EP1000
3D
Printing

Technology

- A new technology – Additive Manufacturing
- Advantages
  - Can create almost anything
  - Ideal for rapid prototyping (visualisation)
  - Can be cost saving for R&D
- Disadvantages
  - Slow (in terms of production)
  - May not have the material strength
  - Machine can be very expensive

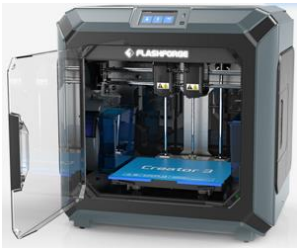
Types of 3D Printing Techniques

- stereolithography (SLA)
- digital light processing (DLP)
- fused deposition modelling (FDM)/fused filament fabrication (FFF)
- ink-jet binder
- polyjet
- cut sheets laminate
- selective laser sintering (SLS), selective laser melting (SLM)
- electron beam melting (EBM)
- gel dispensing printing (GDP)

Additive Technology – 3D printing

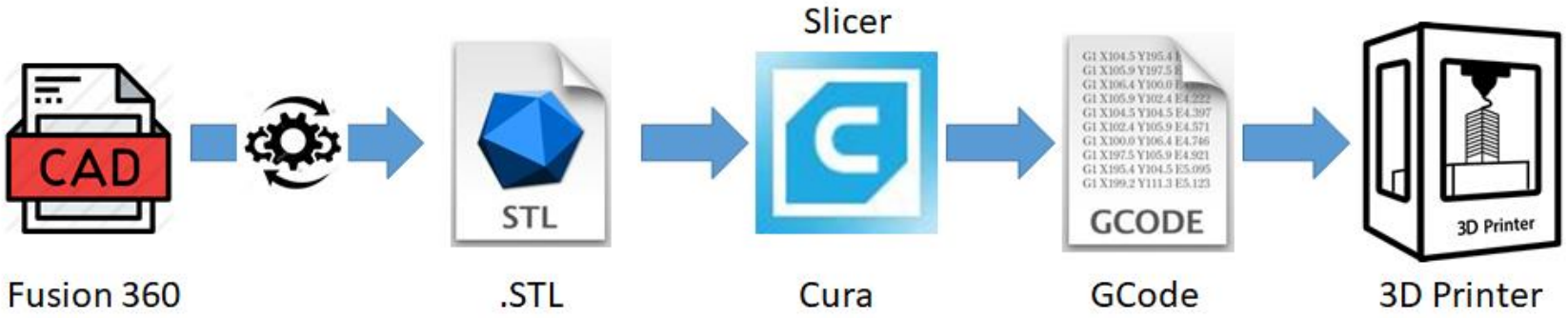
Almost anything can be 3D Printed (If you can model it, it can be printed)

Comparison of 3D Printers at Fablab



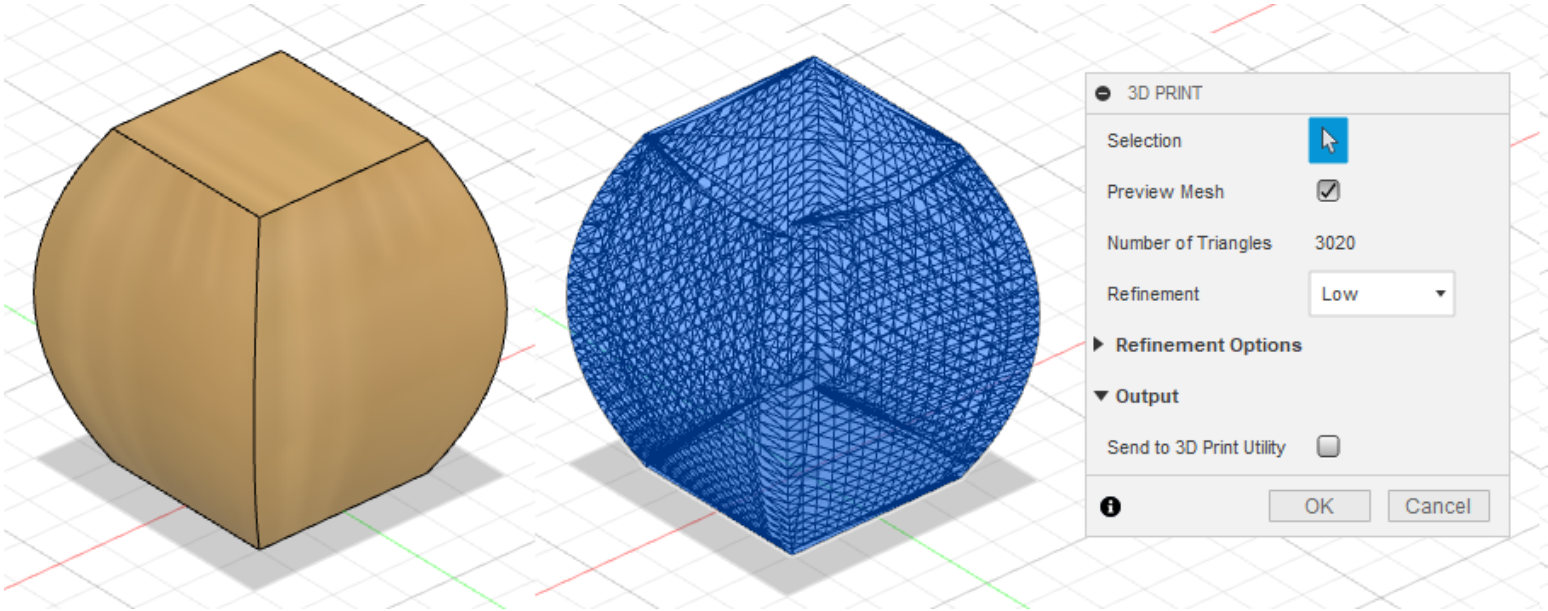
Printer	Ultimaker 2+	Prusa i3 mk3s	Flashforge Creator 3	Creality Ender 3 Pro v2
Size (mm)	210x210x205	250x210x210	300x250x200	220x220x250
Filament Dia	2.85mm	1.75mm	1.75mm	1.75mm
Type	FDM	FDM	FDM	FDM
Speciality	Old, reliable, fast	Reliable, precise,	Dual extruder, reliable, fast	General purpose, cheap

3D Workflow



1. Create a 3D Model of the object using CAD
2. Extract the 3D Model representation (.STL, .OBJ, .3MF)
3. Use a slicer (e.g. CURA) to slice the model into layers.
4. Extract each layer into a machine readable file (.GCODE)
5. Send to the 3D Printer for fabrication

3D Model extraction



- Fusion 360 > Tools > Make > 3D Print
- Refinement: Medium
- Output to File (.STL) – Standard Tessellation Language (other .OBJ, 3MF)

Slicing Software – CURA

- Ultimaker CURA
- A program that converts digital 3D models into printing instructions for a given 3D printer to build an object.
- Slicer virtually “cuts” 3D object into many horizontal layers
- 3D printer constructs the object by re-creating layers (additive manufacturing)

Best 3D Slicing Software in 2021

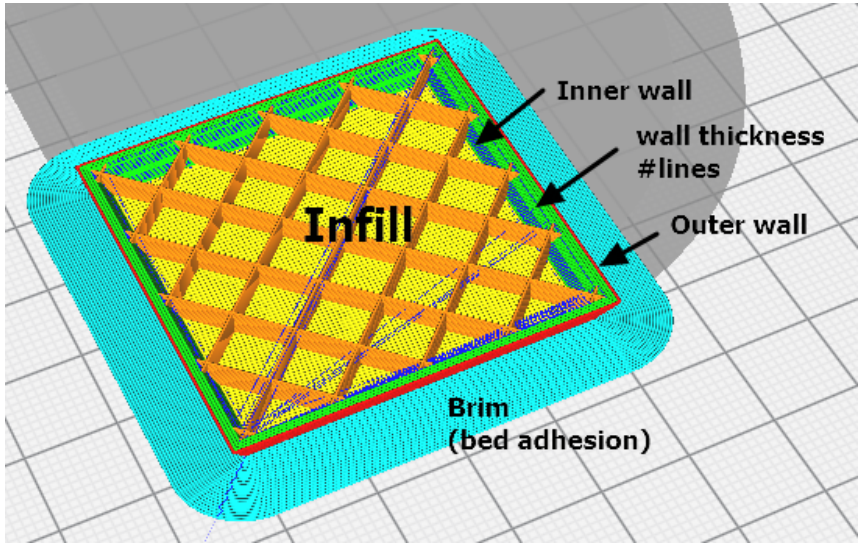
Slicer Graphic Area Elements

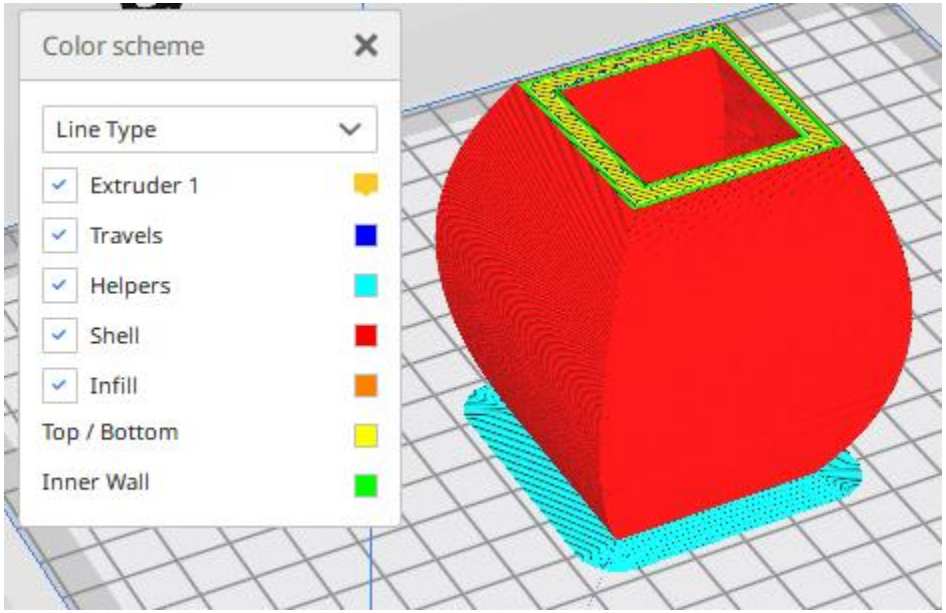
- Offers a 3 Graphical area where you can visualize how the model transforms into a layered representation
  - Printing bed plane
  - Visualization and camera control – allows viewing from different angles
  - Model positioning control – allows repositioning for better prints
  - Layer Preview
    - Shell – external lines that define the outline
    - Outer wall – material line that defines the surface
    - Inner wall – material line that defines the shell thickness
    - Infill – the contents between the outer and inner walls
    - Supports – structures for overhangs
  - Adhesion layers
    - Raft – a plate/layer between bed and the print
    - Brim – lines of material around the first layer, allowing wider contact
    - Skirt – single line for testing of material flow

Slicer Settings

- Layer height – determines how thick each layer is
- Line Width – horizontal thickness of each extruded line
- Shell thickness – thickness of the object wall
- Top, Bottom thickness – quantity of solid lines on the top/bottom of print
- Infill Settings – density of material between inner, outer walls
- Extruder Temperature – temperature to melt the filament, flow
- Print Speed – how fast to move the extruder head
- Cooling Settings – affects the cooling of the filament as it is printed
- Support Settings – type of support, if any, for overhangs
- Adhesion structure settings – how the model will be placed on the bed

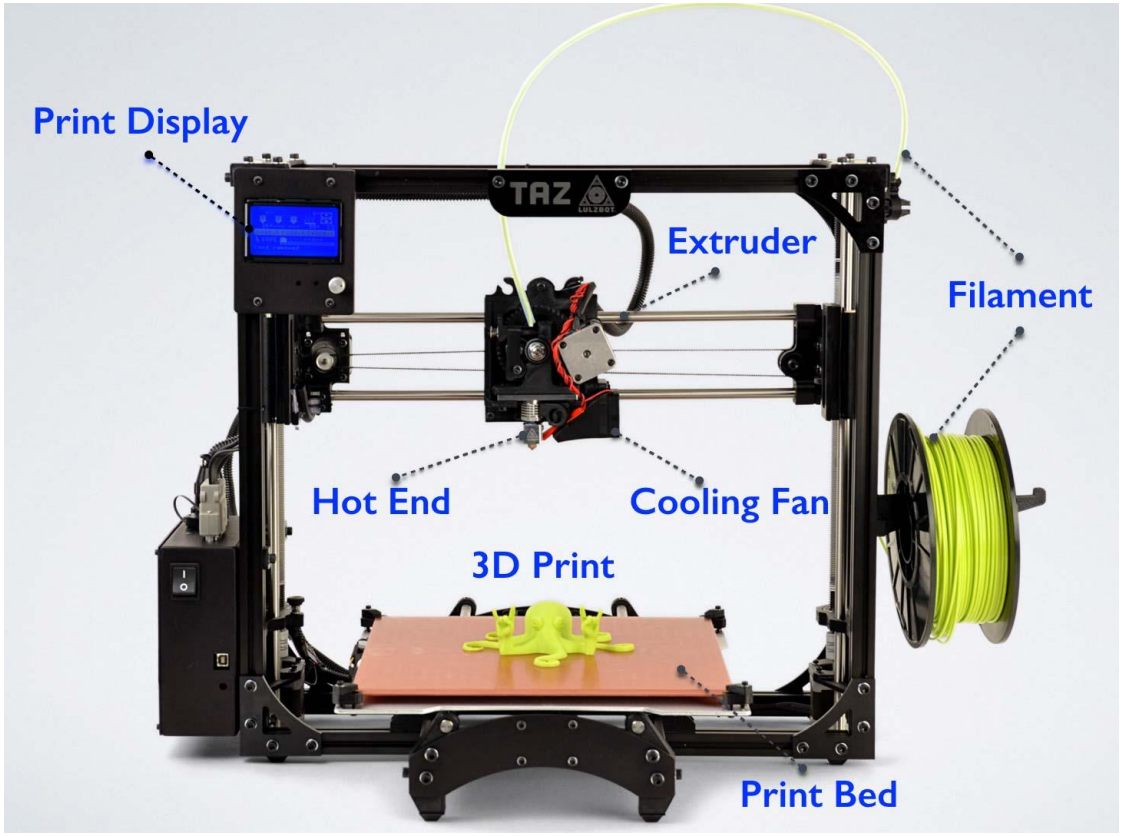
Slicer parameters





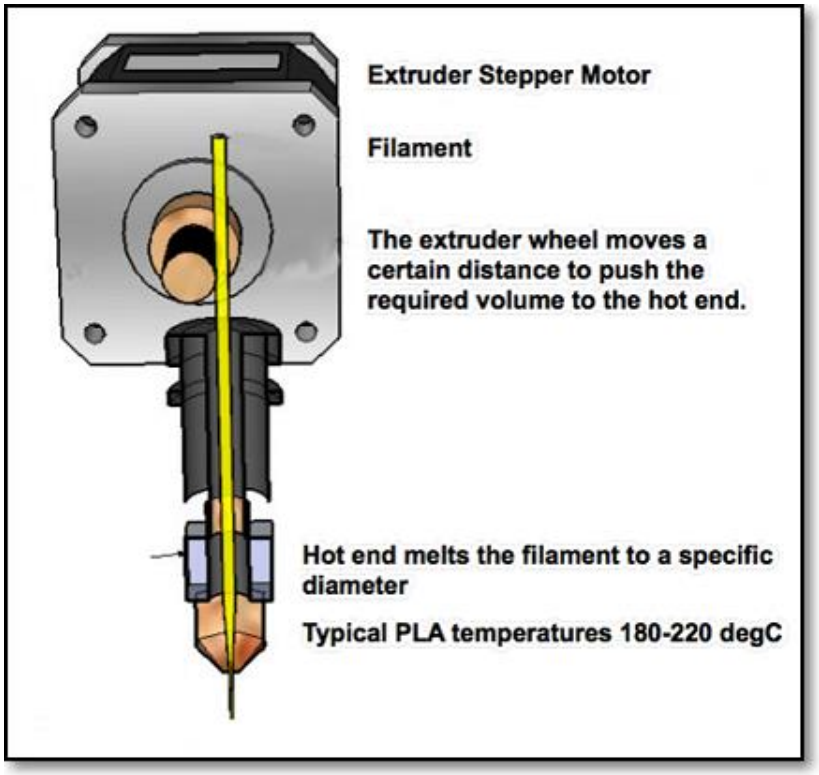
- Slice, then use Preview with Color Schemes to see the different effects of your slicer settings

FDM Printer Elements



- Most FDM printers are similar.
- Filament is fed into the extruder unit
- Extruder unit heats up, melts the filament and forces it out to form a thinner filament string
- Filament string is placed on bed/model
- Cooling is applied using fans

Extruder Module

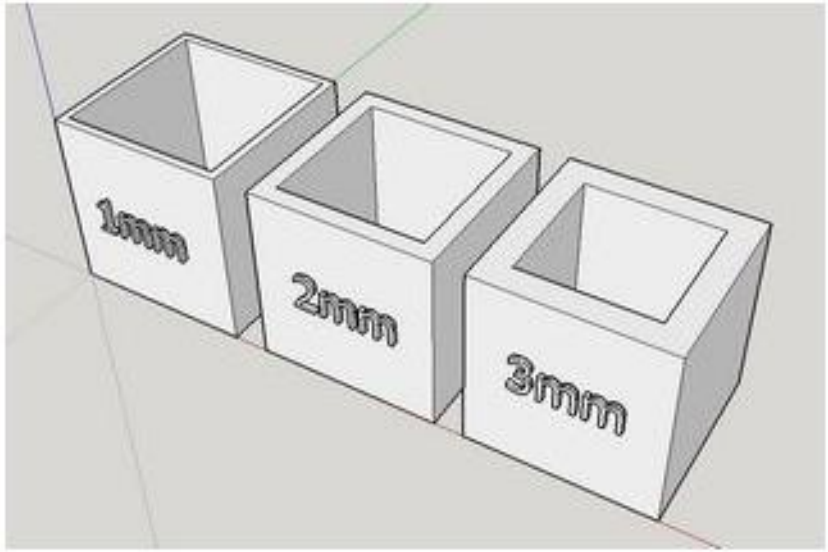


- Heated tube with a nozzle
- Extruder heats up filament and melts it, forcing molten filament through tube and nozzle
- Nozzle controls diameter of extruded material
- Flow rate controlled by stepper motor and the feed of the filament.
- User adjustable 20%~150%
- Some printers can have 2 extruder heads

Layer Height

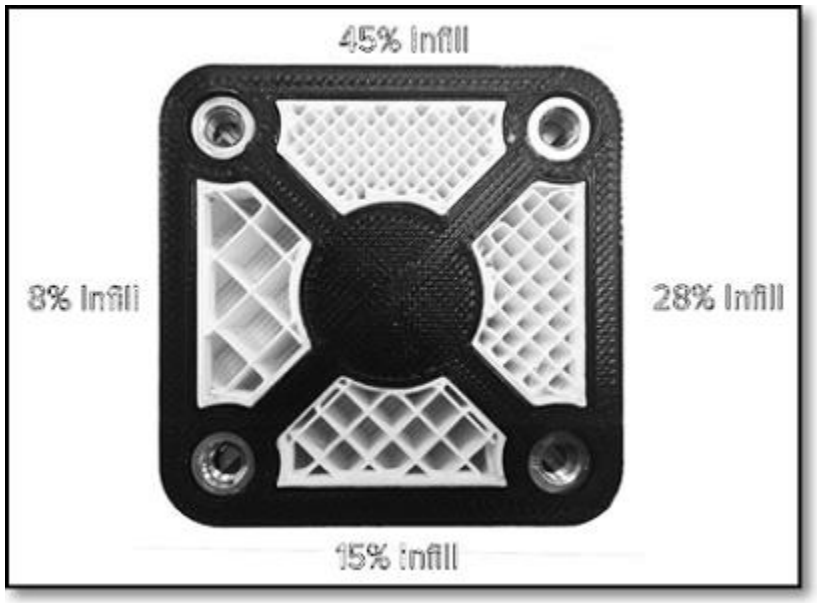
- Height of each printed layer
- Thinner layers give finer prints, however, slower
- Range: 0.1 ~ 0.4 mm (Rule-of-thumb: ½ nozzle diameter)
- Affects resolution, smoothness and time taken to print.

Wall Thickness



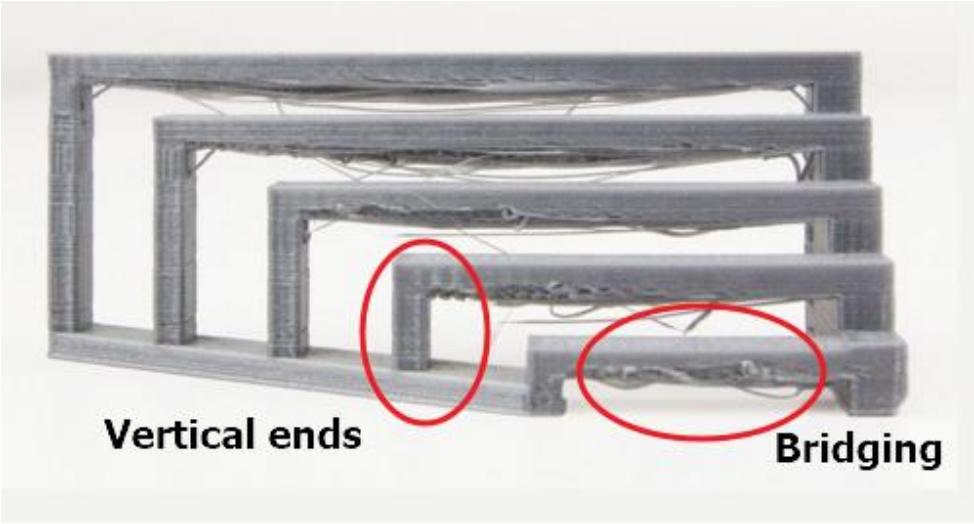
- Number of strands to make up the thickness of the wall.
- E.g wall thickness = 3mm using 0.2 layer
  Inner wall : 0.8 mm
  Outer wall : 0.8 mm
  Wall thickness: 1.4 mm
  (7 passes of the nozzle)
- Affects time and strength of print

Infill Percentages



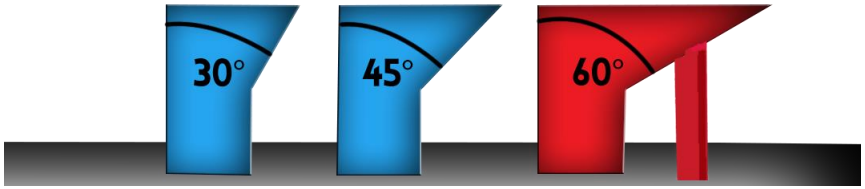
- Controls the amount of fill in the internal cavities.
- Higher percentages take longer time, but higher strength
- Can specify the type of infill pattern structure (lines, triangles)
- Typical infill: 10 ~ 20%

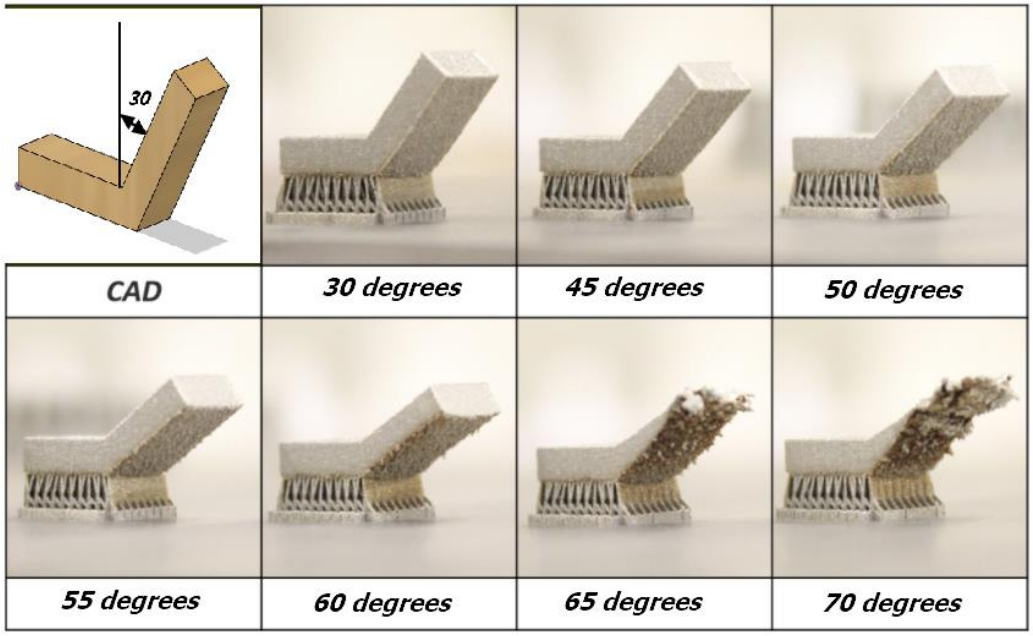
Bridging



- Prints between 2 points that have no connection to each other.
- 3D Printer prints the base layer, then the vertical structures.
- Horizontal structures have no supports other than at the end.
- There is a max distance you can bridge without stringing

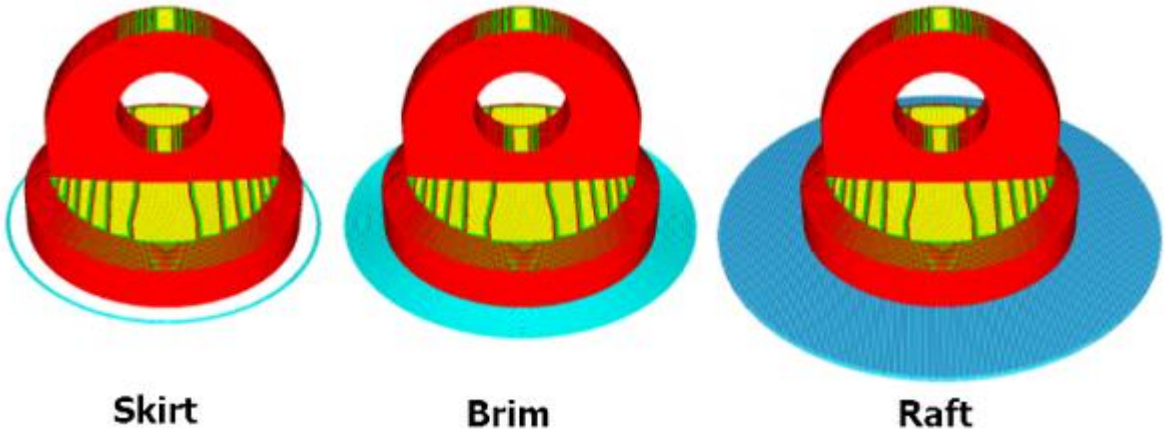
Overhang limits





- 3D printers cannot print on air
- There is a limit where there is insufficient support
- Most printers can handle an overhang of 45^{o}
- Use a test print to determine your overhang angle

Platform adhesion

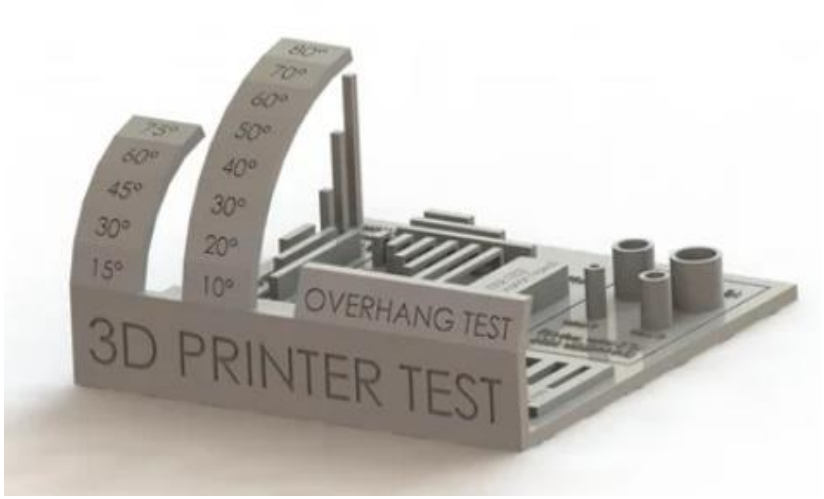


- Printer bed is heated to improve adhesion
- Sometimes print object requires help in adhering to plate
- A Raft is a base of extruded material, the object is built on it.
- Brims allow some extra adhesion to the base of object
- Skirts are used to clean/extrude material to test

Supports

- Supports are used to aid in printing overhangs
- Supports make impossible prints possible
- Supports are removed after printing
- Sometimes, the positioning of print can help in reducing the need for supports

Benchmarking your printer



Ref: All-in-one 3D Printer test

- Testing your printer to determine practical limits
- Not every printer performs the same each time
- Very important that the bed is level.
  (All3DP 3D Printer Bed Levelling)
- Search for printer test models with explanations on the tests:
  (All3DP 2022)

Recommended Cura Settings

Printer	Ultimaker 2+
Material	PLA
Nozzle	0.4mm
Layer Height	0.2mm
Wall Thickness	0.8mm (4x)
Infill	10~20%
Print Speed	50~80 nm/s
Temperature	190~205°C
Supports	Depends
Bed Adhesion	Depends

Tips

- Recommended layer height approx. ½ nozzle size
- Skin layers > 0.8mm
- More infill takes more time
- Try not to use supports, cleaning up is a pain
- Check for bed adhesion, at best use brim
- Print time < 1 hr if possible
- Always be around until the base is built

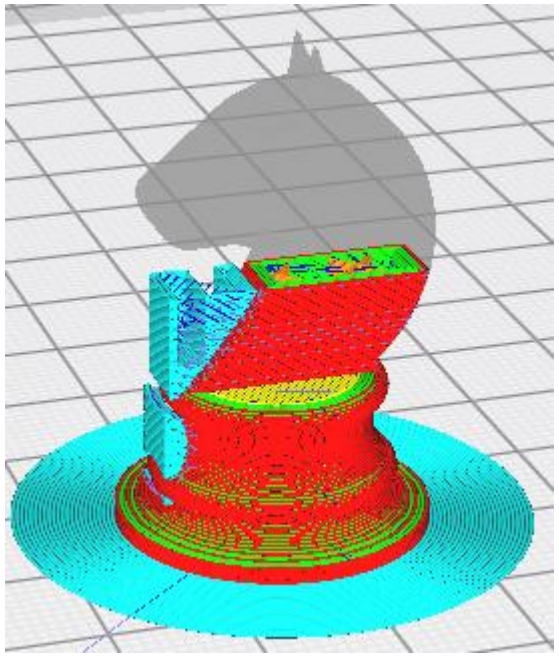
Assignment: Chess Piece

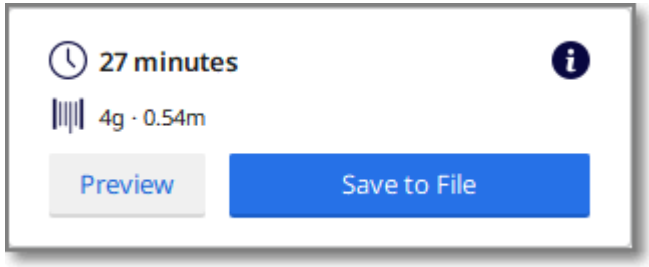
- Convert your chess piece to an STL file
  - Check that the size is no larger than 30x30x50mm
  - Hollow out the chess piece
- Use CURA to slice and prepare your print
  - Layer height 0.2~0.25mm
  - Keep skin/walls to 0.8~1 mm
  - Infill 15% is more than enough
  - Do you require supports? Why?
  - Ensure that there is enough bed adhesion
  - Keep your print down to < 1hr

Assignment: Chess Piece – Knight

- Take note of
  - CURA settings (use screen shots/captures)
  - The time and amount of material used
  - Photos of the 3D object being printed
  - Photos of your unfinished printed piece
  - Photos of your finished piece
- Document your work and findings
- Register for 3D Printing Course in FLMS (under EP1000 Schedule)
- Take the 3D Printer Certification Quiz
  - Found on Brightspace (EP1000 Page)
  - Use the chess piece as your project

Chess Piece: Print





Knight files (zip)

EP1000
3D
Printing
End