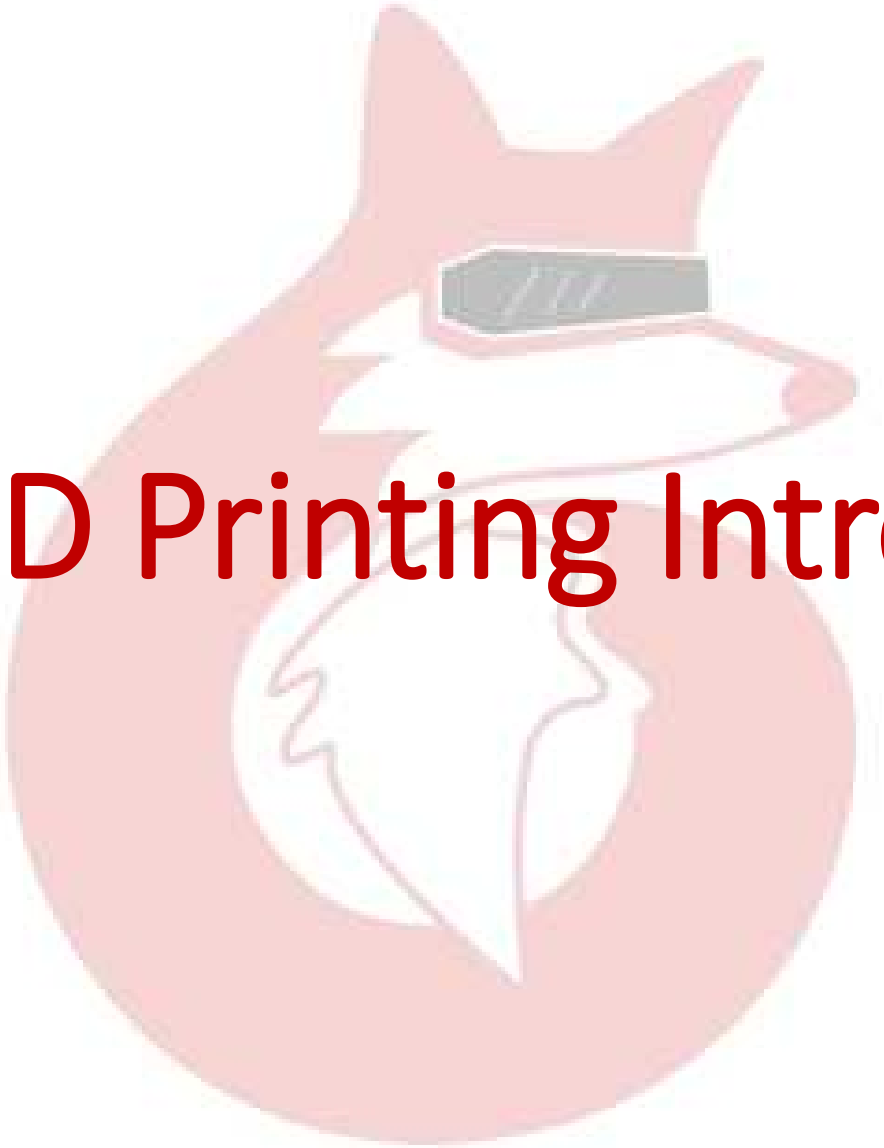


3D Printing Intro

Why Learn to 3D Print?

- Very affordable way to create parts and prototypes
- Can create items not possible via traditional methods
- Easy to learn
- Fast turnaround
- It’s fun!

The Printing Process

- Virtually all types of 3D printers subdivide the part into layers and print layers one on top of another, layer by layer until complete
  - Fused Deposition Modeling (FDM)
  - Resin Printing
  - Selective Laser Sintering (e.g., for metal printing)

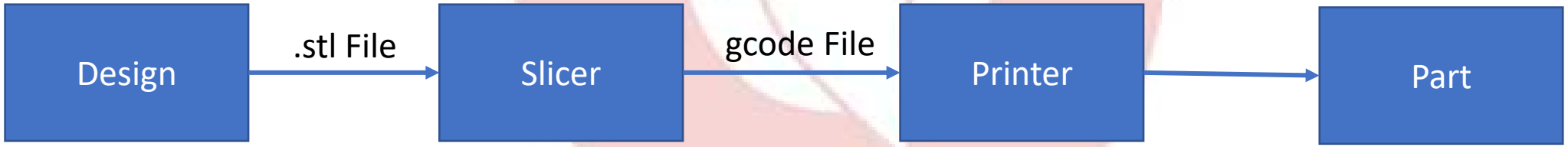
  - Example of FDM Printing (At 0.33 in video)
    - https://youtu.be/l1UHEkJKnDc

Three Steps to 3D Printing a Part

1. Design (or copy) the part to be printed

2. Slice the part into very thin layers

3. Print the layers one at a time, building the part up



fox.build

Part Design – Design Programs

- Part design is typically done in a Computer Aided Design (CAD) program, such as:
  - OnShape – Free, requires registration
  - Fusion 360 – Free, requires registration
  - FreeCAD – Free, Open-source
  - OpenSCAD – Free, Open-source, script based
  - Solidworks – $99/year subscription for Makers
  - SketchUp – Free, browser based, requires registration
  - Tinkercad - – Free, requires registration

Part Design – Output File Types (Slicer Input)

- Traditionally a .stl (stereolithography) file
- Newer file formats are gaining popularity
  - .obj – 2nd most popular type, retains fidelity better than .stl, supports multi-color
  - .amf (Additive Manufacturing) – Higher fidelity and more compact than .stl, but similar
  - .3mf – Supported by many large players in the industry, similar to .amf

Part Design – Copying a Design

- Design may be in 3D solid format or .stl (or .obj, .amf, .3mf)
  - .stl can be input directly into the slicer
  - 3D Solid format is often CAD program specific
- There are many online sites that allow sharing of designs
  - Thingiverse.com – Probably the most well known, has over 2 million designs
  - Printables.com – Supported by Prusa, a printer manufacturer
  - Pinshape.com – Over 70,000 Makers and Designers registered
  - 3Dexport.com - Many designs have a fee, but there is a free section
  - Cults3d.com – Many designs have a fee
  - MyMiniFactory.com – Many designs have a fee

Slicing

- You have a design, now what?
- The design must be “Sliced” into layers that the printer can print
- There are a number of slicers available, but the 2 most popular are:
  - Cura – Developed by Ultimaker, a printer manufacturer
  - Prusa Slicer – Developed by Prusa, another printer manufacturer.
- Some printer manufacturers have slicers specific to their printers, but in general a slicer can slice for any printer. Most are free.
- The slicer outputs “G-code”, which is a text file that tells the printer how to move, temperature settings, when to extrude filament, etc.

Slicer Parameters

- In addition to the model file the slicer has a lot (potentially hundreds) of input parameters the user can adjust.
- The following parameters MUST be considered for every slicing job
  - Material type: This affects the required temperature and sometimes speed
  - Layer height: Determines resolution and print time
  - Support: Is support needed, and if so, what type of support
  - Bed adhesion: Skirt, brim, raft, or none
- For other parameters, start with the defaults and tweak if necessary

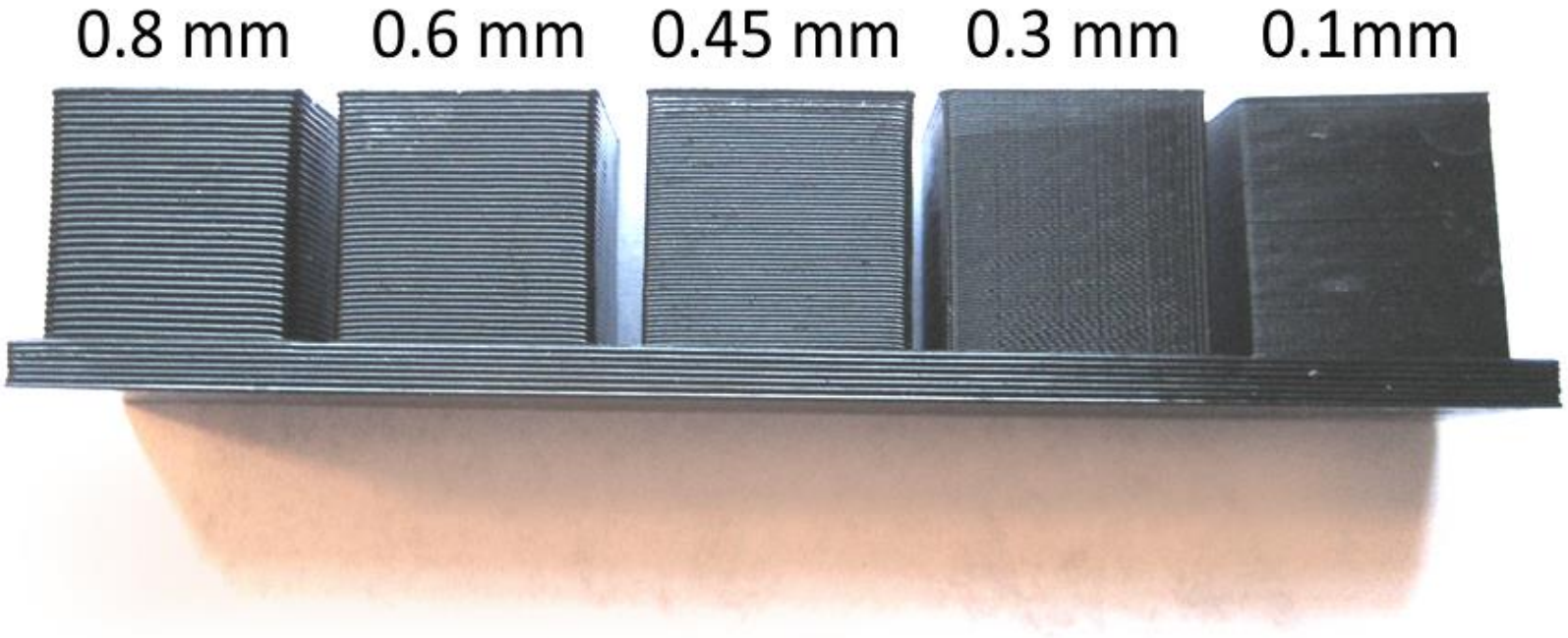
Material Types

- Some of the more common materials and printing temperatures:

Material	Extruder Temp (°C)	Bed Temp (°C)	Notes
PLA	190-220	-	Easiest to print, biodegradable, not good with heat
ABS	220-250	95-110	Best printed in an enclosure, good with heat, same as Legos
TPE/TPU	225-245	45-60	Flexible, challenging to print
HIPS	230-245	100-115	Dissolvable, good for support
PETG	230-250	75-90	Water resistant, easy to print, can be food-safe

Layer Height

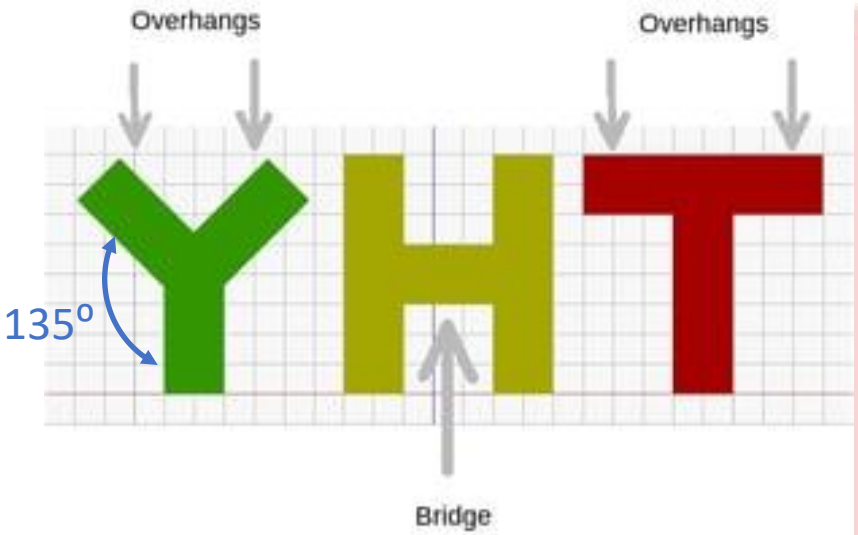
- Cannot be greater than nozzle diameter (0.4mm is by far the most common nozzle diameter)
- Higher layer height => faster printing, less resolution

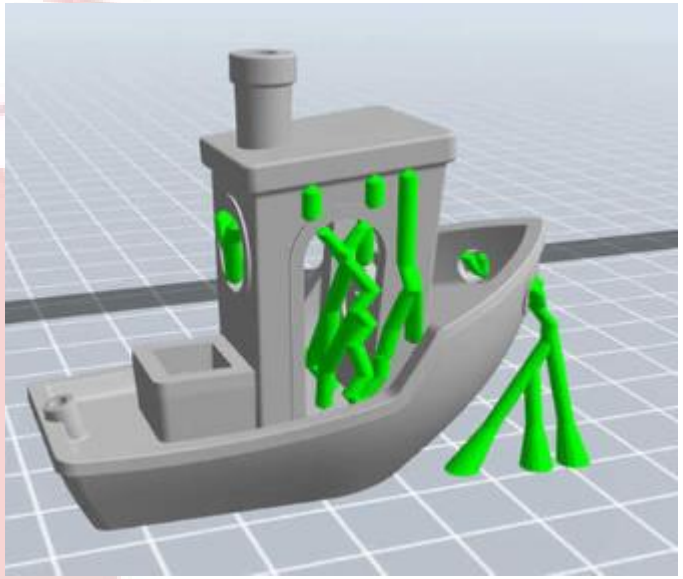


fox.build

Support

- Support is needed for overhangs of more than 45 degrees and bridges longer than about 5mm





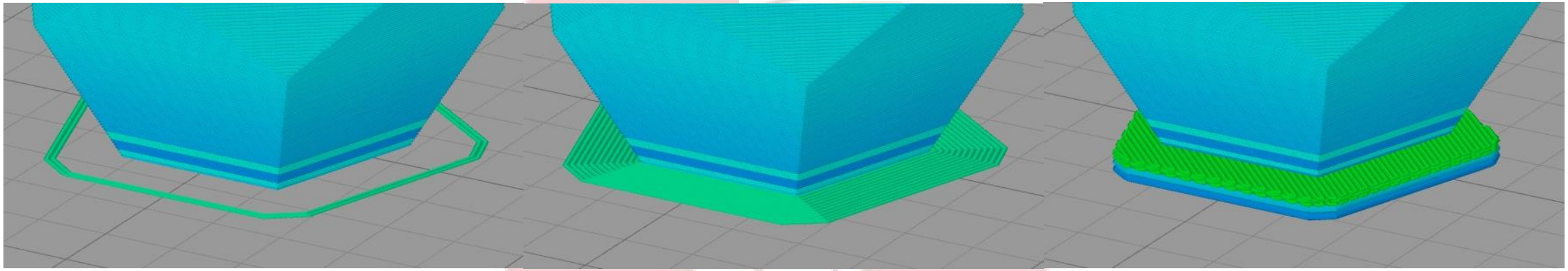
Tree-like support

- Tree-like support can be tricky, linear works pretty well

fox.build

Bed Adhesion

- The reason for heating the bed is to improve bed adhesion and reduce warpage
- Parts with a small base may need special attention to bed adhesion
  - The 3 main types of bed adhesion are skirt, brim, and raft



Skirt. Primes extruder, doesn't really help with adhesion.

Brim, like on a hat. Helpful for parts with a small footprint.

Raft. Essentially support under the entire part. Helps stabilize and reduce warping, especially with ABS.

fox.build

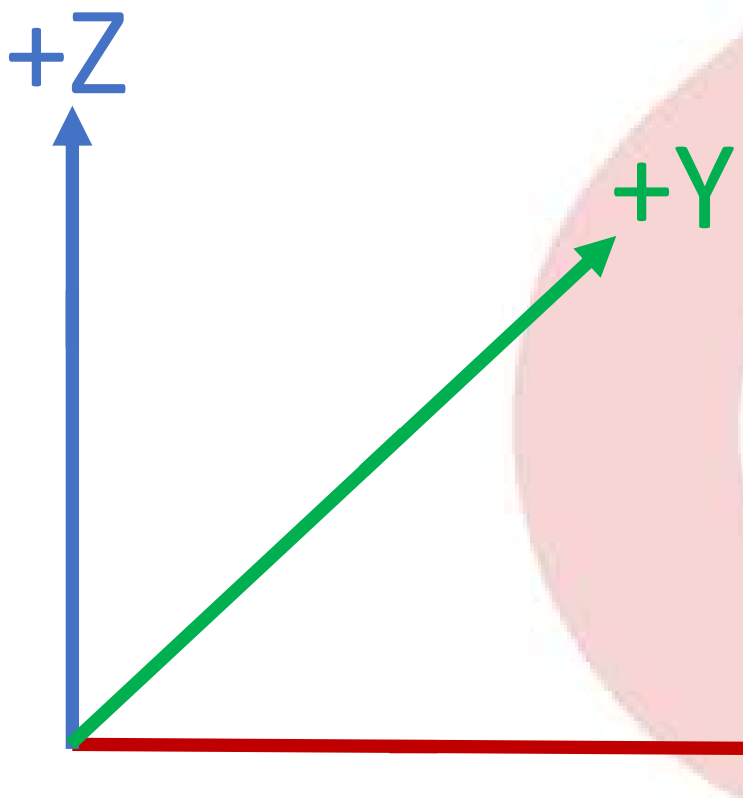
Bed Adhesion - Continued

- In addition to the adhesion options in the slicer some printers require additional treatment to the bed itself
  - School glue
  - Blue painter's tape
  - Hair spray

- PEI (Polyetherimide) beds typically do not need treatment
  - The Fox.Build Prusa and Dremel have PEI beds
  - The Maker Select and the Tarantuala require school glue

Printer Coordinate System

- Standard Cartesian Coordinate System, should match the slicer's view



- X and Y define the plane of the bed
- Z is the height above the bed

Printing

- There are three typical ways to send your sliced file (G-code) to the printer
  - Using an SD card in the printer's card slot
    - Select "Print from SD", navigate to the file, and select it
  - Via a USB connection to the printer from the PC
    - Use the slicer software to send the file to the printer
  - Using OctoPi, a Raspberry Pi app that provides a browser interface to the printer
    - Connect to the printer using the printer's IP address in the your browser
    - Drag and drop the sliced file to the OctoPi list of files
    - Select the print icon associated with the file
    - Print can be monitored from anywhere on the network

Potential Errors

- Thermal Runaway
  - Indicates the firmware is calling for more heat but the printer isn't responding
  - Could be due to a loose/bad thermistor connection
  - If on the 2nd layer might be due to the cooling fan over-cooling. Typically, this is because the insulating boot has fallen off the heater block
- Layer Shift
  - Most likely caused when something impeded the motion of the printer.
    - Obstacles around the printer
    - Sticky bearings
    - A "blob" of plastic oozed onto the part and hardened, preventing movement
- Not Sticking to the Bed
  - Dirty bed
  - Z-height not correctly calibrated (i.e., 1st layer is too high off the bed)

Questions and Answers and Demos

- CAD Output
- Slicer
- Printing

fox.build

References

- All3dp.com
- 3Dprint.com
- RepRap.org
- simplify3d.com/support/

fox.build