

# **3D Printing: The (Additive) Manufacturing Technique for Everybody**

Dylan Ross

Monday Night Meeting — 2019/04/22

# Overview

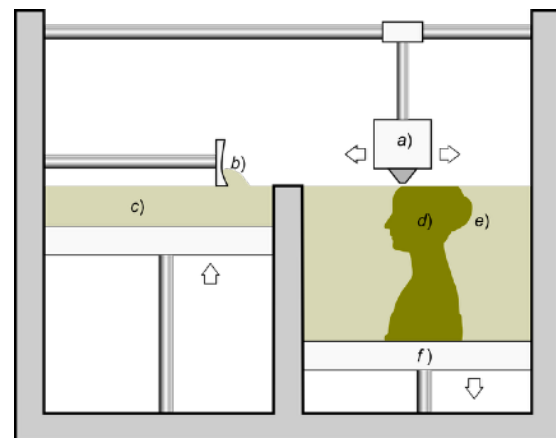
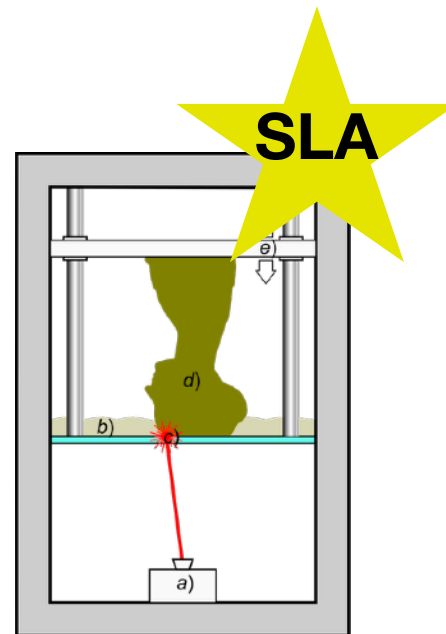
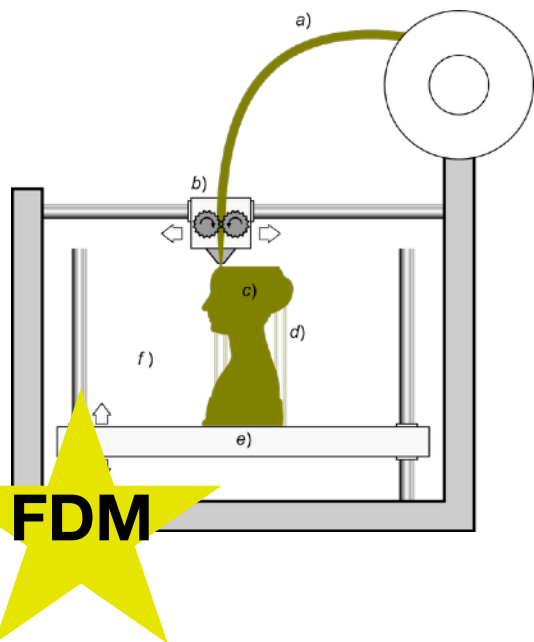
- Introduction
  - compare/contrast with other manufacturing techniques
  - types of 3D printers
  - definitions, terminology
- Practical Guide to 3D Printing
  - obtaining/producing 3D models
  - slicing, gcode
    - ▶ material selection
    - ▶ quality settings, infill
    - ▶ printing orientation, supports, adhesion
  - benchmarking printer performance
  - health and safety
- Showoff

# Introduction

# Additive Manufacturing vs. Machining (*a.k.a* Subtractive Manufacturing)

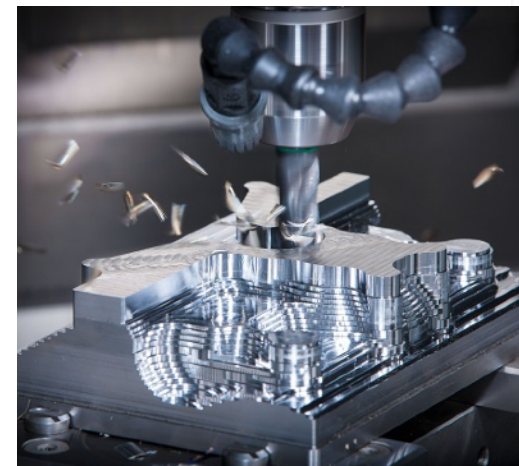
## AM

- add material to build final object from nothing
- two general processes:
  - extrusion
  - selective fusion



## SM

- remove material from raw stock to reveal desired geometry
- three principle machining processes:
  - turning
  - milling
  - drilling

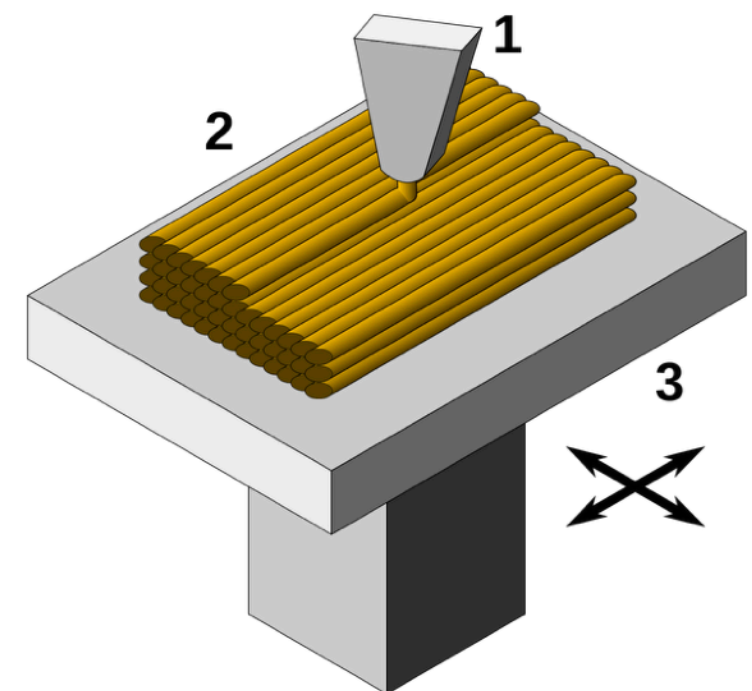
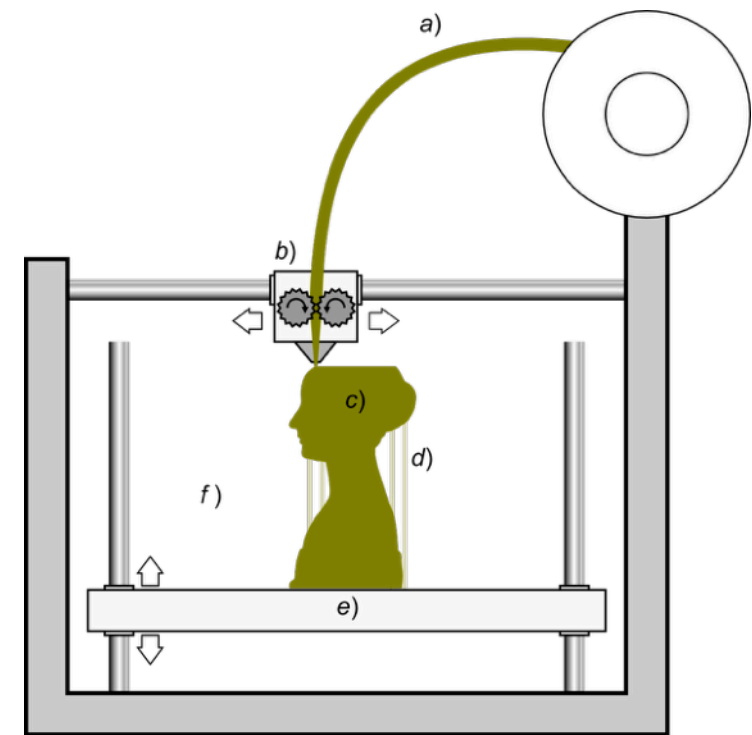


10 x 54

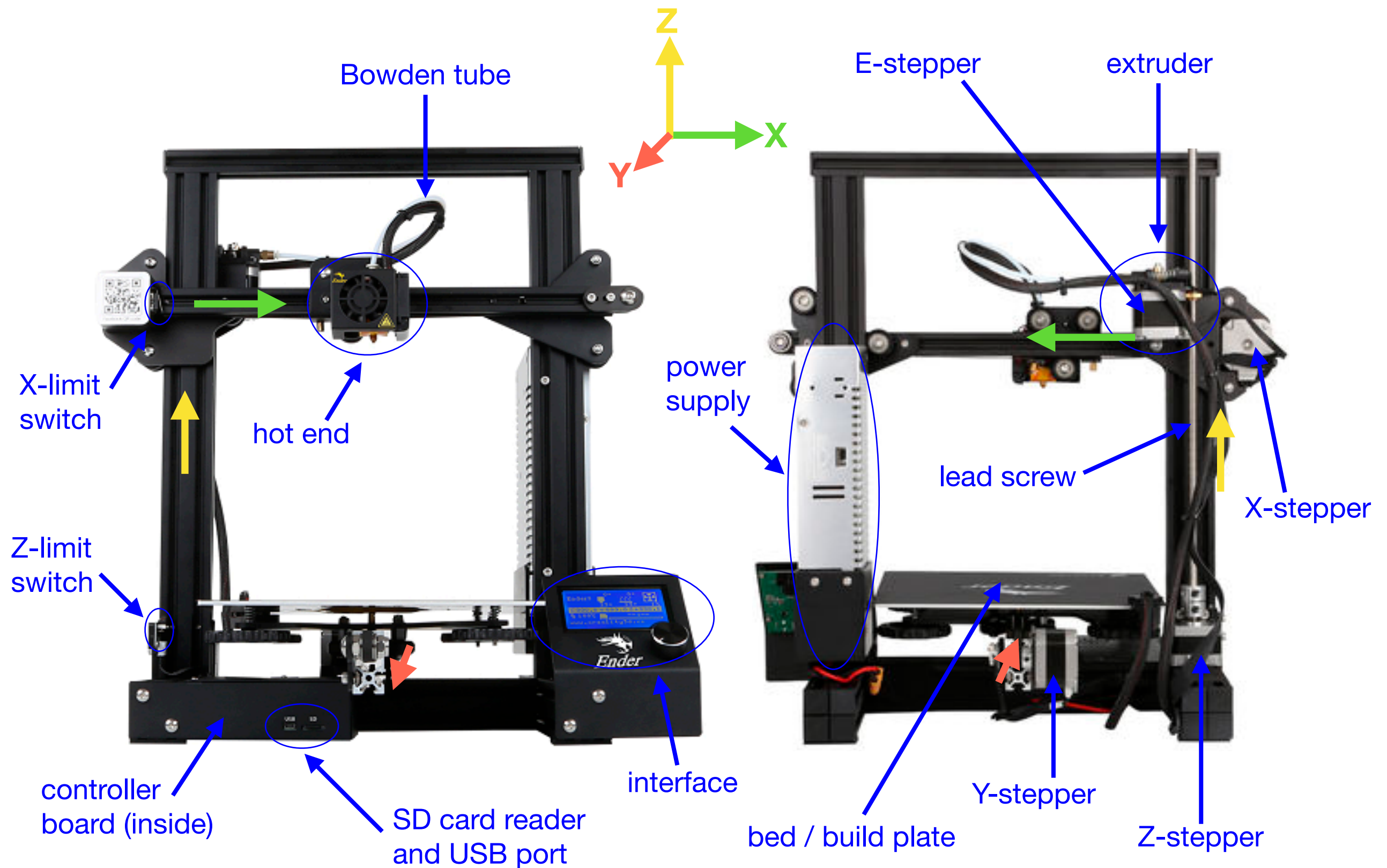


# Fused Deposition Modeling (FDM)

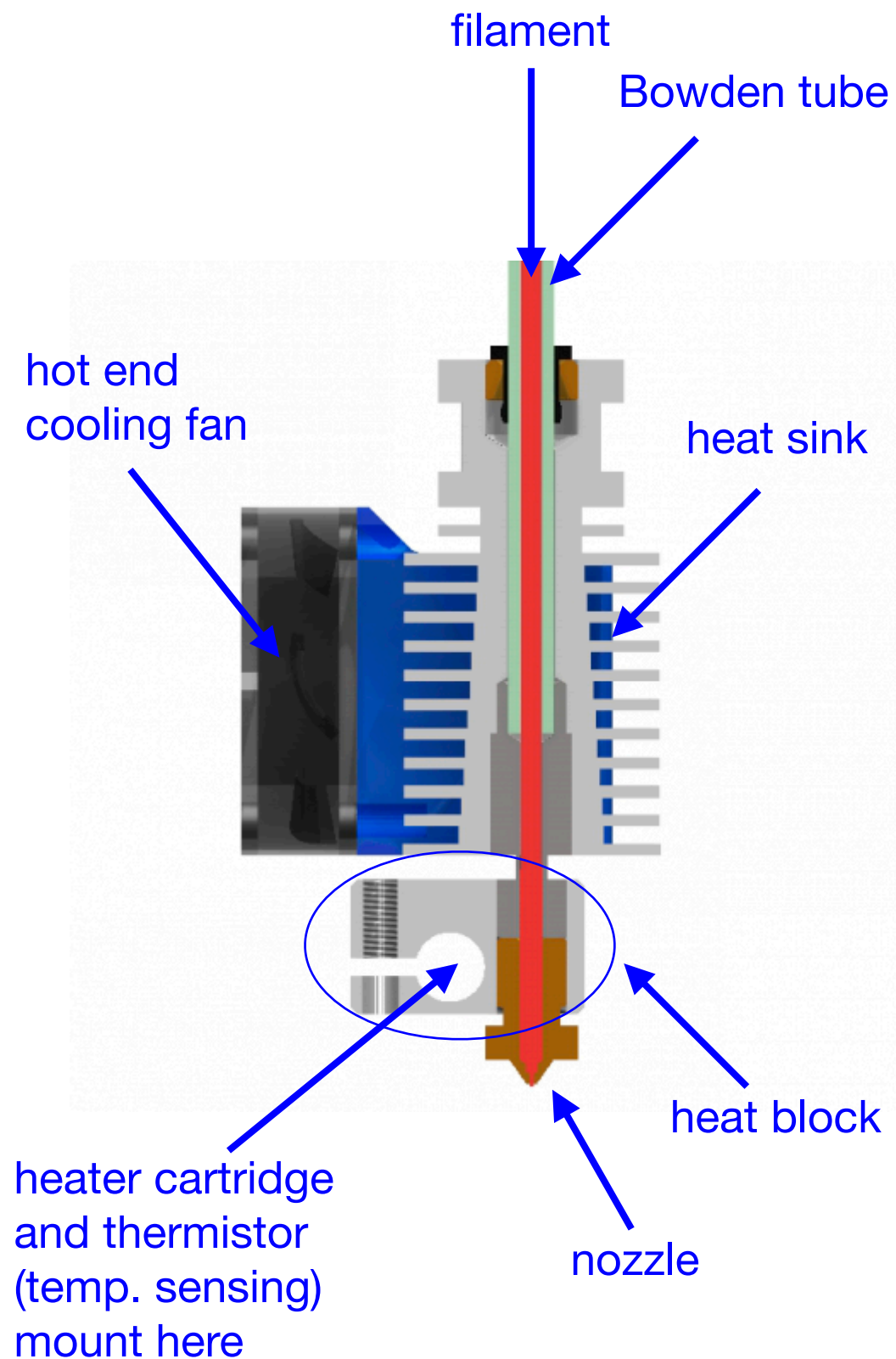
- thermoplastic filament extruded through heated nozzle
- nozzle moves relative to build surface, depositing molten plastic
- molten plastic cools and hardens in place after being deposited
- model is built up layer by layer



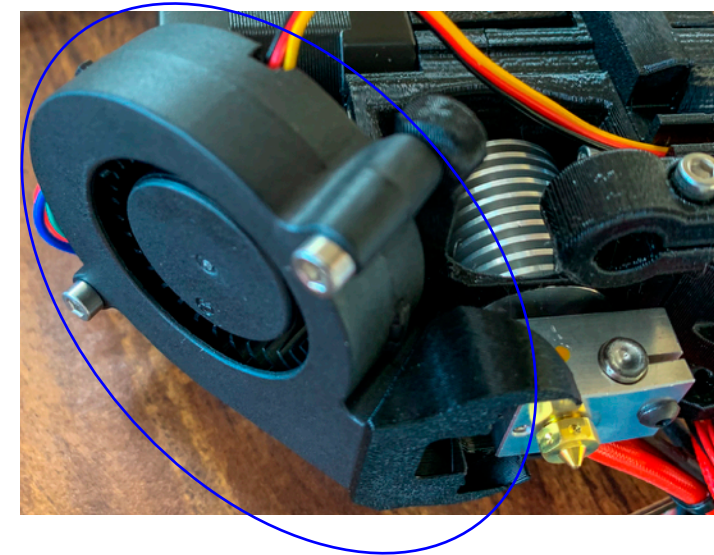
# Anatomy of an FDM Printer



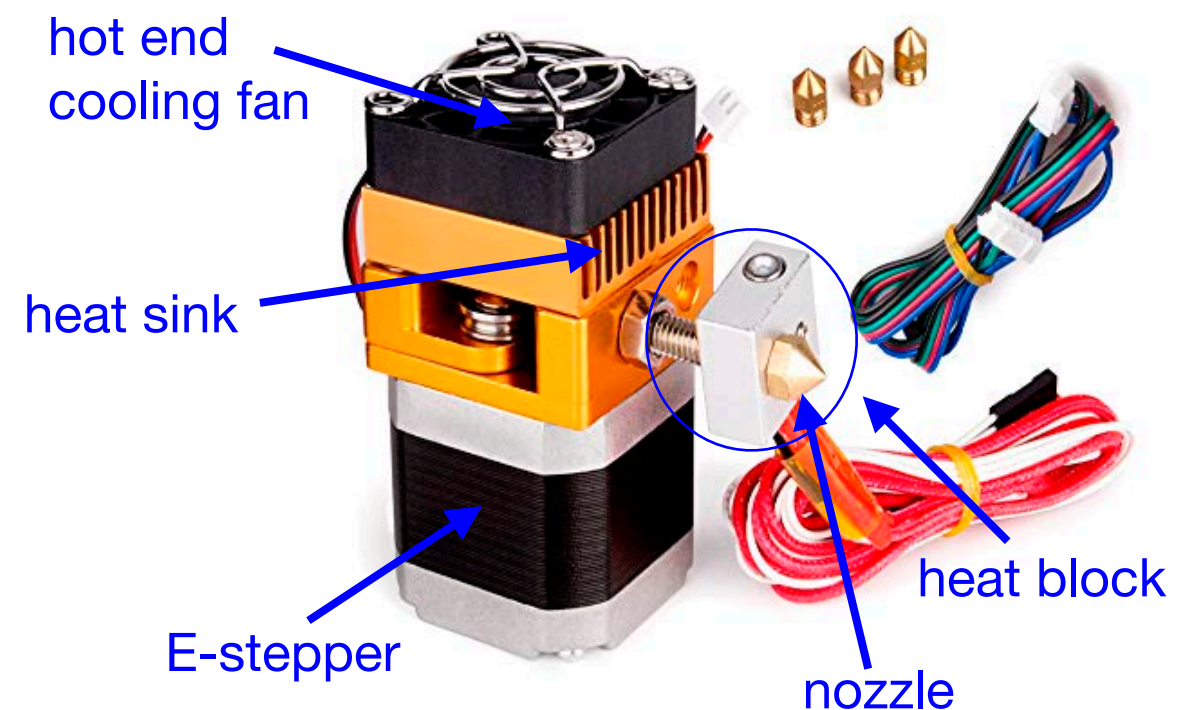
# Anatomy of an FDM Printer — Hot End



*separate **part-cooling fan** sometimes included in the hot end assembly*



*extruder sometimes directly coupled to hot end — no Bowden tube*



# Practical Guide to 3D Printing

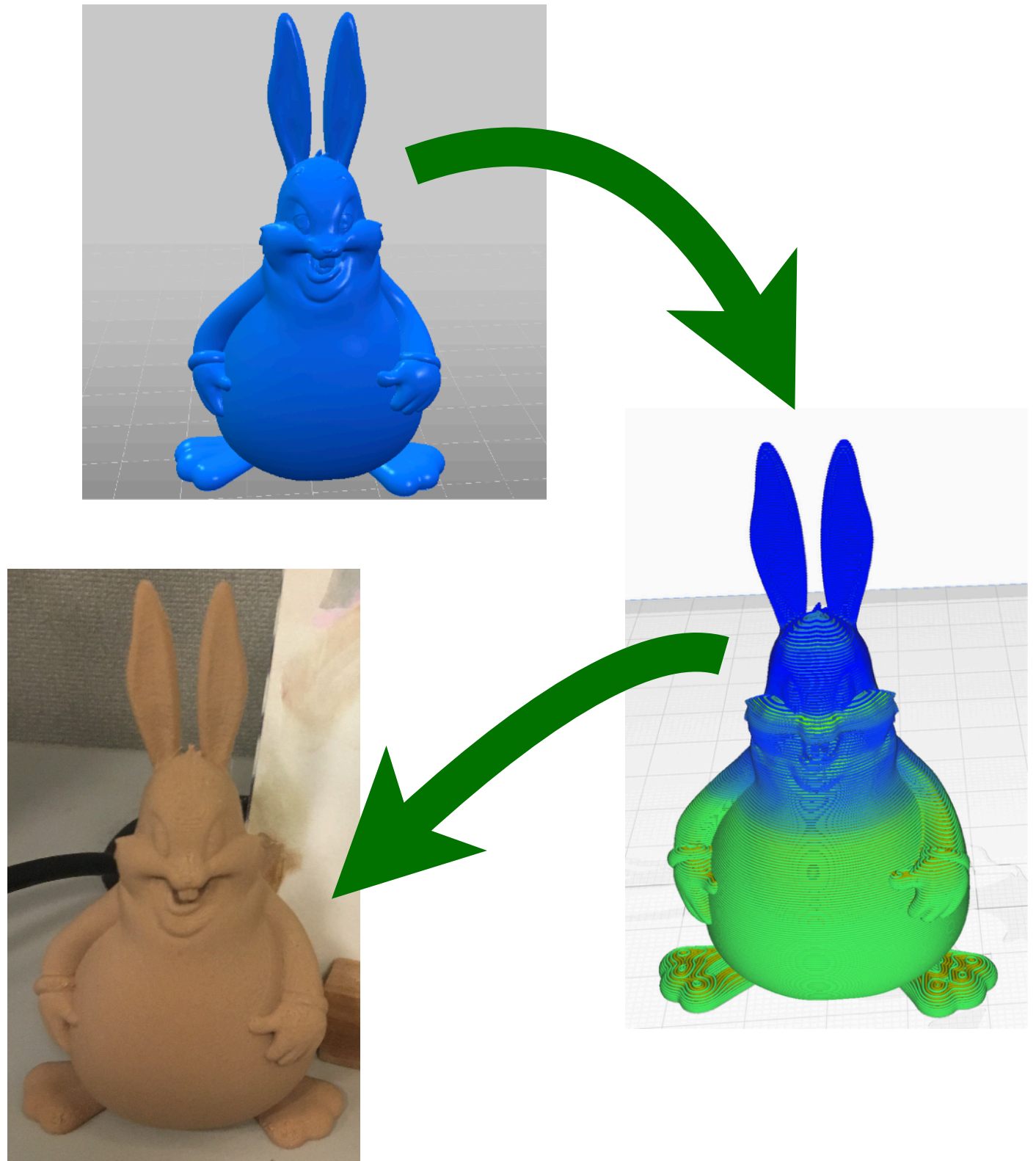
*Throughout this  
guide, whenever I have  
miscellaneous tips or otherwise  
feel the need to editorialize, I'll  
use this little guy*





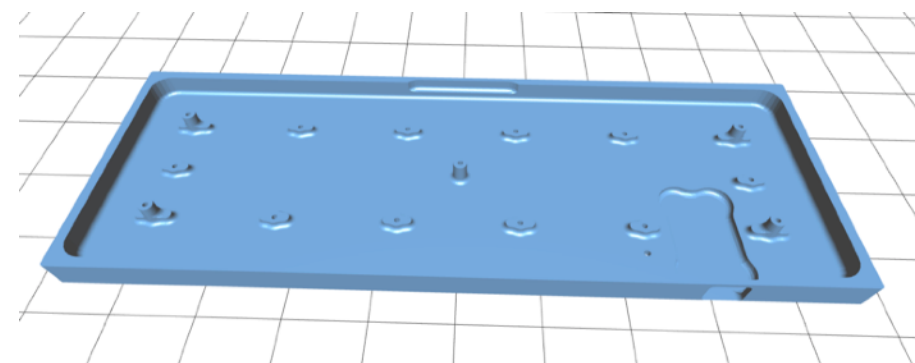
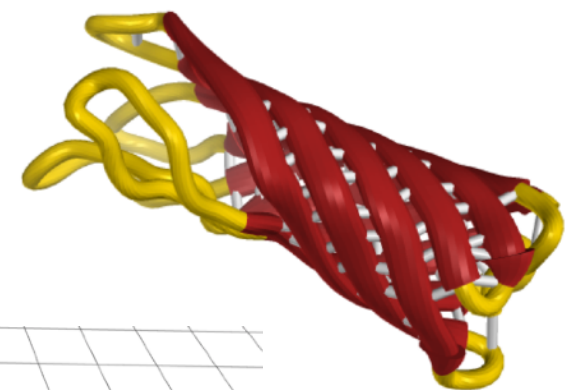
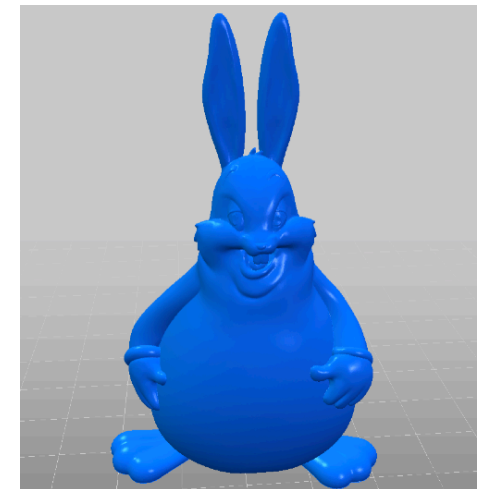
# 3D Printing in 3 Easy Steps!

1. obtain a 3D model of an object
2. produce printer-specific instructions for building the object
3. print the object



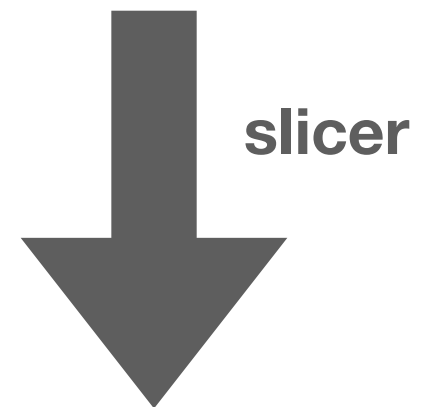
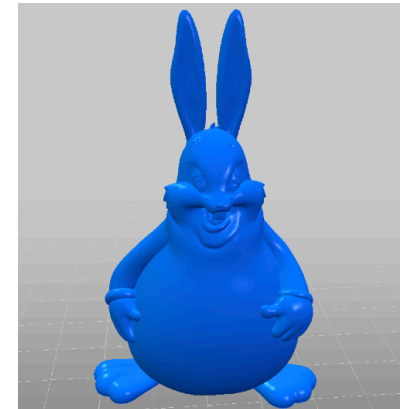
# Where can we get 3D models to print?

- Design models using CAD software (or modify existing models)
  - Tinkercad (<https://www.tinkercad.com/>)
  - Fusion 360 (<https://www.autodesk.com/products/fusion-360/overview>)
  - FreeCAD (<https://freecadweb.org/>)
  - Blender (<https://www.blender.org/>)
  - UCSF Chimera (<https://www.cgl.ucsf.edu/chimera/>)
- Obtain pre-made models
  - Thingiverse (<https://www.thingiverse.com/>)
  - NIH 3D Print Exchange (<https://3dprint.nih.gov/>)
  - GitHub (<https://github.com>)



# Slicer Introduction

- Program responsible for converting 3D model into instructions that a 3D printer can follow to build an object (gcode)
- generated gcode is specific to the model of 3D printer and filament type, generally not transferrable between different printers or filaments
- Slicer settings are critical determinants of the quality of the printed part
- popular slicer programs:



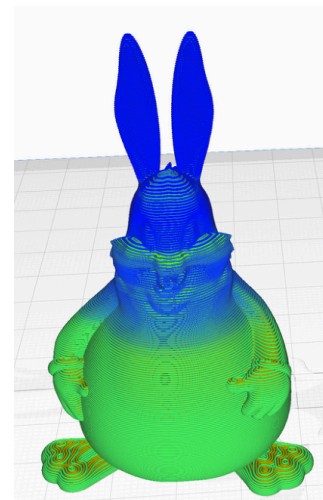
**Ultimaker Cura**

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



**Simplify3D**

<https://www.simplify3d.com/>

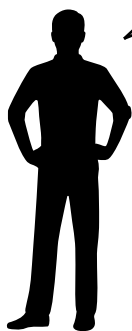


```
G90 ; set to absolute positioning
M82 ; set extruder to absolute mode
G28 ; home all axes
; The lines below are to prep the nozzle
before a print begins
G1 Z0.2 F1200 ; raise nozzle 0.2mm
G92 E0 ; reset extrusion distance
G1 Y1 ; move Y-Axis (bed) 1mm to prep for
purge
G1 X100 E12 F600 ; move X-carriage 100mm
while purging 12mm of filament
G92 E0 ; reset extrusion distance
...
```

# Key Slicer Settings: Material

- Filament choice primarily determines these specific slicer settings:
  - printing temperature
  - bed temperature (if necessary)
  - part cooling (if necessary)
- Filaments vary in terms of printability (by both type and manufacturer) — some are a real headache!
- Filament type has a huge impact on the mechanical properties of the final printed piece

*I have found the default profiles in Cura to be good starting points in terms of settings. When it comes to using different types/brands of filament, I typically only ever need to adjust the print temperature up or down in 5 °C increments as appropriate*



For a more encyclopedic treatment of popular (and exotic) 3D printing filament types:  
<https://all3dp.com/1/3d-printer-filament-types-3d-printing-3d-filament/>



## PLA — poly lactic acid

- printing temp: 190-220 °C
- little to no warping
- bed heating not required, but works well with bed temp: 20-60 °C
- easy to print
- many exotic varieties available
- decent solvent resistance

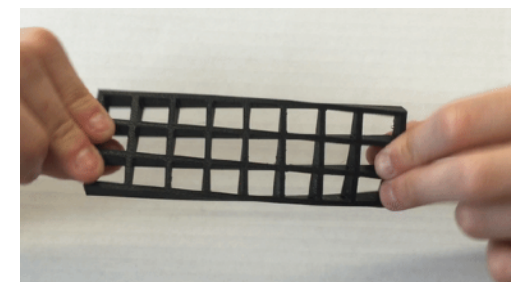
## ABS — acrylonitrile butadiene styrene

- printing temp: 210-250 °C
- prone to warping, printing enclosure essentially required
- bed temp: 80-110 °C
- higher strength and durability than PLA
- soluble in esters, ketones



## PETG — polyethylene terephthalate, glycol modified

- printing temp: 220-250 °C
- little warping
- bed temp: 50-75 °C
- higher strength and durability than PLA
- decent solvent resistance



## TPU — thermoplastic polyurethane

- printing temp: 210-230 °C
- little to no warping
- direct-drive extruder highly recommended
- bed heating not required, but works well with bed temp: 30-60 °C
- flexible! comes in a variety of hardness ratings



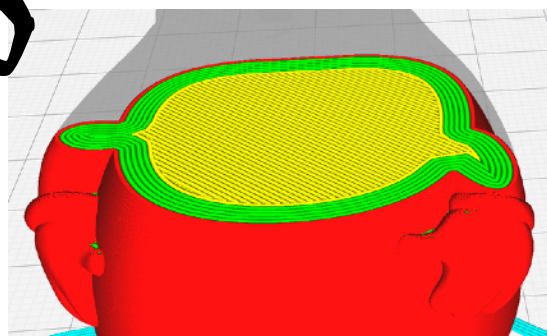
# Key Slicer Settings: Quality and Infill

## Quality:

- Print quality mostly determined by layer height
- Smaller layer heights take longer to print: tradeoff between quality and print time
- Rule of thumb: min/max layer height determined relative to nozzle size
  - min layer height = 20% of nozzle size (0.08 mm for 0.4 mm nozzle)  
below this, increased risk of heat creep/clogging
  - max layer height = 75% of nozzle size (0.3 mm for 0.4 mm nozzle)  
above this, increased risk of under extrusion

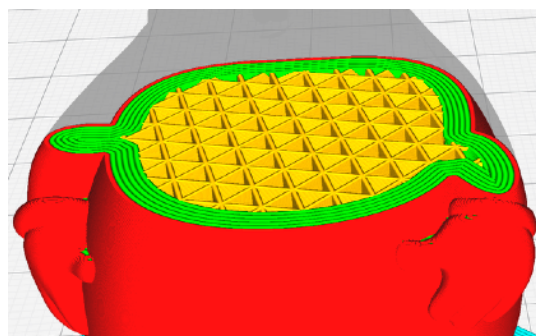
Depending on the printer, optimal layer heights will usually be multiples of some “magic number” (determined by step angle of Z-stepper and thread pitch of lead screw, i.e. the minimum increment in which the printer can move the Z axis)

**100% infill**



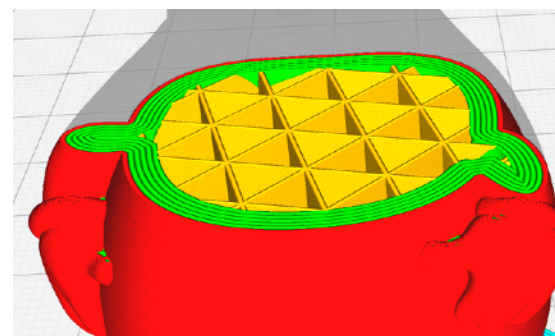
~5.5 hr

**50% infill**



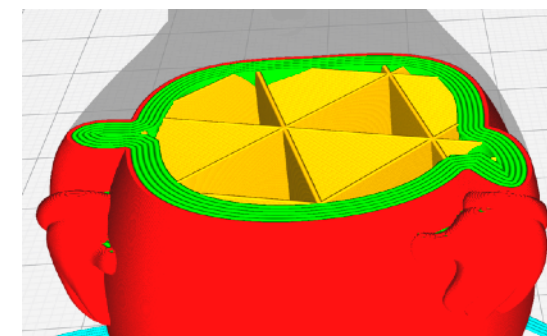
~2.5 hr

**25% infill**



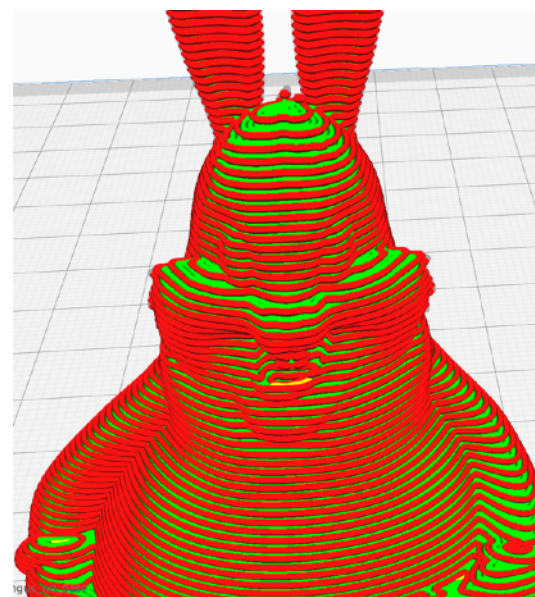
~2 hr

**12.5% infill**



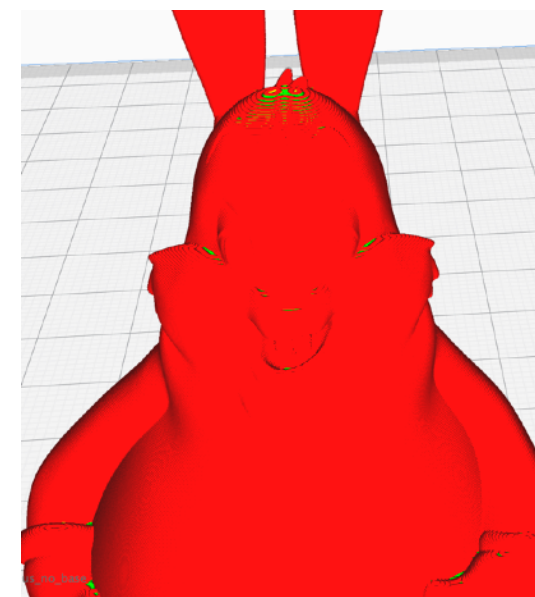
~1.75 hr

**0.5 mm layer height**



~0.5 hr

**0.1 mm layer height**



~2.5 hr

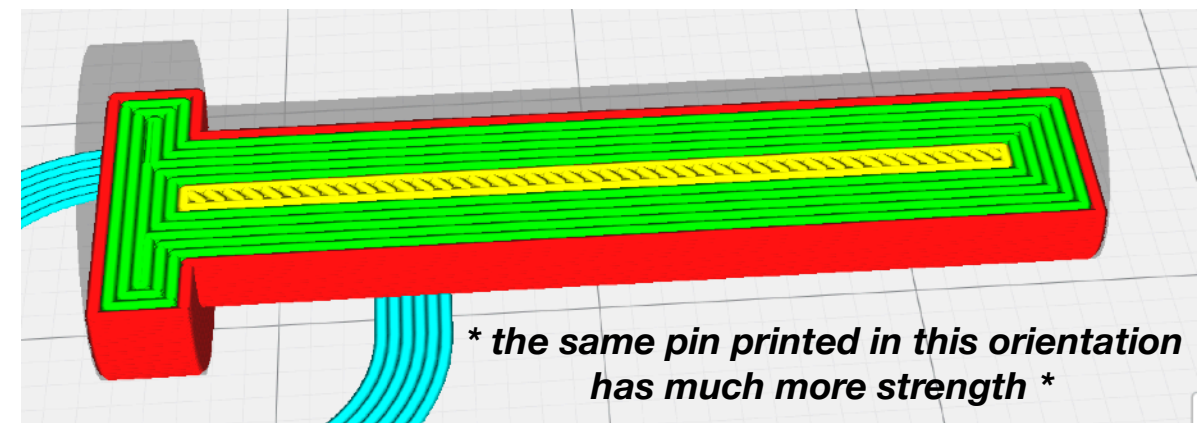
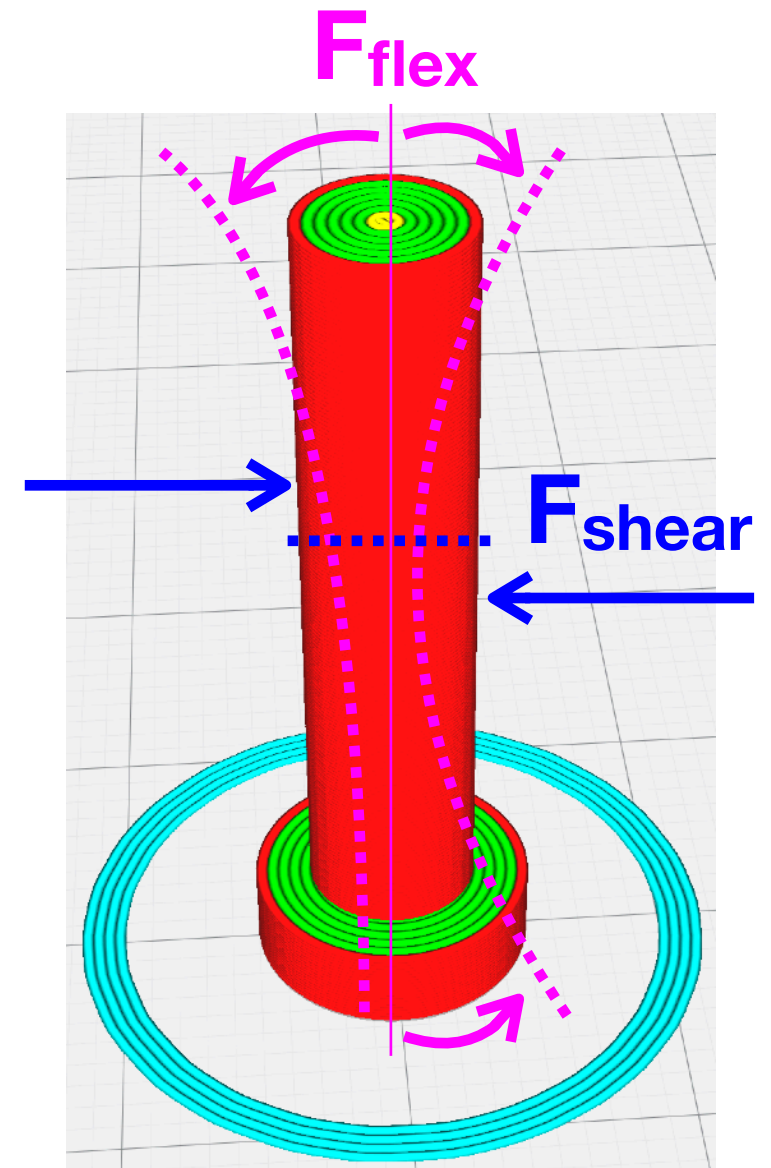
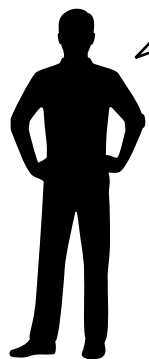
## Infill:

- most parts do not need to be completely solid to be functional
- using an infill setting of <100% saves time and filament
- Cura has multiple infill patterns to choose from (triangles rectilinear, gyroid, etc.)
- Denser infill takes longer to print: tradeoff between infill density and print time

# Key Slicer Settings: Printing Orientation

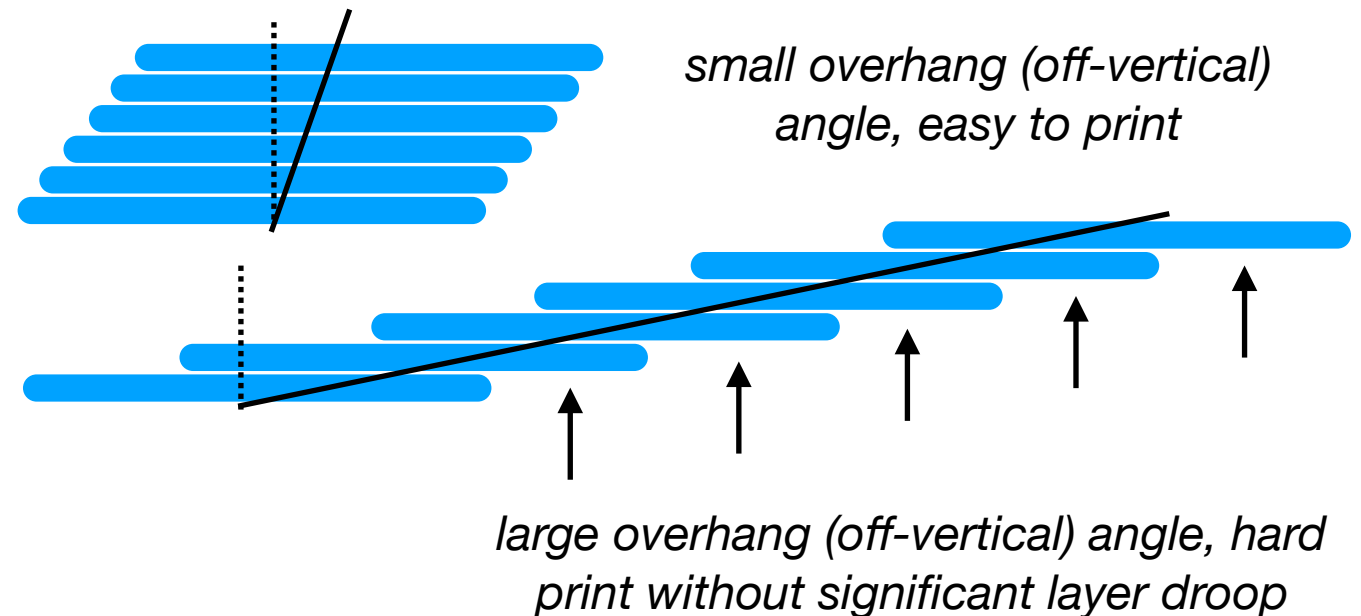
- For structural parts, it is important to consider the orientation that you are printing in
- Generally, FDM printed parts are not very resistant toward shear force parallel to layer lines, or flexing perpendicular to layer lines

For complex parts that need to handle different kinds of stresses, sometimes it helps to split the part into multiple pieces, print each in optimal orientation, then join them later



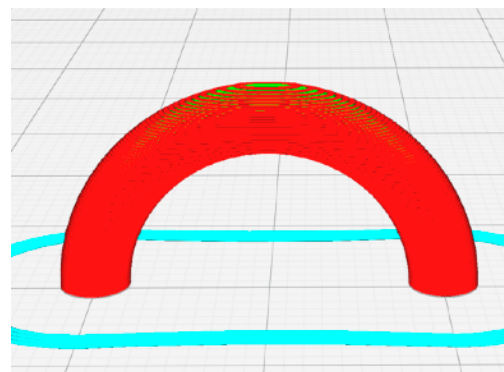
# Key Slicer Settings: Supports

- FDM works by “stacking” layers on top of one another, overhangs are features without lower layers to “stack” on top of
- Depending on the printer, different degrees of overhang are possible to print successfully
- Most slicer software has the option to add support structures for overhanging features that are easy to remove when the print is complete
- any feature with an overhang (off-vertical) angle above a specified threshold automatically gets support structures added beneath it
- The appropriate support angle threshold will vary depending on the capabilities of your specific printer and the material you are using

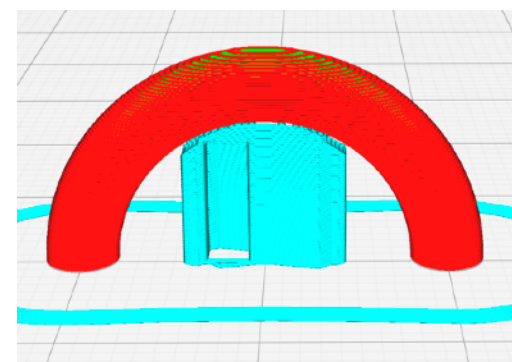


When printing with PLA, better part cooling can enhance performance with unsupported overhangs

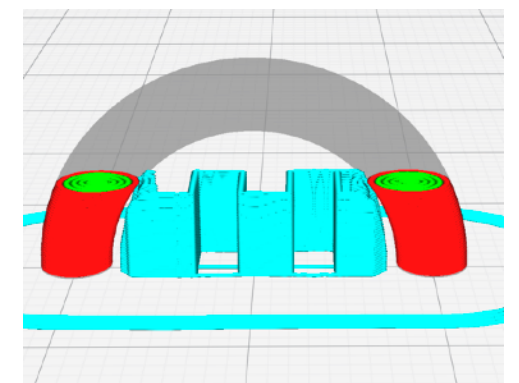
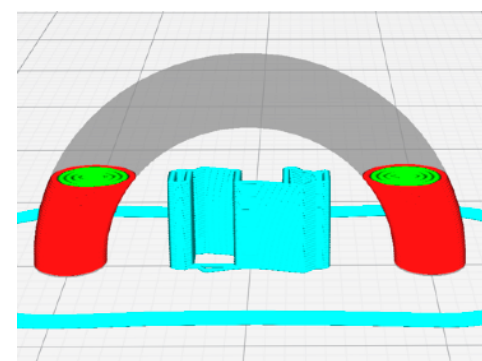
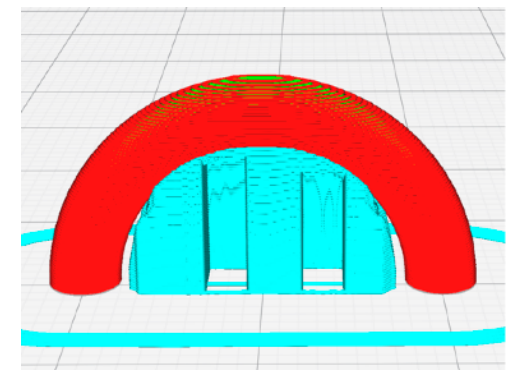
no support



overhang threshold: 60°



overhang threshold: 30°



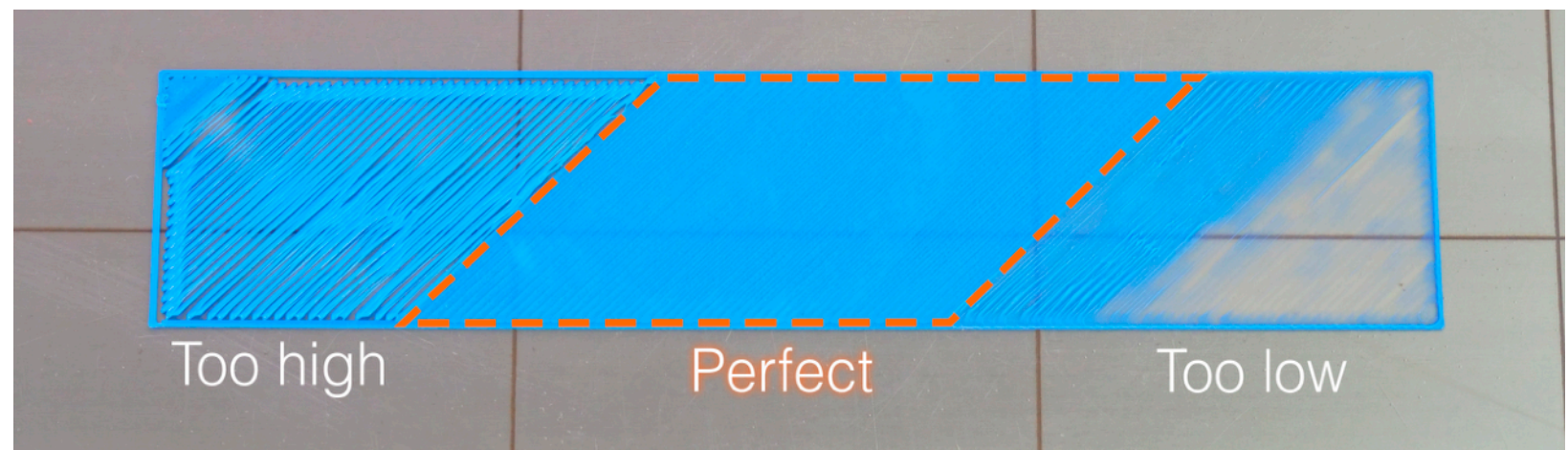


# Bed Adhesion and the Critical First Layer

- Proper bed adhesion is critical to the success of a print
- Bed adhesion is influenced by a number of factors:
  - build plate material (glass, PEI, painter's tape, BuildTak, *etc.*)
  - bed plate adhesion modifiers (hairspray, glue stick, sugar water)
  - bed heating (depends on filament type)
  - print speed (slower print speed usually helps, Cura automatically slows the first layer down by default)
  - **bed leveling** (extremely important! Depending on the printer this is done manually or automatically using IR/touch/induction sensors)

The best conditions for getting good print adhesion are highly dependent upon your specific printer and the type of filament you are trying to print. I find the best place for helpful and curated practical info are online communities for specific models of printers (subreddits/facebook groups)

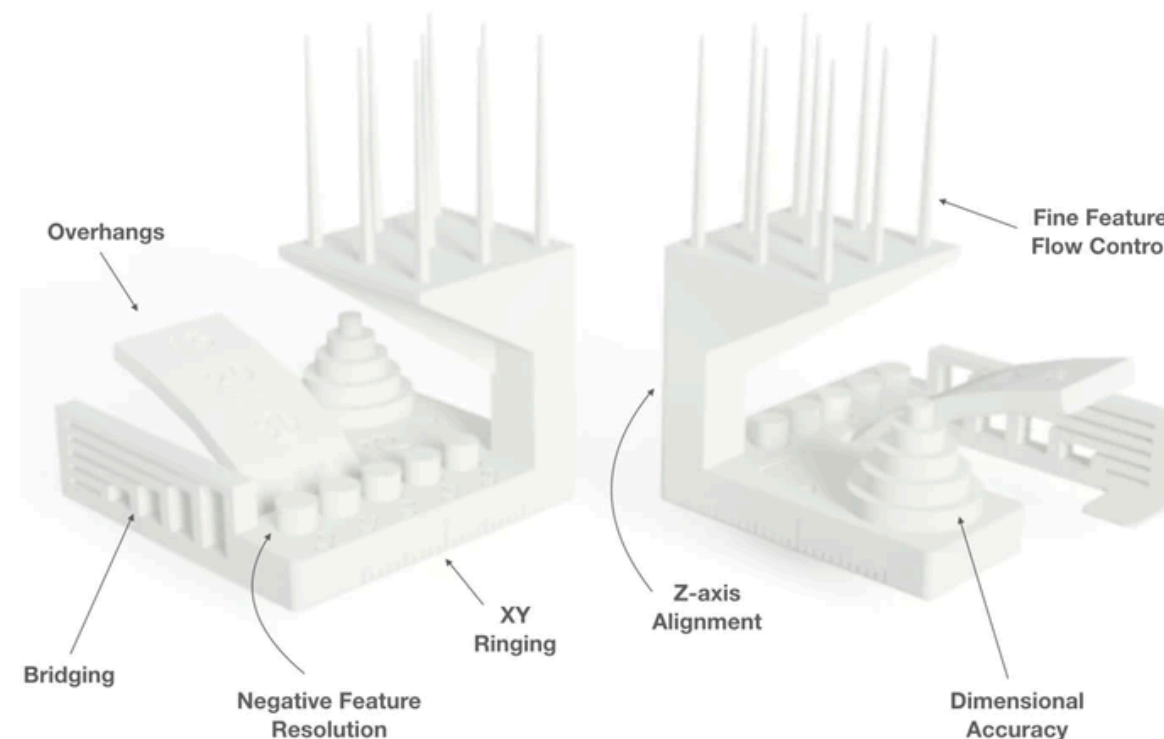
*first layer print quality can indicate if bed is the proper distance from the nozzle*



<https://help.prusa3d.com/article/ib8joi6hm5-first-layer-does-not-stick>

# Benchmarking 3D Printer Performance — All in One Benchmark

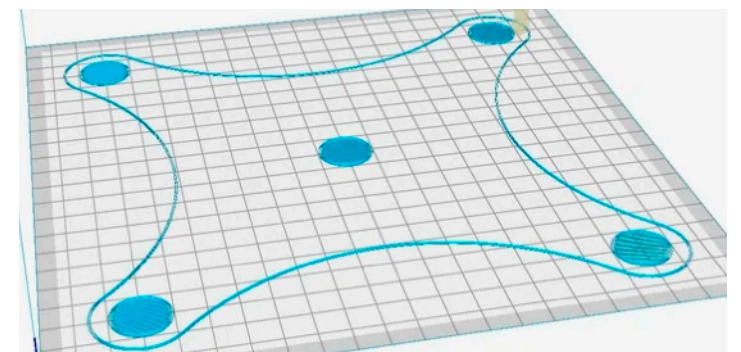
- Specific aspects of 3D printer performance can be benchmarked using specially designed test objects
- Defects in these test objects can be diagnostic of specific issues — great for troubleshooting or just squeezing the maximum performance out of your machine
- There are tons of objects out there for benchmarking, I like this specific one from Autodesk and Kickstarter (below) because it tests several printer performance attributes and includes a detailed, (mostly) objective scoring protocol so you can not only diagnose issues with your printer but also get an idea for the level of performance that you should expect based on the results of others with your model of printer



<https://www.kickstarter.com/blog/toward-better-3d-printers-a-new-test-from-autodesk-and-kickstart>  
<https://github.com/kickstarter/kickstarter-autodesk-3d/tree/master/FDM-protocol>

# Benchmarking 3D Printer Performance — Other Benchmarks

- **XYZ calibration cube** — Quick check for dimensional accuracy, X or Y axis ringing, Z banding (<https://www.thingiverse.com/thing:1278865>)
- **Temperature Tower** — Useful for determining the optimal printing temperature for a given type/brand/color filament (<https://www.thingiverse.com/thing:2493504>)
- **Bed Level Test** — aids in manual bed leveling, prints a single layer with circles at each corner of the print bed (<https://www.thingiverse.com/thing:2987803>)



# Health and Safety

- **Fire Hazard** — 3D printers generate a lot of heat and move around a lot. always check all electrical connections for proper contact and signs of overheating. check wires for signs of wear in high-motion areas. keep flammable materials away from the printer and keep power supply/electronics free of obstruction and dust
- **Toxic Fumes & Ultra-fine Particles (UFPs)**— Depending on the filament being used, toxic fumes may be emitted during printing. ABS is especially known for its volatile organic compound (VOC) emissions during printing, PLA has negligible VOC emissions. All 3D printing filaments emit some level of UFPs while printing. Different types of filament and printing conditions influence the level of emission. ABS is one of the worse UFP emitters, PLA emits far less. Proper ventilation is important to dilute and eliminate these emissions during printing. Enclosures can mitigate these risks as long as they are properly enclosed and ventilated.
- **Moving Parts** — Most 3D printers have exposed moving parts which can be a hazard if you come in contact with them while the printer is in motion. Be cautious around a 3D printer during operation, do not wear baggy clothing/jewelry while operating one, stop the printer before attempting to touch/adjust it in any way. Enclosures mitigate these risks

***Bottom Line: If at all possible, use some type of enclosure because it mitigates risks from moving parts and printer emissions***

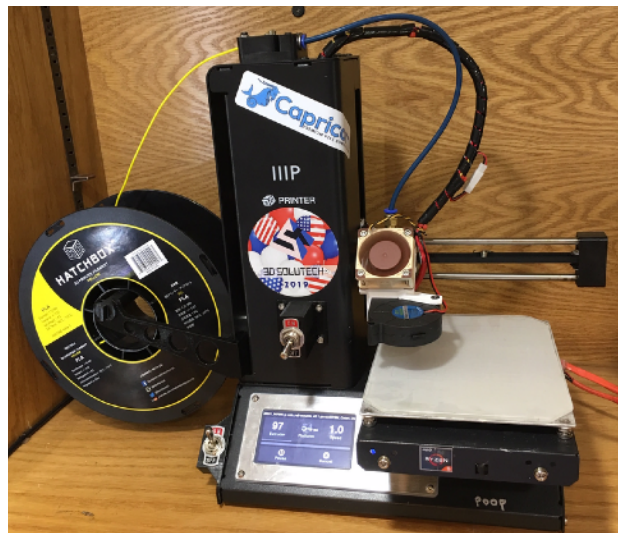
If an enclosure is not possible, keep your printer in a well-ventilated area (at least 4 air changes/hr is ideal) and stick with PLA

[https://www.ehs.washington.edu/system/files/resources/Shops-3D\\_Printers\\_Focus\\_Sheet.pdf](https://www.ehs.washington.edu/system/files/resources/Shops-3D_Printers_Focus_Sheet.pdf)

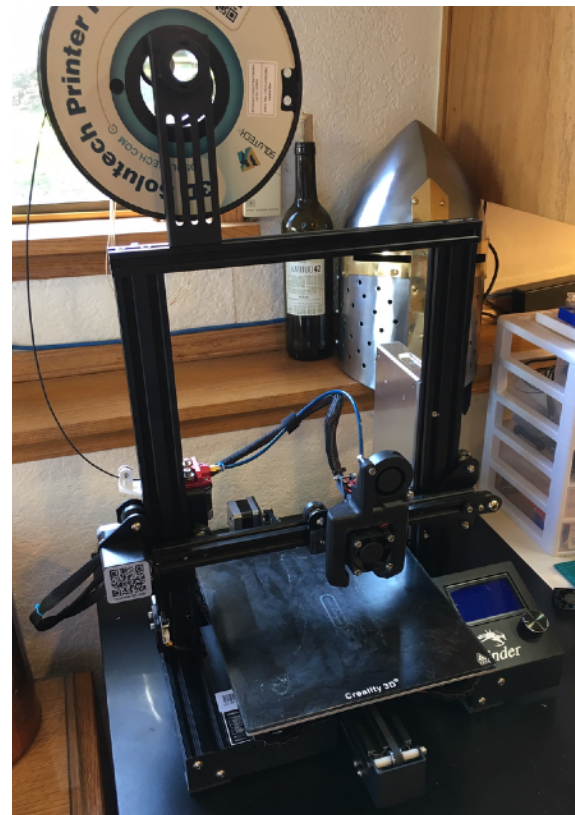




# Thanks!



My printers:  
(top) Monoprice Select Mini v2  
(right) Creality Ender 3 Pro



Printed a case for a custom mini mechanical keyboard. Case printed in 6 pieces and glued together. Painted then handwired.



D&D terrain and miniatures



Clip thing for my kitty's window bed

