimeter that surrounds your model. A skirt provides no benefit to bed adhesion, but it acts to "prime" the nozzle before the actual printing of your model begins and to ensure your bed is level.

- **Brim:** A multi-line skirt that extends from the base of the part to be printed (i.e. the two are connected). A brim, which is meant to be removed after printing, provides added adhesion without wasting too much filament or time. It can also be an effective solution against warping.
- **Raft:** A 3D printed platform upon which a part is printed. Its role is to effectively eliminate any problems that might occur between the part and the print bed. Of these three options, a raft uses the most filament and takes the most printing time, but it's also the most effective.

Filament Settings

PLA

- Printing (Nozzle) Temperature: 190-220°C
    - Recommended Starting Nozzle Temperature: 207°C
- Bed Temperature: 50-60°C
    - Recommended Starting Bed Temperature: 60°C
- Speed:
    - Print Speed: 25–65mm/s
        - Recommended Starting Print Speed for Large Objects: 50mm/s
        - Recommended Starting Print Speed for Small Objects: 30mm/s
    - Infill Speed: This should be the fastest speed (besides travel speed)
    - Wall Speed: This should be slower than infill speed
    - Top/Bottom Speed: This should be your slowest speed
    - Travel Speed: 100-150 mm/s
        - Recommended Starting Travel Speed: 150mm/s
- Retraction: 4-6mm @ 20-30 mm/s
    - Recommended Starting Retraction Settings: 6mm @ 25m/s

PETG

- Printing (Nozzle) Temperature: 225 - 240 °C
    - Recommended Starting Nozzle Temperature: 240°C
    - Caution: Do NOT go above 240°C or you risk melting the PTFE Tube
- Bed Temperature: 70 °C
- Speed: 20-40 mm/s
    - Recommended Starting Speed: 30mm/s
- Retraction: 4-6.5mm @ 25-35 mm/s
    - Recommended Starting Retraction Settings: 5.5mm @ 25mm/s
- Fan Speed:
    - 100% = High Quality, Low Strength
    - 0% = Lower Quality, Higher Strength
    - Recommended Starting Fan Speed: 50%

ABS

- Print temperature: 210°C – 240°C
  - Recommended Starting Nozzle Temperature: 230°C
- Print bed temperature: 80°C – 110°C
  - Recommended Starting Bed Temperature: 100°C
- Speed: 45-65 mm/s
  - Recommended Starting Speed: 50mm/s
- Retraction: 4 mm @ 40 mm/s

TPU

- Print Temperature: 210 - 230 °C
- Print Bed Temperature: 30 - 60 °C
- No Retraction
- Print speed: Around 30 mm/s

Stuff to Acquire to Improve Your Prints

Stuff to Buy in Order to Make Your Life Easier

- Glue stick (purple disappearing)
  - This will help with bed adhesion
- 75% Alcohol
  - Used to clean your build plate (or anything else)
- Dish Soap
  - Used to clean the oils (and glue) off your build plate
- Microfiber towel
  - Used to ensure a spotless build plate
- Crazy Glue
  - Used to glue two parts together
- Caliper
  - Used to help in the design process and to ensure tolerances are correct
- Thickness (feeler) Gauge
  - Used instead of paper to level your bed more accurately
- Dry Enclosure
  - Used to prevent filament from absorbing too much moisture
  - De-Humidifier
    - https://www.amazon.com/Improved-Eva-dry-333-Renewable-Dehumidifier/dp/B000H0XFCS/ref=sr_1_1?dchild=1&keywords=Eva-Dry+Wireless&qid=1604083334&s=home-garden&sr=1-1

Printer Upgrades

- Yellow Springs
  - Creates more stable build plate
- Glass Bed
  - Creates a more flat build plate
- Filament Guide
  - Prevent filament from rubbing against the Z-Screw
  - https://www.thingiverse.com/thing:2917932
- Slot Covers
  - Prevents stuff from entering the t-slots
  - https://www.thingiverse.com/thing:3379068

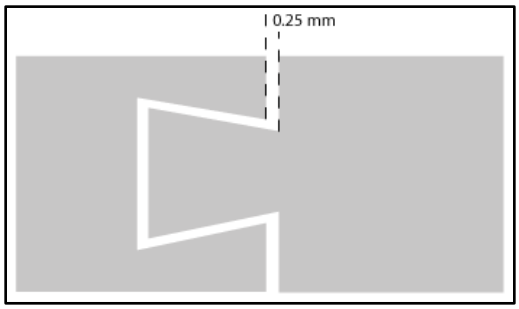
Programs

- Cura
  - https://ultimaker.com/software/ultimaker-cura
- Autodesk Fusion 360
  - https://www.autodesk.com/products/fusion-360/free-trial

Design Knowledge

Tolerances: (0.25mm) - This rule applies when designing objects that should fit together.

- As a general rule each face of one of the part should be offset by this distance.
- As an example if you have a 10mm diameter hole the tube fitting inside it should be 9.5mm. the

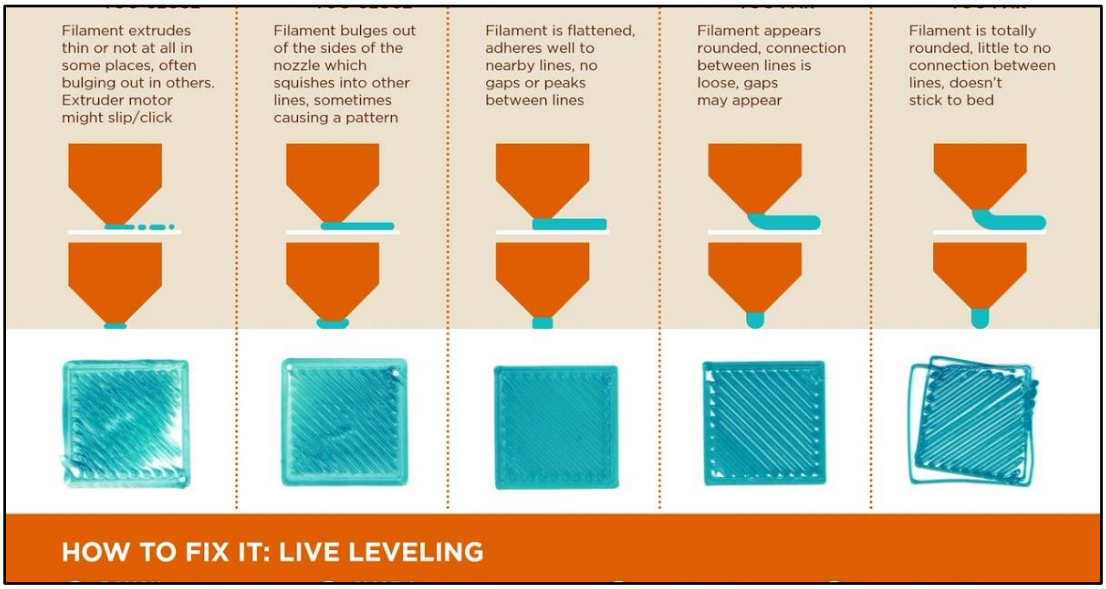


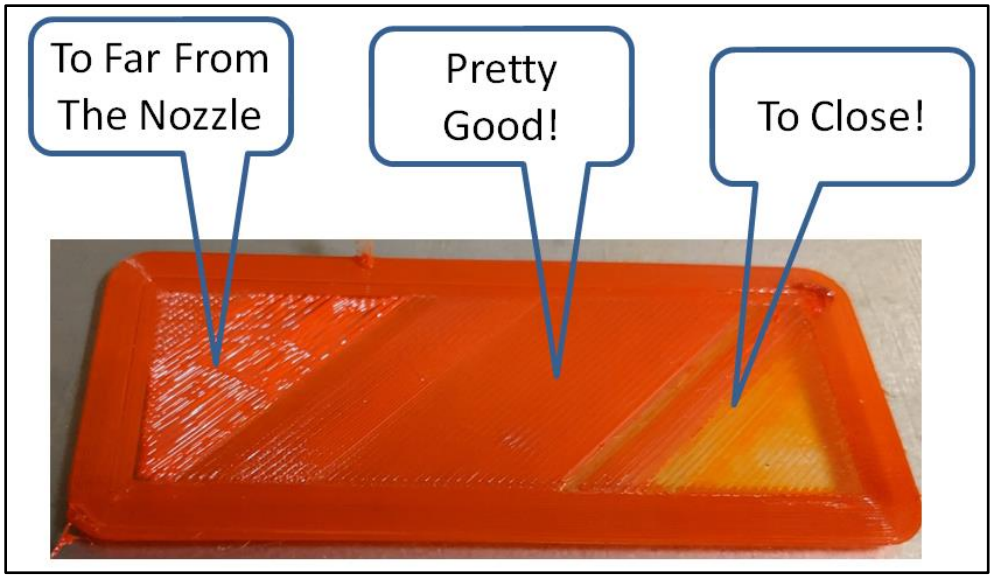
Trouble Shooting

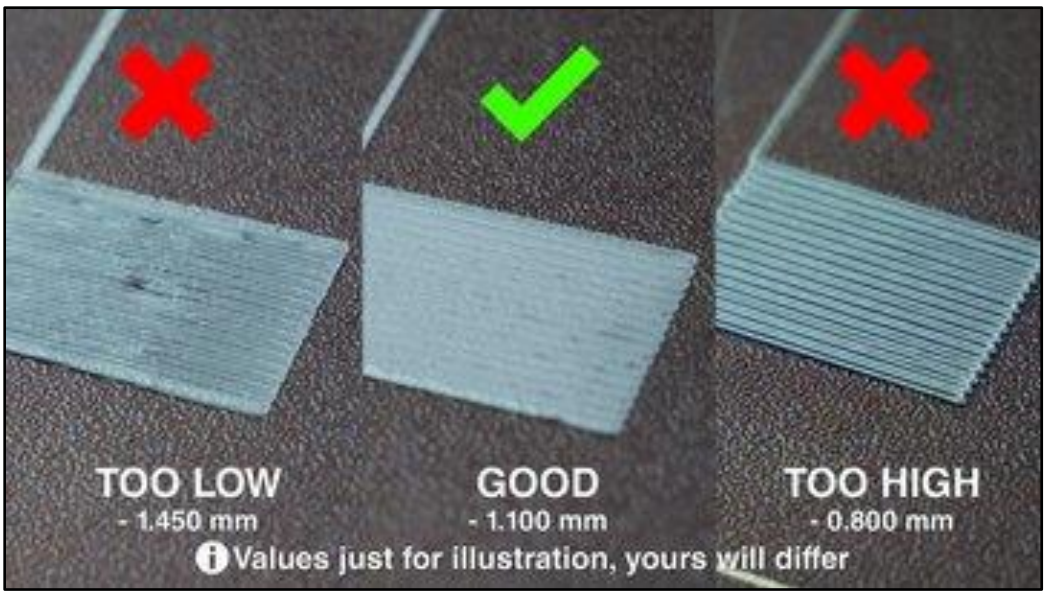
Bed not level

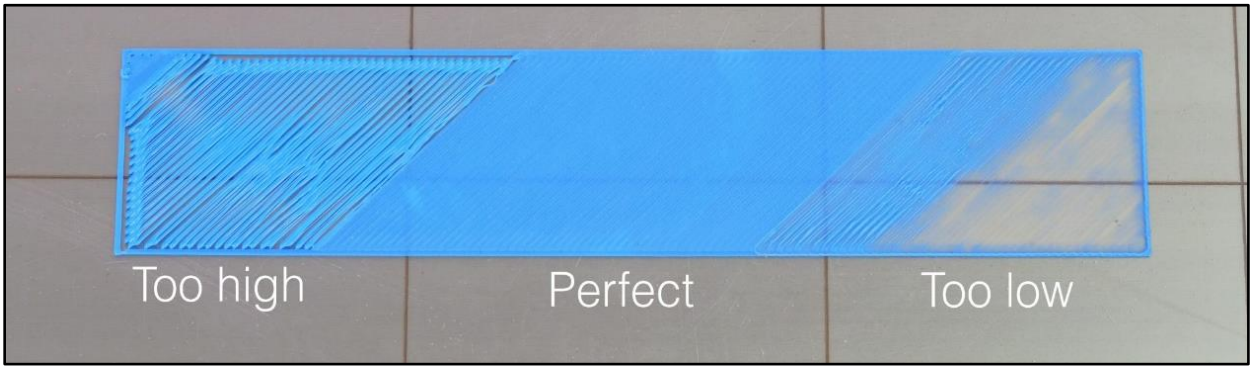
- Causes
  - Bed Not level

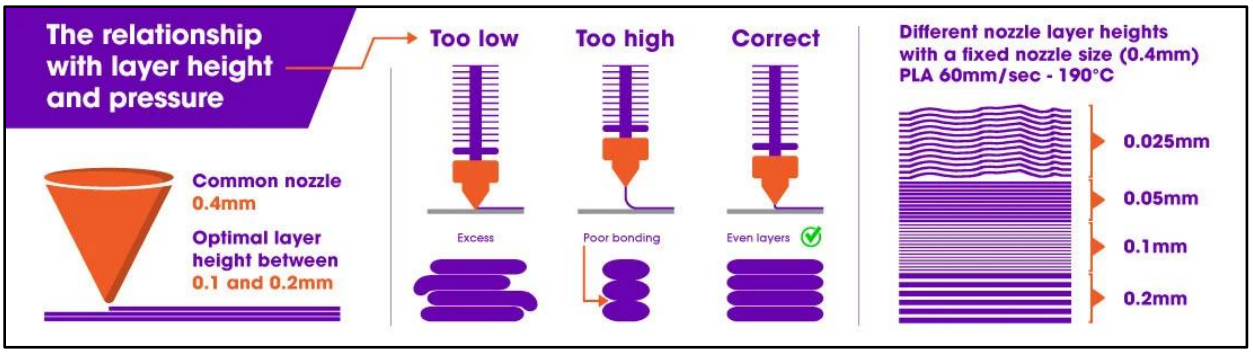
- Solutions
  - Level again
  - Learn to live level (leveling your bed while printing)
  - Buy thickness gauges to use instead of paper
  - Get a BL Tough






TOO LOW
- 1.450 mm
GOOD
- 1.100 mm
TOO HIGH
- 0.800 mm
Values just for illustration, yours will differ

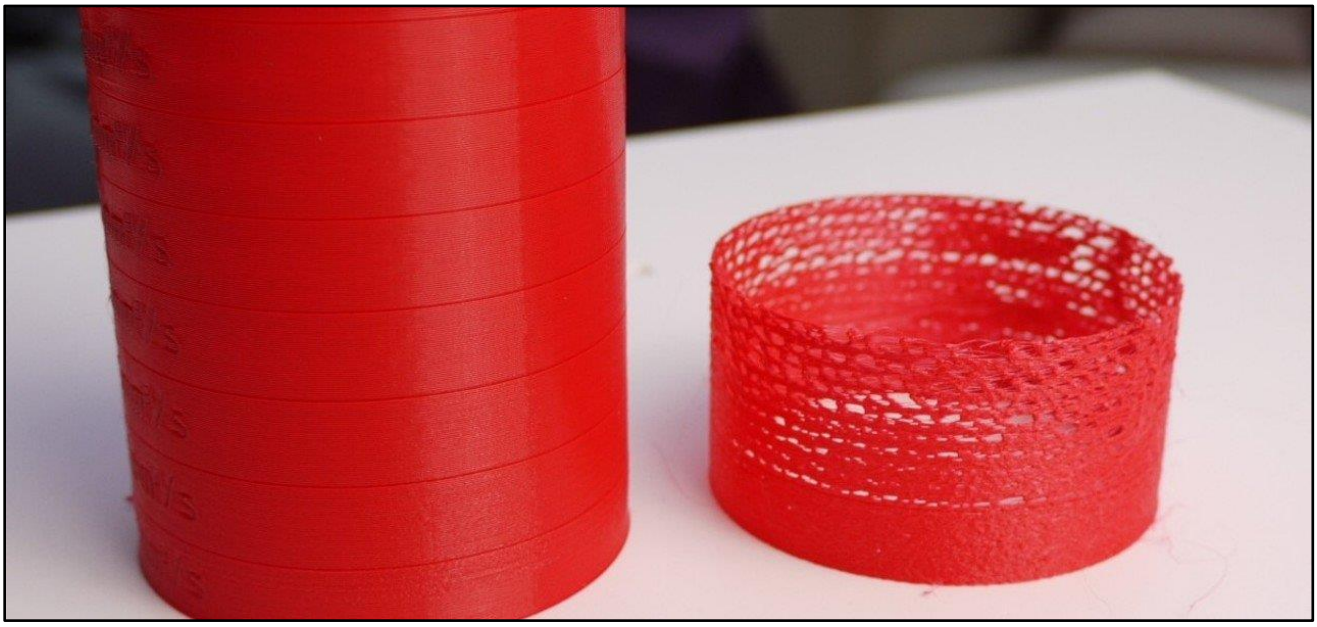

Too high
Perfect
Too low

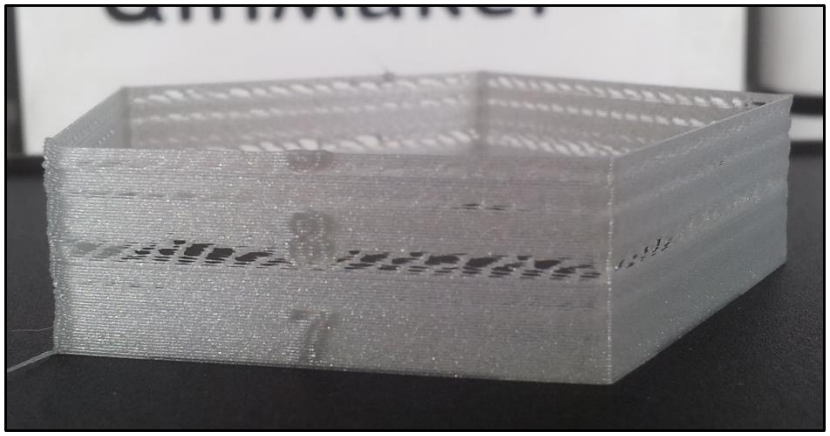

The relationship with layer height and pressure
Common nozzle 0.4mm
Optimal layer height between 0.1 and 0.2mm
Too low
Too high
Correct
Excess
Poor bonding
Even layers
Different nozzle layer heights with a fixed nozzle size (0.4mm) PLA 60mm/sec - 190°C
0.025mm
0.05mm
0.1mm
0.2mm

Under Extrusion

- Causes
    - Many things

- Solutions
    - Nozzle is clogged (clean using piano wire)
    - PTFE Tube is not flush with the nozzle
    - Filament is jammed in the PTFE Tube or hot end
    - Retraction settings are too high
    - Raise the nozzle temperature
    - Calibrate E-Steps

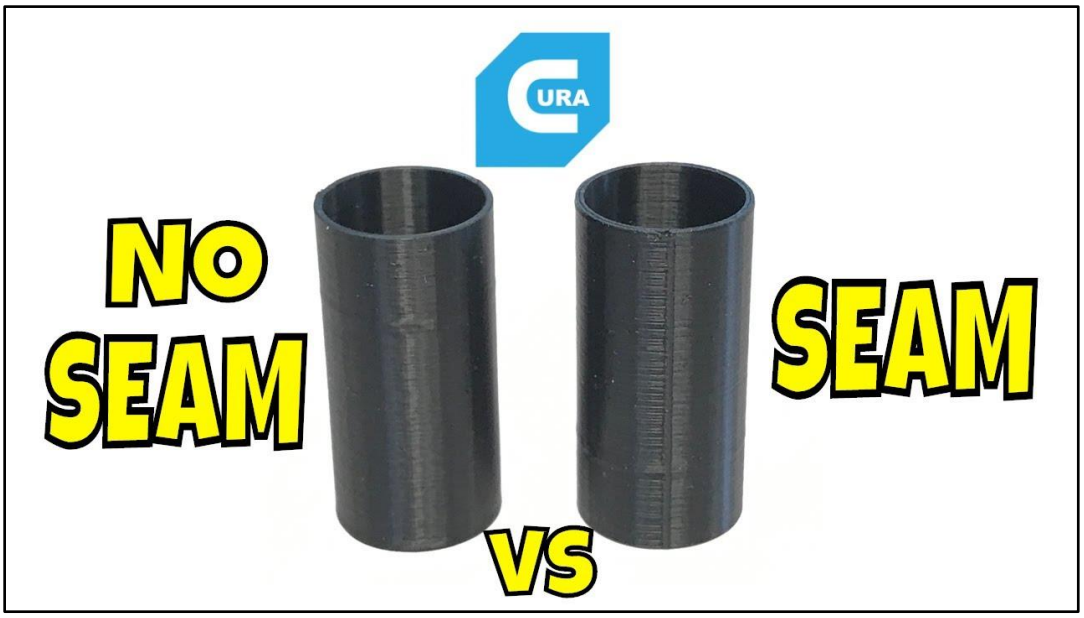


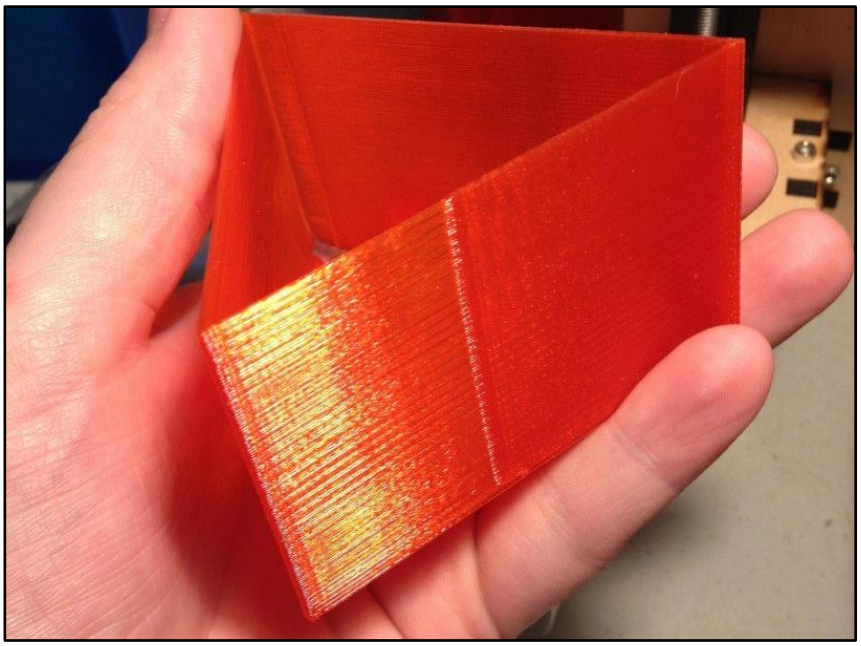


Z-Seam

- Causes
  - The nozzle starts at that location when beginning a new layer

- Solutions
  - You cant really get rid of it
  - You can change the location of it to "hide it" in Cura settings
    - "Z Seam Alignment" – Try selecting "Sharpest Corner"

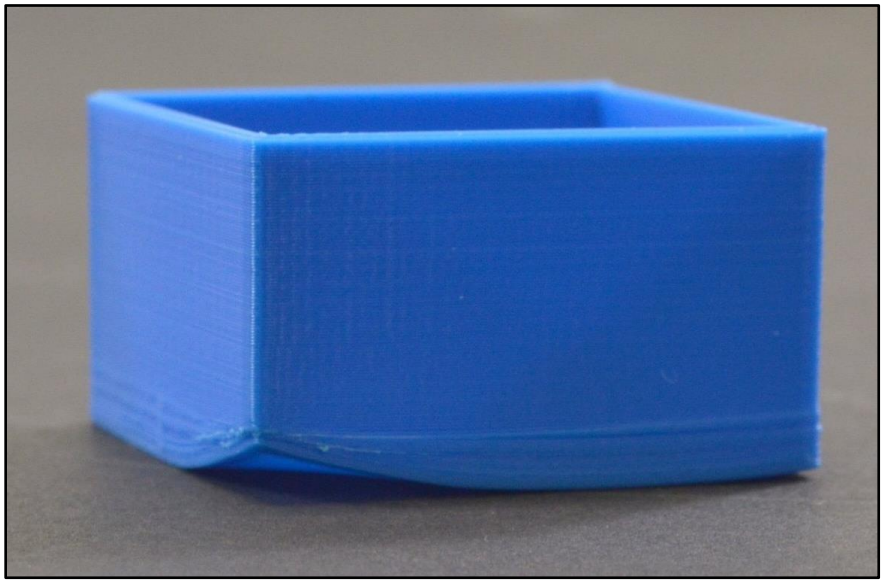


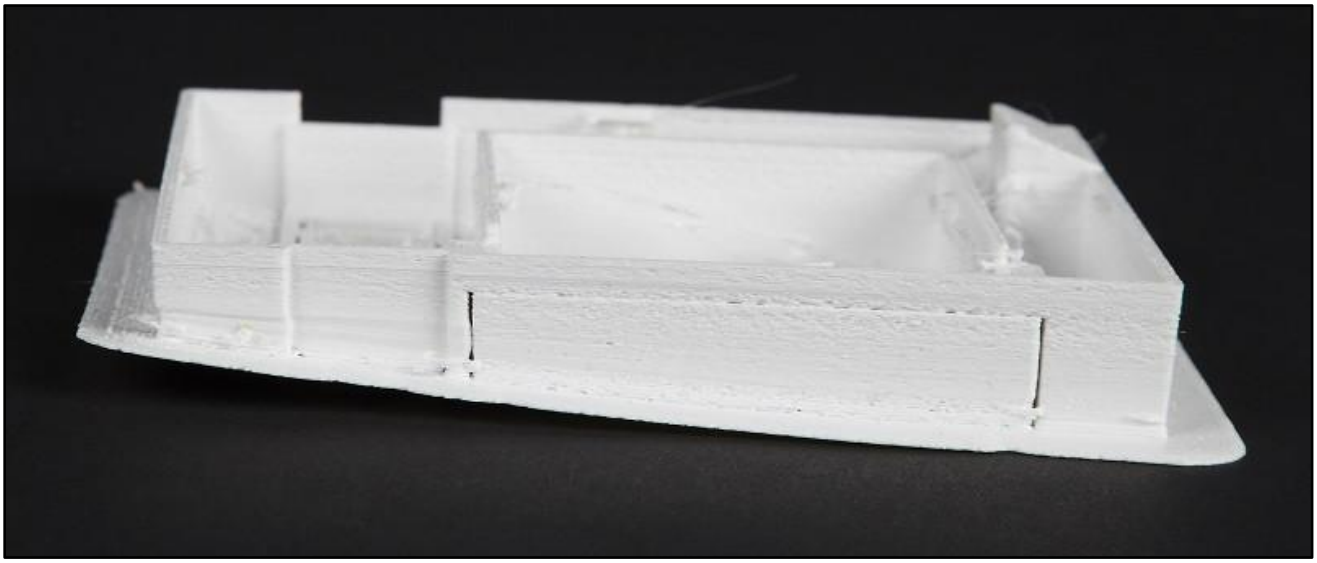


Warping

- Causes
    - Poor bed adhesion

- Solution
    - Higher bed temperature
    - Apply glue stick to the build surface
    - Put a raft on your print

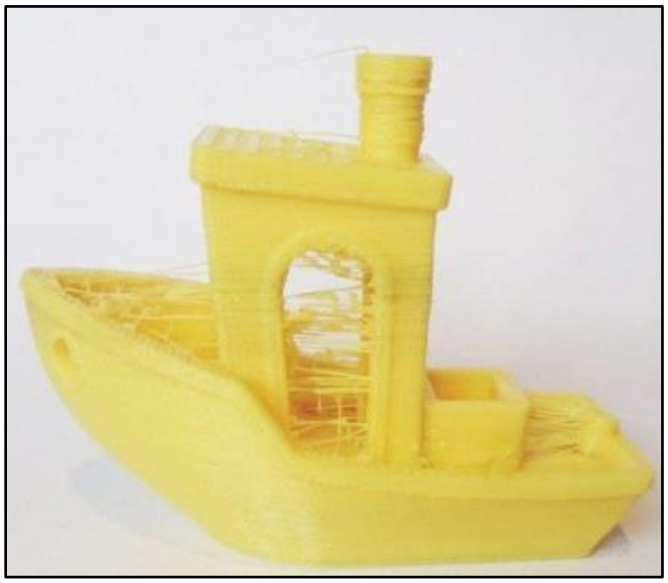


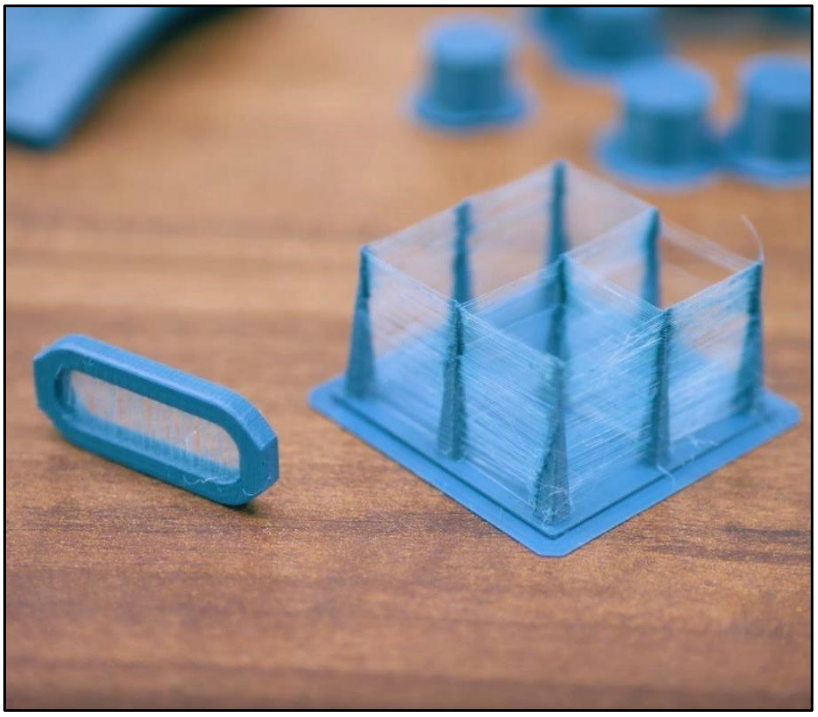


Stringing

- Causes
    - Filament comes out of the nozzle while the nozzle is moving

- Solution
    - Tweak Retraction Settings
    - Nozzle temperature is not correct





Fun Things to Print

Lithophanes

- Lowest possible layer height
- Speed: Low (30mm/s for PLA)
- Infill – 100%
- Only use LED light bulbs
- https://itslitho.com/
- http://3dp.rocks/lithophane/

Maps

- https://touch-mapper.org/en/

Miscellaneous Stuff

- Board Game Pieces
- Can Handles
  - https://www.thingiverse.com/thing:1658338
- Toothpaste Roller
- Bag Clips
  - https://www.thingiverse.com/thing:330151
- The Moon
- Key Chains
- Tensegrity Table
- Scan yourself using an Xbox Kinect
- Coozie
- Ear Savers
- Frisbee Clip
  - https://www.thingiverse.com/thing:2537161
- Golf Ball Belt Clip
- Universal Bottle Opener
- Gifts
- Jewelry

Websites

- https://www.creality3dofficial.com/blogs/news/ultimate-3d-printing-beginners-guide
- https://www.thingiverse.com/
- https://all3dp.com/