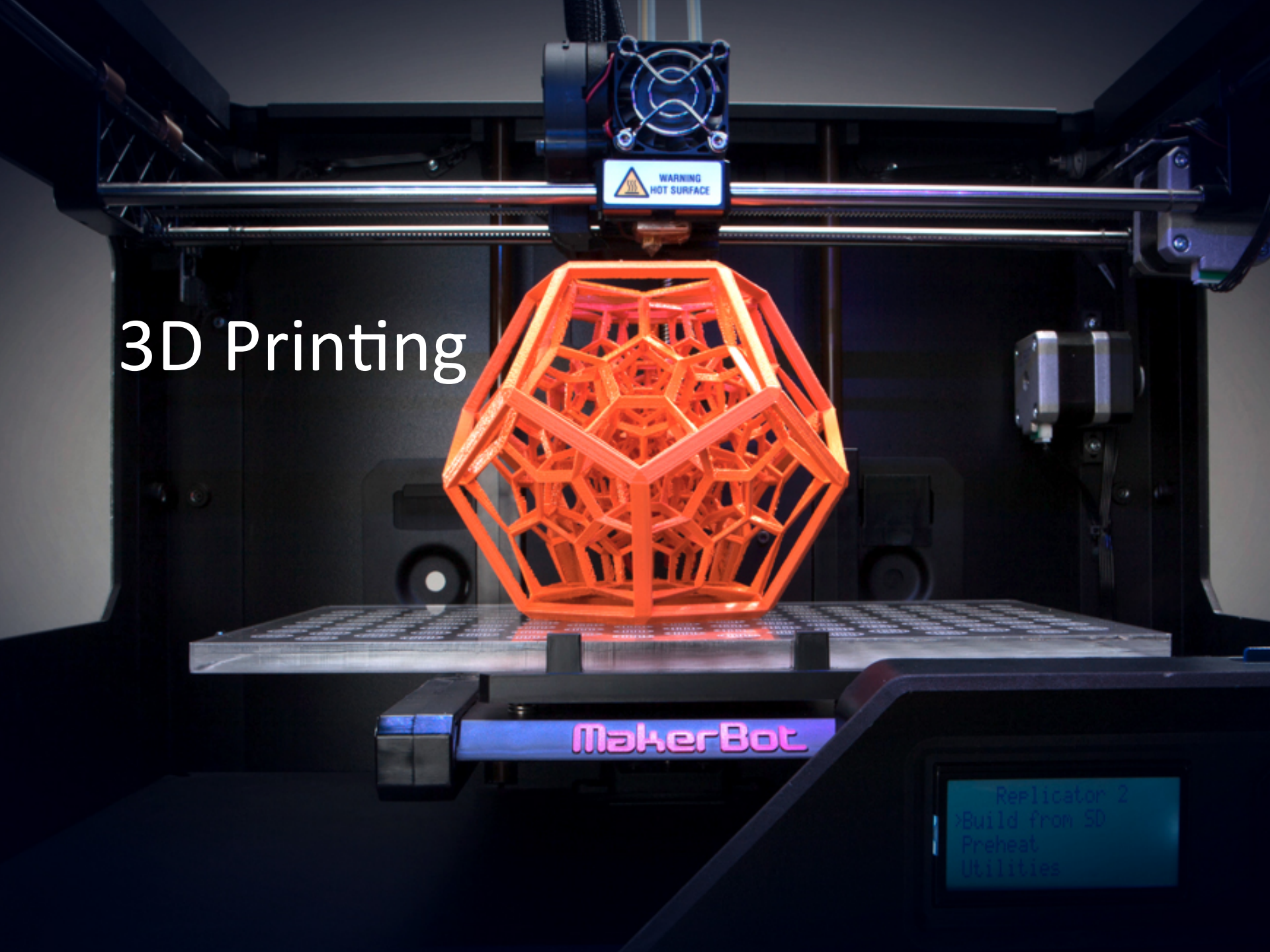
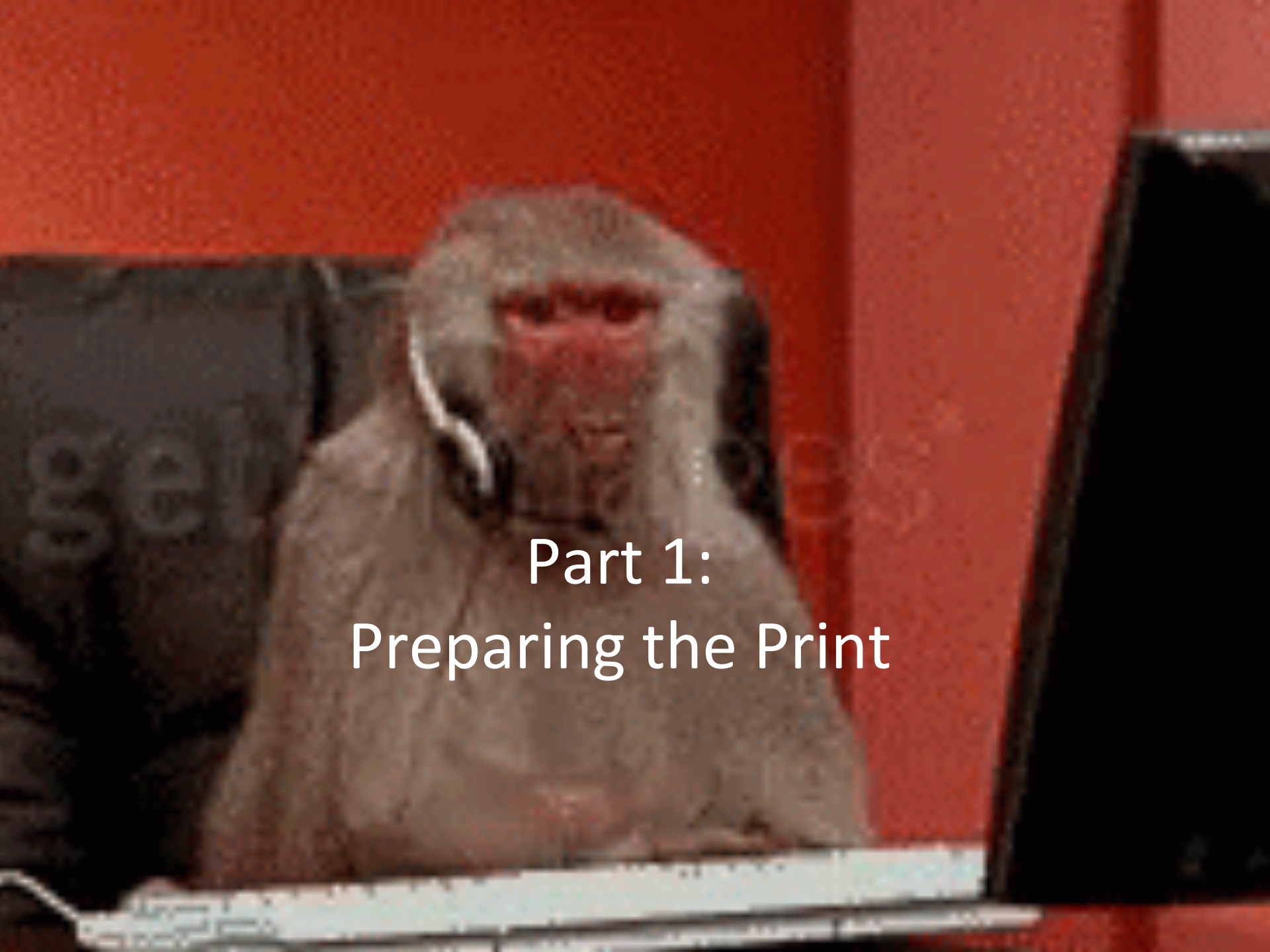

WARNING
HOT SURFACE
MakerBot
Replicator 2
>Build from SD
Preheat
Utilities

3D Printing


Part 1:
Preparing the Print

The process

- Your Idea
- Creating a 3D model
- Slicing the 3D model into a Gcode file
- Selecting settings
- Uploading to the printer

The process

- Your Idea
- Creating a 3D model
- Slicing the 3D model into a Gcode file
- Selecting settings
- Uploading to the printer

The process

- Your Idea
- Creating a 3D model
- Slicing the 3D model into a Gcode file
- Selecting settings
- Uploading to the printer

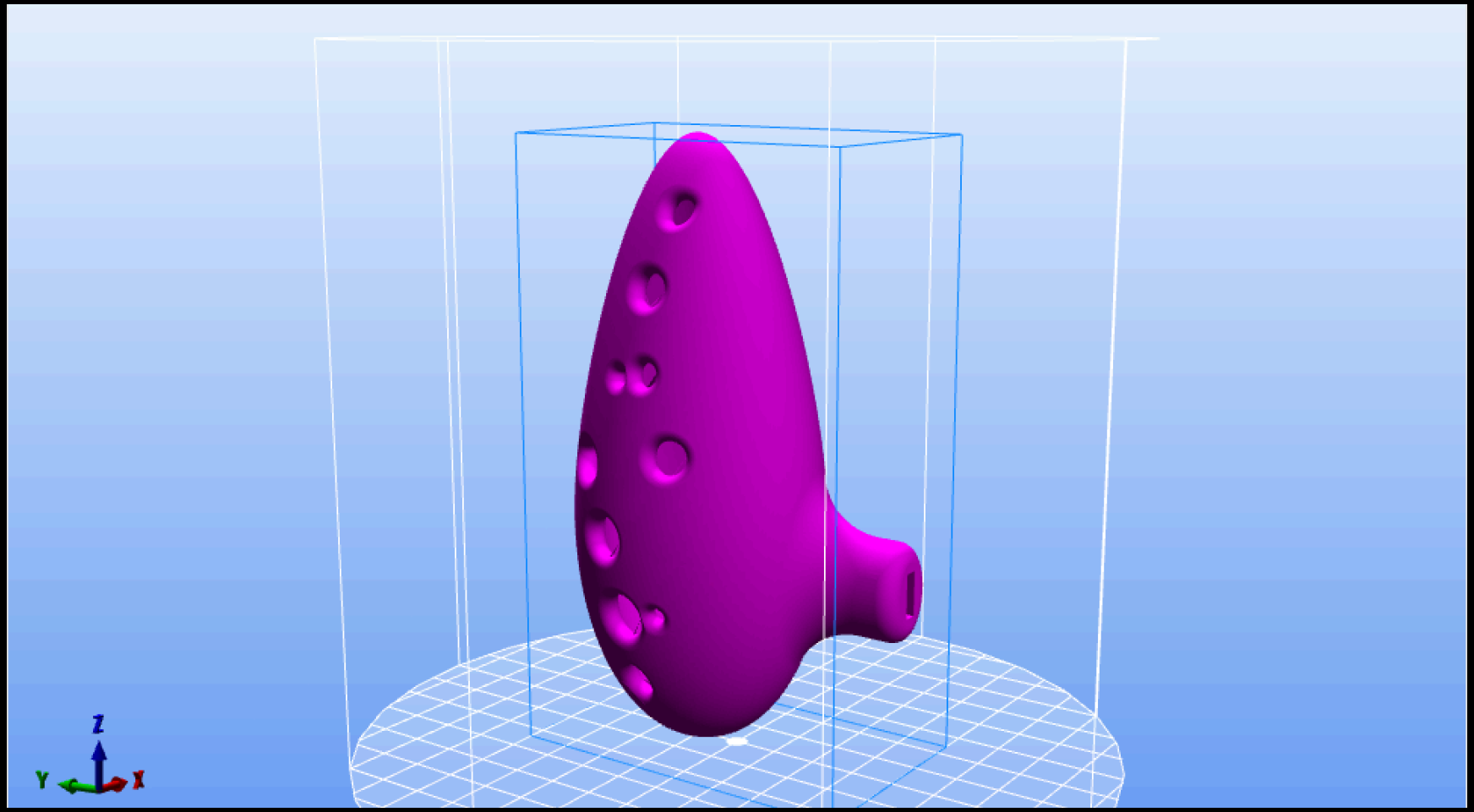
The process

- Your Idea
- Creating a 3D model
- Slicing the 3D model into a Gcode file
- Selecting settings
- Uploading to the printer

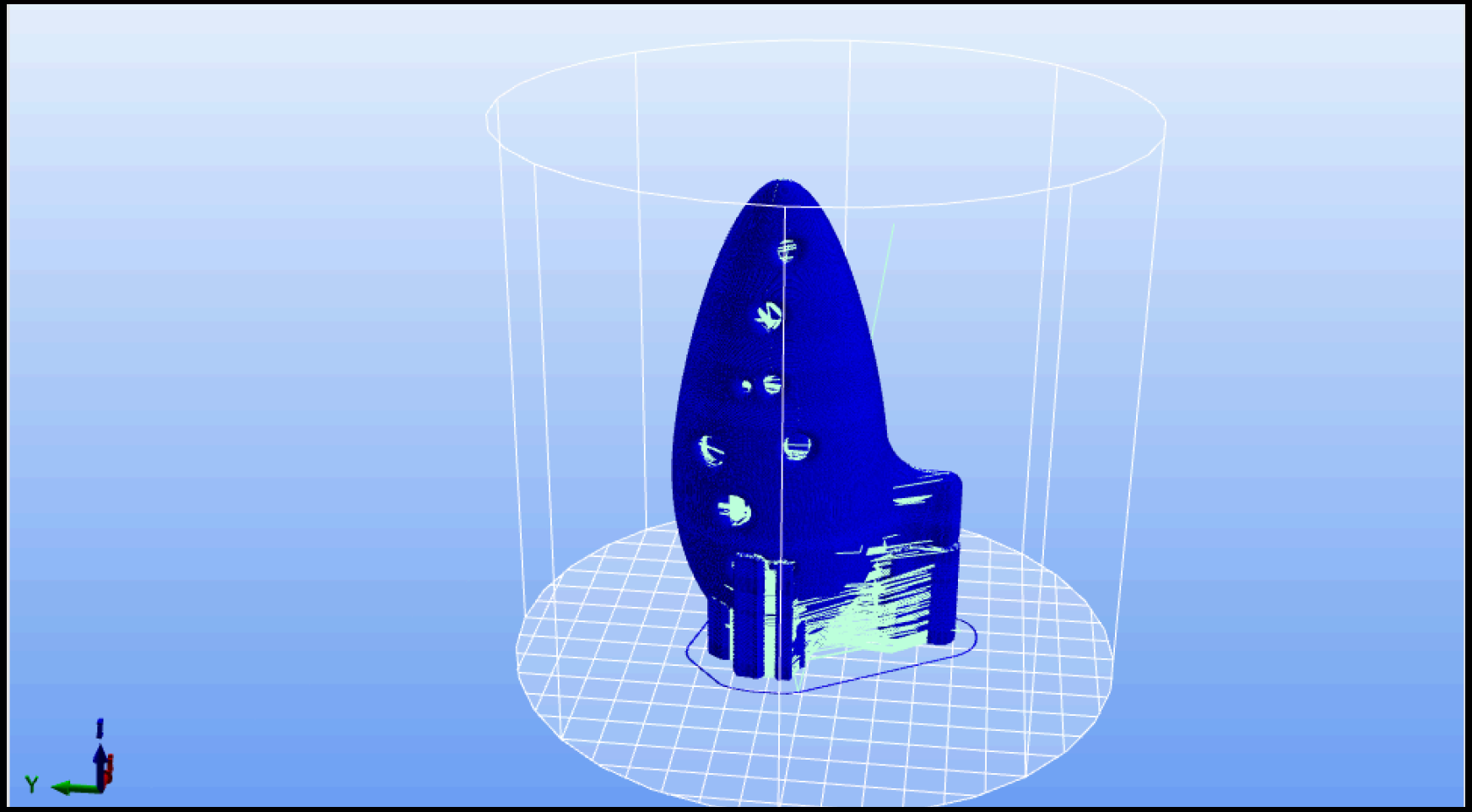
Slicing the 3D Model

- Involves converting the model into set of instructions for the printer
- Start with a .stl or .CAD file
- Import it into a slicer
- Slicer converts the model into a Gcode file

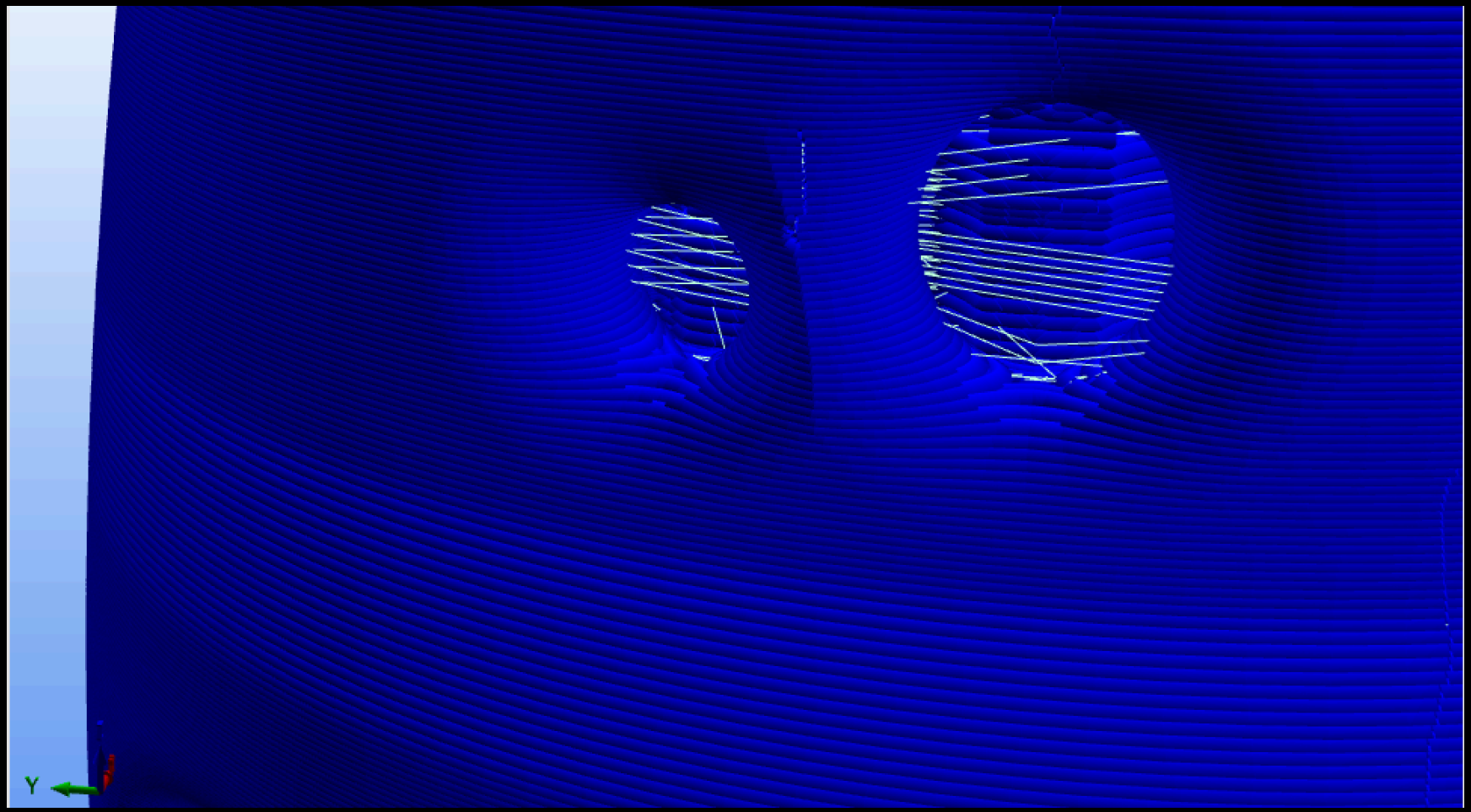
Slicing



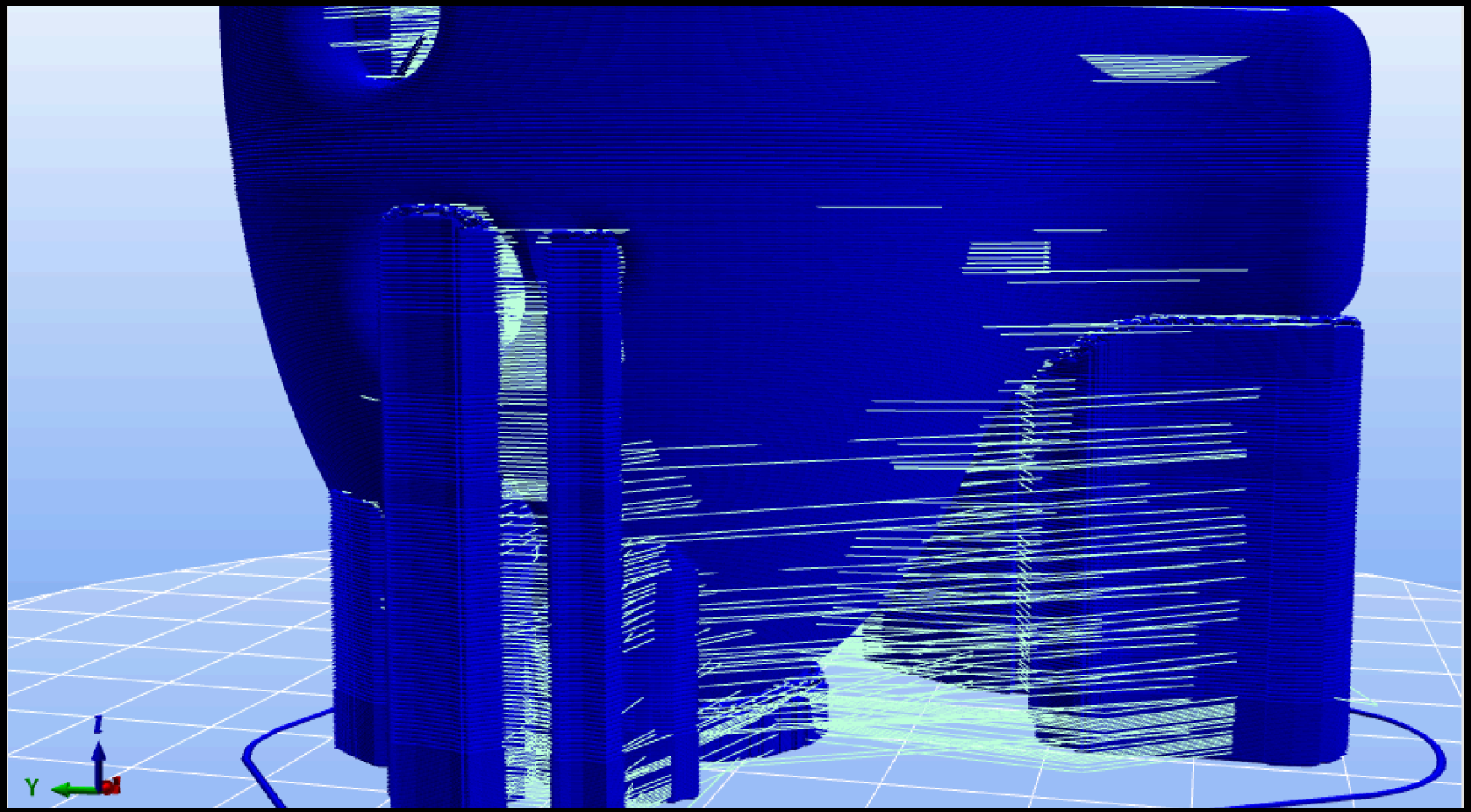
Slicing



Slicing



Slicing



Slicing the 3D Model

- Involves converting the model into set of instructions for the printer
- Start with a .stl or .CAD file
- Import it into a slicer
- Slicer converts the model into a Gcode file

Slicing the 3D Model

- Involves converting the model into set of instructions for the printer
- Start with a .stl or .CAD file
- Import it into a slicer
- Slicer converts the model into a Gcode file

Gcode

- Gcode is a set of instructions that the printer's firmware can understand.

Gcode

```
; infill extrusion width = 0.31mm (0.00mm^3/s)
; solid infill extrusion width = 0.35mm (0.00mm^3/s)
; top infill extrusion width = 0.35mm (0.00mm^3/s)
; support material extrusion width = 0.28mm (0.00mm^3/s)

M107
M104 S210 ; set temperature
G28 ; home all axes
G1 Z5 F5000 ; lift nozzle

; Filament gcode

M109 S210 ; set temperature and wait for it to be reached
G21 ; set units to millimeters
G90 ; use absolute coordinates
M82 ; use absolute distances for extrusion
G92 E0
G1 Z0.250 F7800.000
G1 E-2.00000 F2400.00000
G92 E0
G1 X-29.329 Y1.142 F7800.000
G1 E2.00000 F2400.00000
G1 F1800
G1 X-4.082 Y-41.312 E4.29148
G1 X-2.988 Y-42.685 E4.37291
G1 X-1.922 Y-43.700 E4.44122
G1 X-1.346 Y-44.188 E4.47626
G1 X-0.786 Y-44.608 E4.50872
G1 X0.576 Y-45.392 E4.58163
G1 X1.606 Y-45.826 E4.63346
G1 X3.228 Y-46.273 E4.71151
G1 X4.894 Y-46.504 E4.78956
G1 X5.691 Y-46.565 E4.82666
G1 X6.996 Y-46.566 E4.88719
G1 X8.666 Y-46.468 E4.96480
G1 X11.012 Y-45.876 E5.07704
G1 X12.717 Y-45.082 E5.16428
G1 X14.568 Y-43.782 E5.26923
G1 X15.348 Y-42.830 E5.32634
G1 X16.238 Y-41.504 E5.40045
G1 X16.573 Y-40.946 E5.43064
```

Gcode

- Gcode has different flavours for different printer firmwares
- For example: marlin, repetier, etc
- The firmware and the slicer have to speak the same “language”

Examples of Gcode (repetier)

- G28: Home all axes
- G21: Set units to mm
- G1: linear movement to specific coordinate Xnn Ynn Znn

Gcode

```
; infill extrusion width = 0.31mm (0.00mm^3/s)
; solid infill extrusion width = 0.35mm (0.00mm^3/s)
; top infill extrusion width = 0.35mm (0.00mm^3/s)
; support material extrusion width = 0.28mm (0.00mm^3/s)

M107
M104 S210 ; set temperature
G28 ; home all axes
G1 Z5 F5000 ; lift nozzle

; Filament gcode

M109 S210 ; set temperature and wait for it to be reached
G21 ; set units to millimeters
G90 ; use absolute coordinates
M82 ; use absolute distances for extrusion
G92 E0
G1 Z0.250 F7800.000
G1 E-2.00000 F2400.00000
G92 E0
G1 X-29.329 Y1.142 F7800.000
G1 E2.00000 F2400.00000
G1 F1800
G1 X-4.082 Y-41.312 E4.29148
G1 X-2.988 Y-42.685 E4.37291
G1 X-1.922 Y-43.700 E4.44122
G1 X-1.346 Y-44.188 E4.47626
G1 X-0.786 Y-44.608 E4.50872
G1 X0.576 Y-45.392 E4.58163
G1 X1.606 Y-45.826 E4.63346
G1 X3.228 Y-46.273 E4.71151
G1 X4.894 Y-46.504 E4.78956
G1 X5.691 Y-46.565 E4.82666
G1 X6.996 Y-46.566 E4.88719
G1 X8.666 Y-46.468 E4.96480
G1 X11.012 Y-45.876 E5.07704
G1 X12.717 Y-45.082 E5.16428
G1 X14.568 Y-43.782 E5.26923
G1 X15.348 Y-42.830 E5.32634
G1 X16.238 Y-41.504 E5.40045
G1 X16.573 Y-40.946 E5.43064
```

The process

- Your Idea
- Creating a 3D model
- Slicing the 3D model into a Gcode file
- Selecting settings
- Uploading to the printer

Choosing Settings

- This is arguably the longest and most tedious part of 3D printing
- You only have the do it the first time you set up your slicing software
- This is the most important aspect for getting quality prints

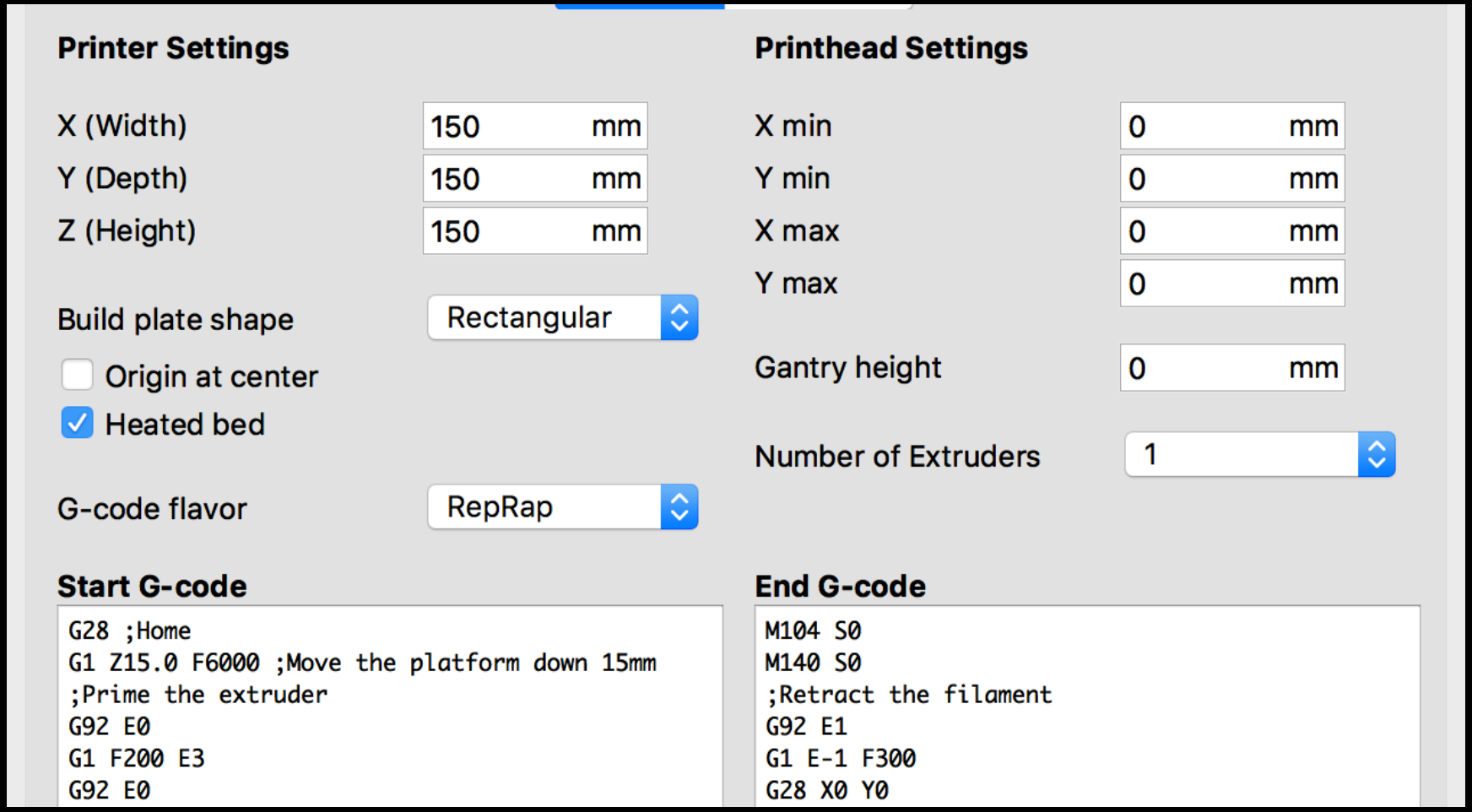
Choosing Settings

- Also most boring, so will be brief
- Takes experience and time to get through all of them
- Two types: machine settings and print settings

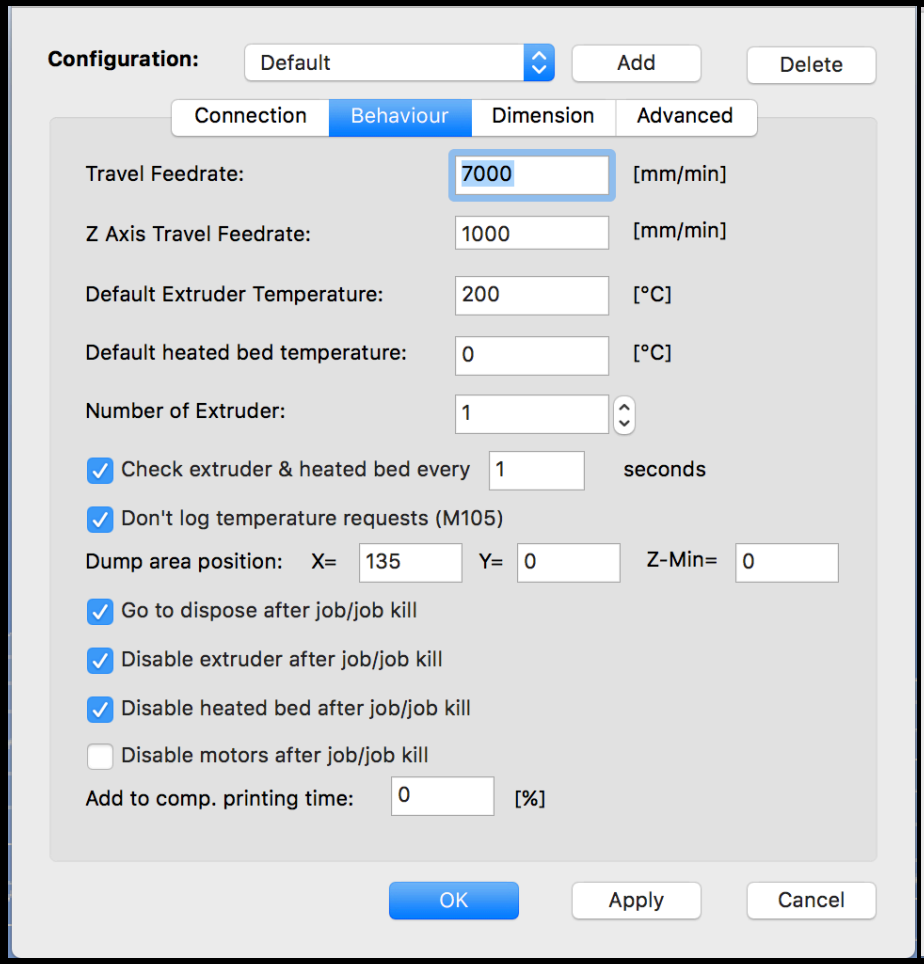
Machine Settings

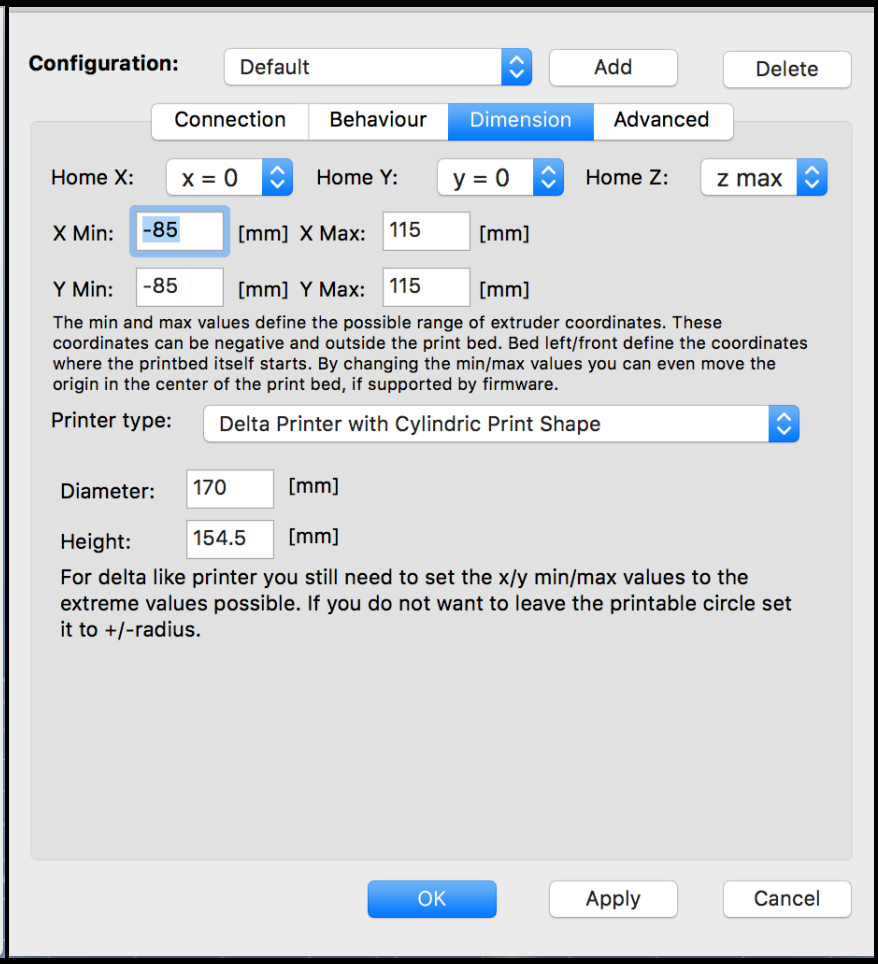
- These are settings specific to your machine
- Includes things like the size of your print space, how far the motors can move, etc.
- Get them wrong and you'll have grinding, scraping, or hot flying cubes of 190º metal

Machine Settings (Cura)



Machine Settings (Repetier)





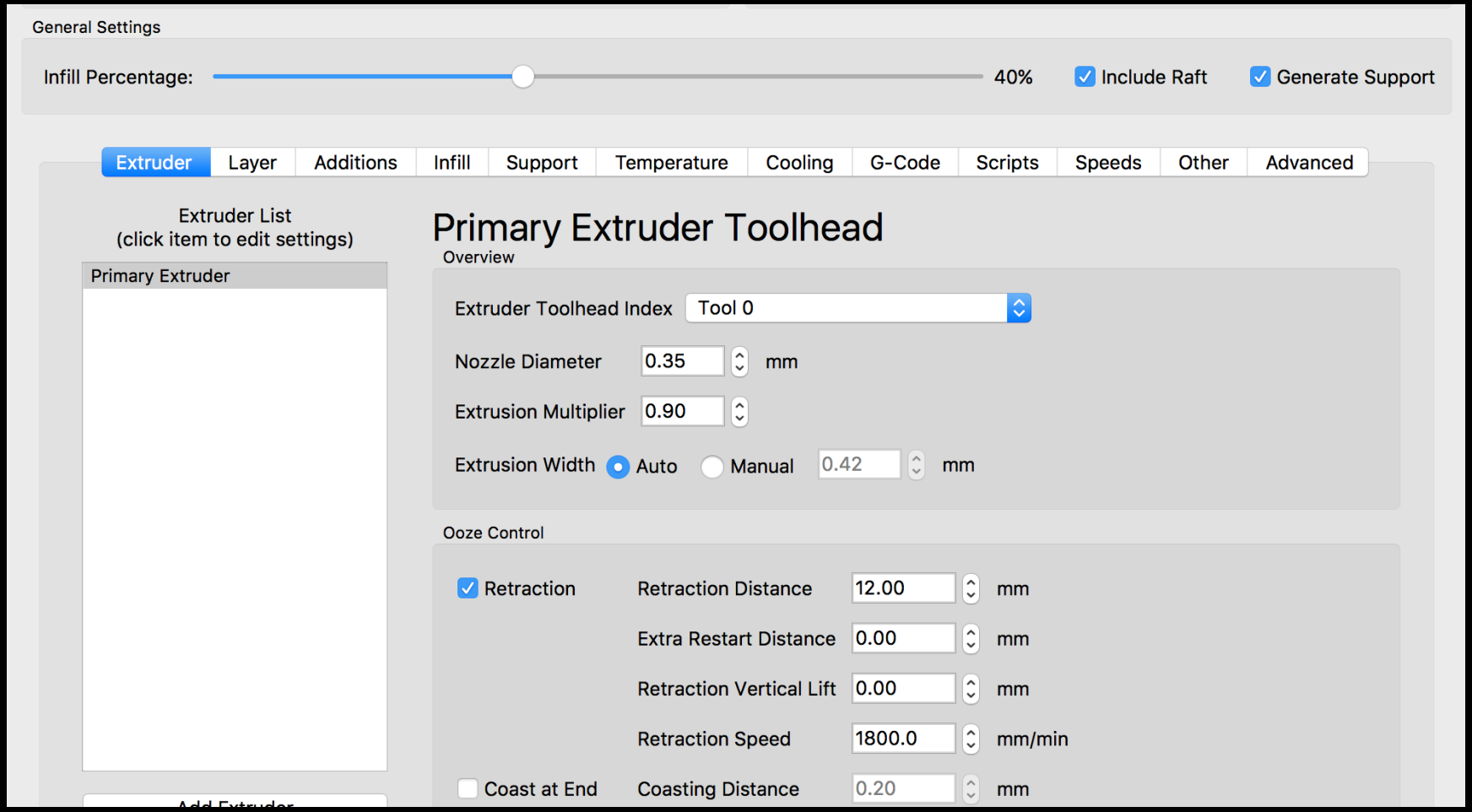
Machine Settings

- Specific to your type of machine, not too difficult to get right
- Slightly more difficult when you build your own 3D printer
- You have to accurately measure things yourself

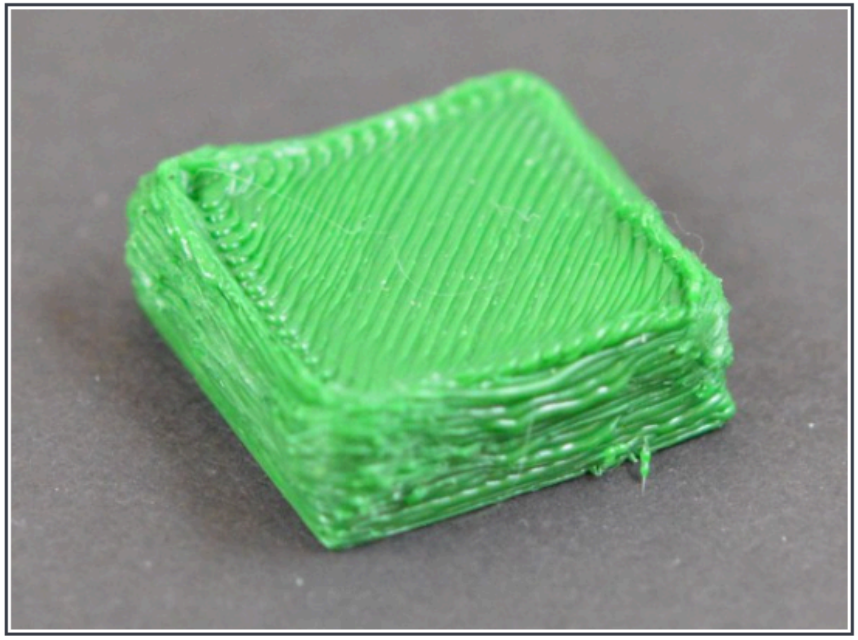
Print Settings

- Settings relevant to the print you are doing
- Can change from print to print
- most features are common to all prints
- Also influenced by hardware, and materials used to print

Extruder


General Settings
Infill Percentage:
40%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
Extruder List
(click item to edit settings)
Primary Extruder
Primary Extruder Toolhead
Overview
Extruder Toolhead Index
Tool 0
Nozzle Diameter
0.35
mm
Extrusion Multiplier
0.90
Extrusion Width
Auto
Manual
0.42
mm
Ooze Control
Retraction
Retraction Distance
12.00
mm
Extra Restart Distance
0.00
mm
Retraction Vertical Lift
0.00
mm
Retraction Speed
1800.0
mm/min
Coast at End
Coasting Distance
0.20
mm

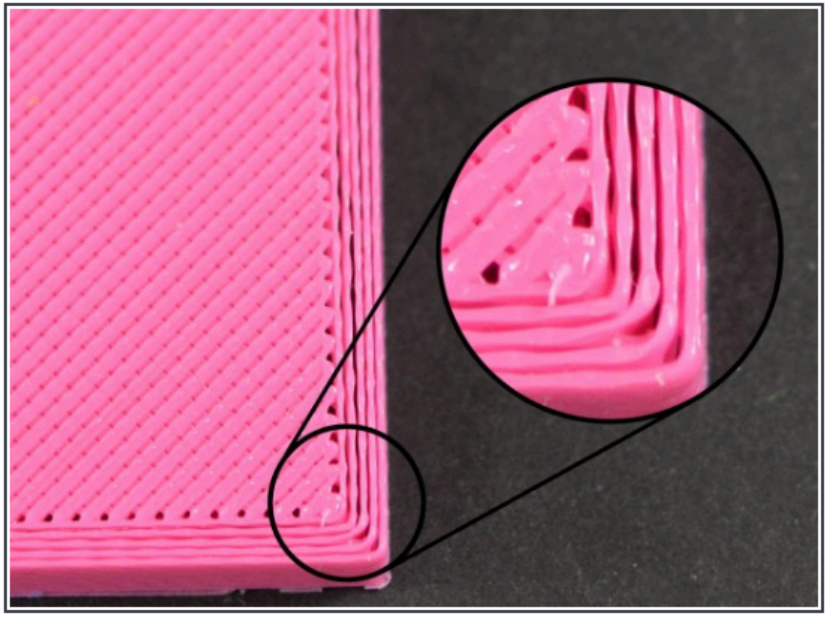
Extruder



Over-Extrusion

Printer extrudes too much plastic, prints looks very messy

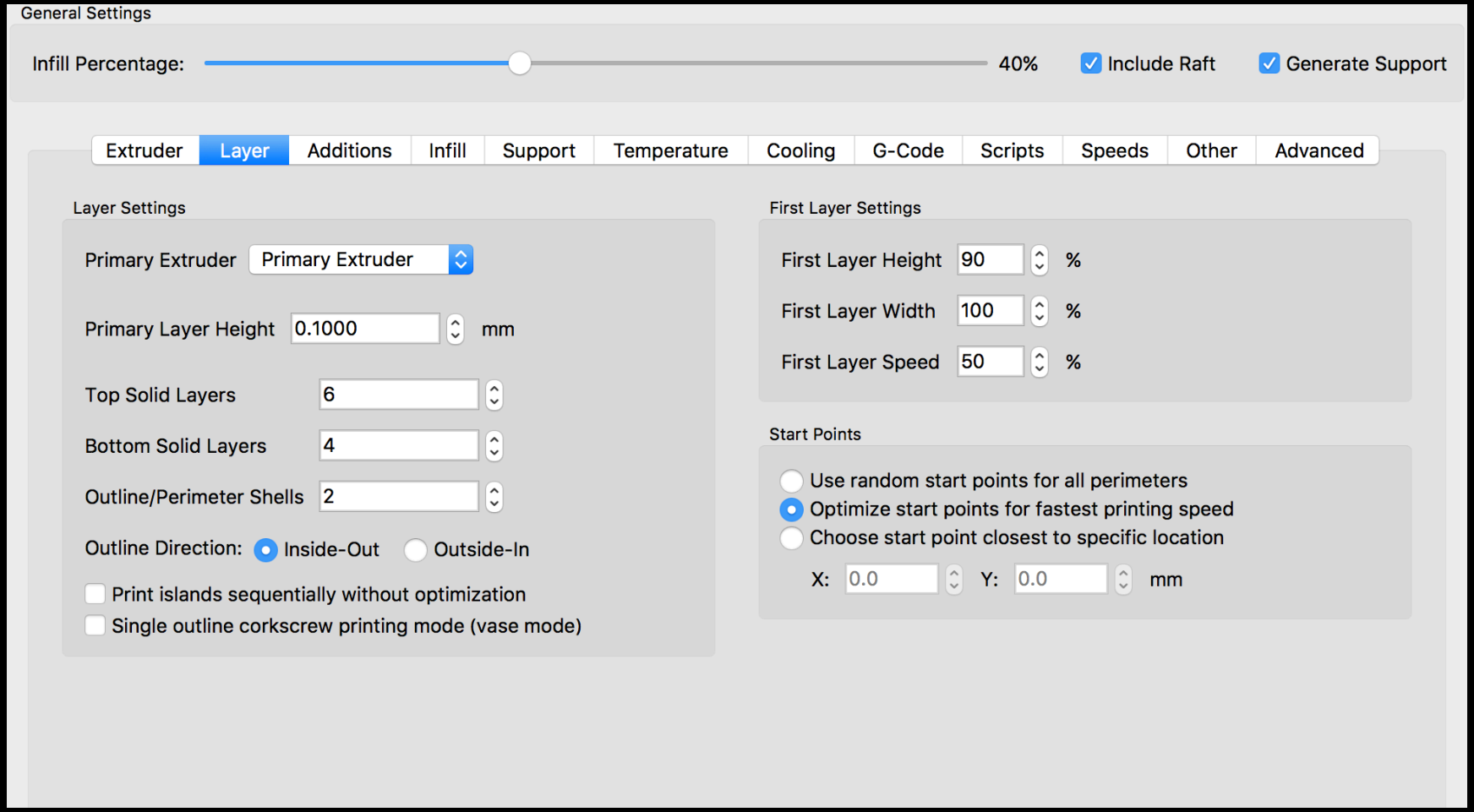
Extruder



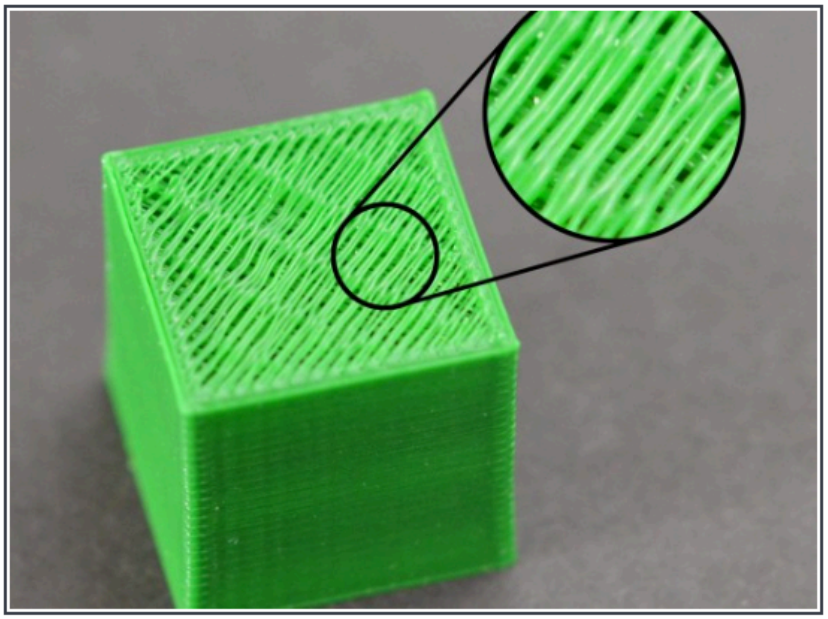
Under-Extrusion

Printer does not extrude enough plastic, gaps between perimeters and infill

Layer


General Settings
Infill Percentage:
40%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
Layer Settings
Primary Extruder
Primary Extruder
Primary Layer Height
0.1000
mm
Top Solid Layers
6
Bottom Solid Layers
4
Outline/Perimeter Shells
2
Outline Direction:
Inside-Out
Outside-In
Print islands sequentially without optimization
Single outline corkscrew printing mode (vase mode)
First Layer Settings
First Layer Height
90
%
First Layer Width
100
%
First Layer Speed
50
%
Start Points
Use random start points for all perimeters
Optimize start points for fastest printing speed
Choose start point closest to specific location
X:
0.0
Y:
0.0
mm

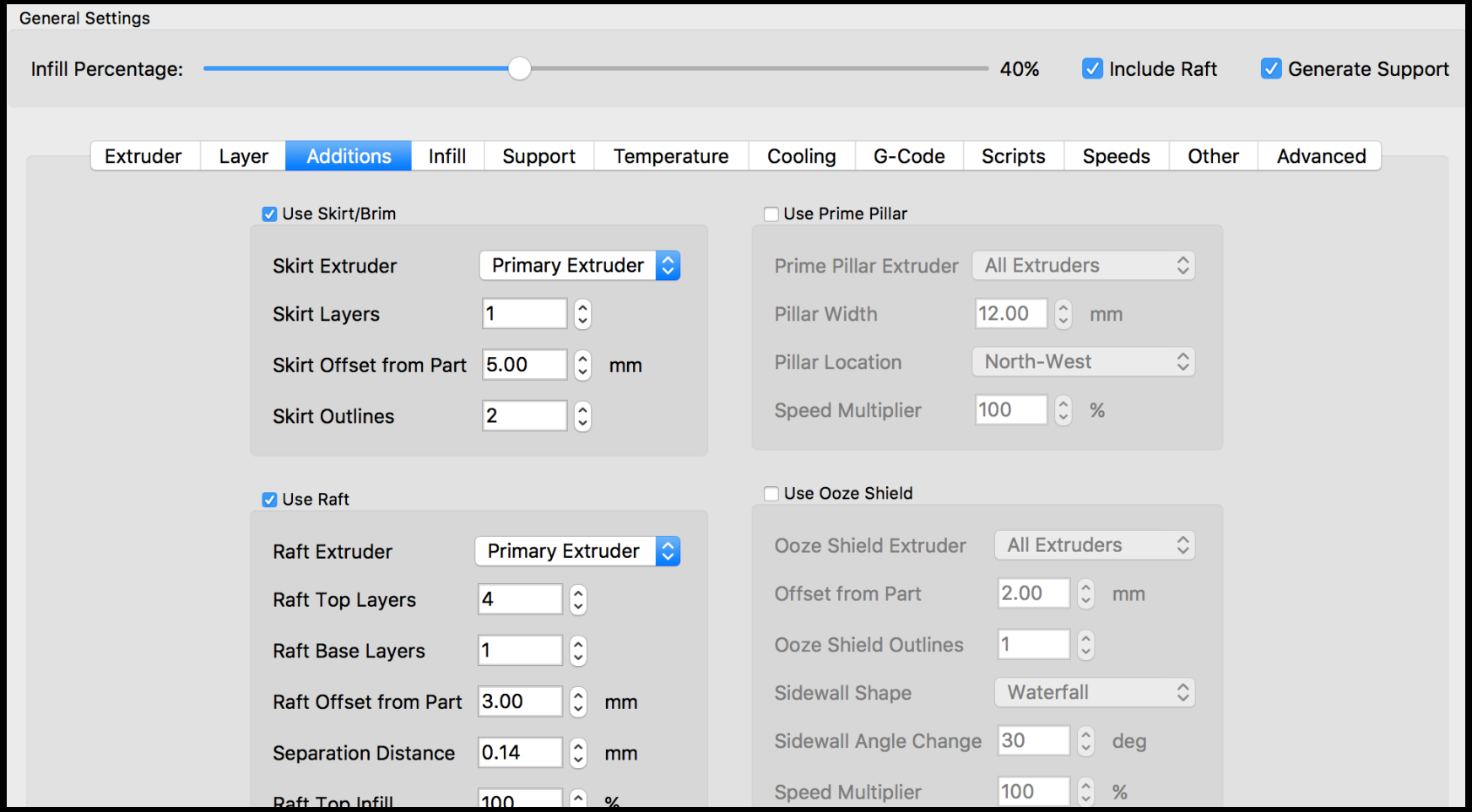
Extruder



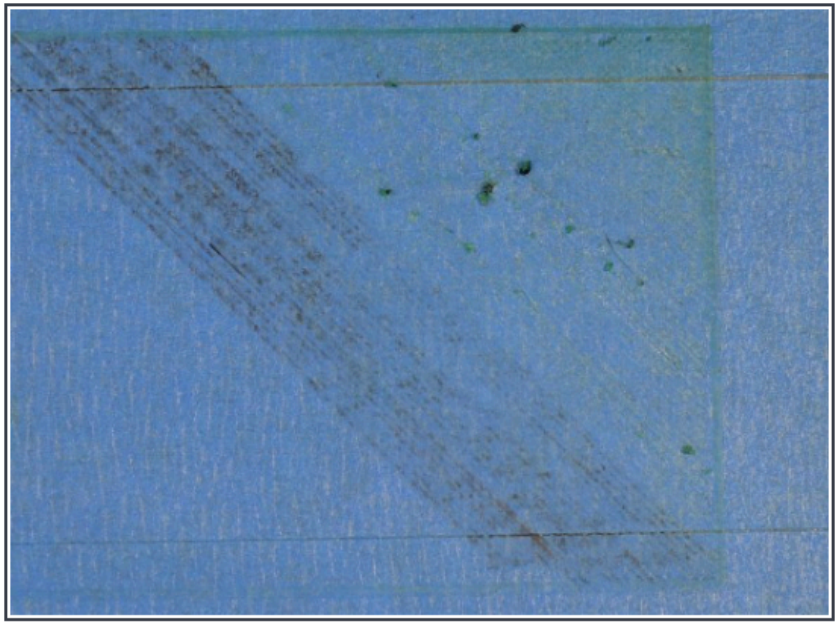
Gaps in Top Layers

Holes or gaps in the top layers of the print

Additions


General Settings
Infill Percentage:
40%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
Use Skirt/Brim
Skirt Extruder
Primary Extruder
Skirt Layers
1
Skirt Offset from Part
5.00
mm
Skirt Outlines
2
Use Raft
Raft Extruder
Primary Extruder
Raft Top Layers
4
Raft Base Layers
1
Raft Offset from Part
3.00
mm
Separation Distance
0.14
mm
Use Prime Pillar
Prime Pillar Extruder
All Extruders
Pillar Width
12.00
mm
Pillar Location
North-West
Speed Multiplier
100
%
Use Ooze Shield
Ooze Shield Extruder
All Extruders
Offset from Part
2.00
mm
Ooze Shield Outlines
1
Sidewall Shape
Waterfall
Sidewall Angle Change
30
deg
Speed Multiplier
100
%

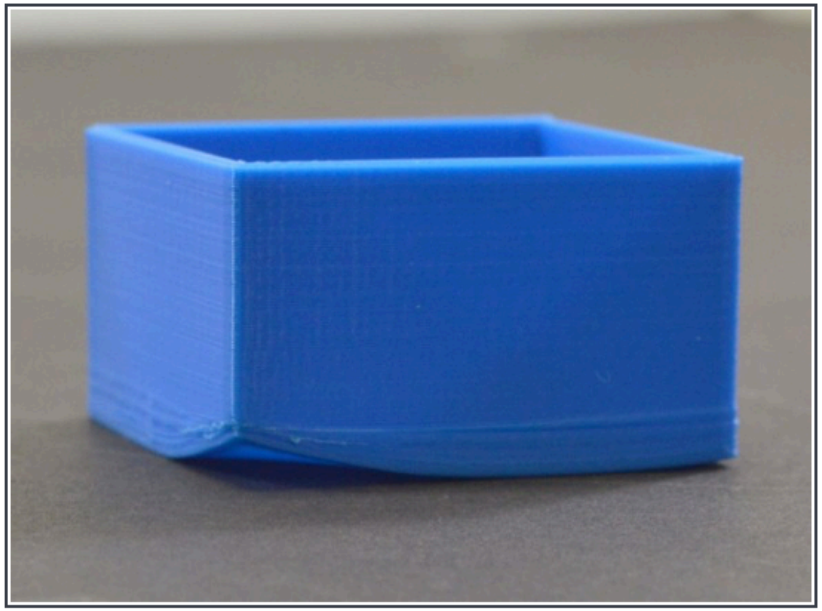
Additions



Not Extruding At Start

Printer does not extrude plastic at the beginning of the print

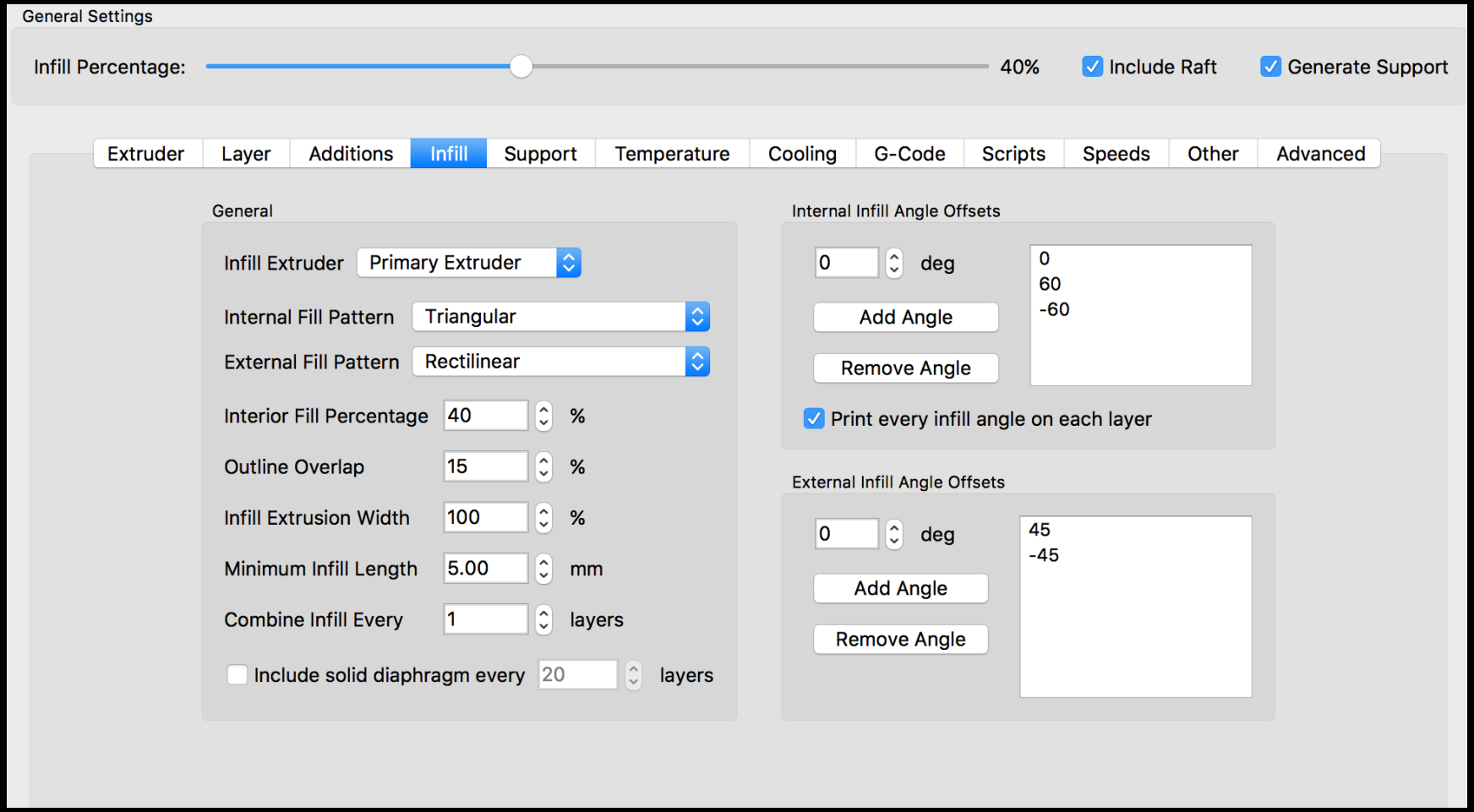
Additions



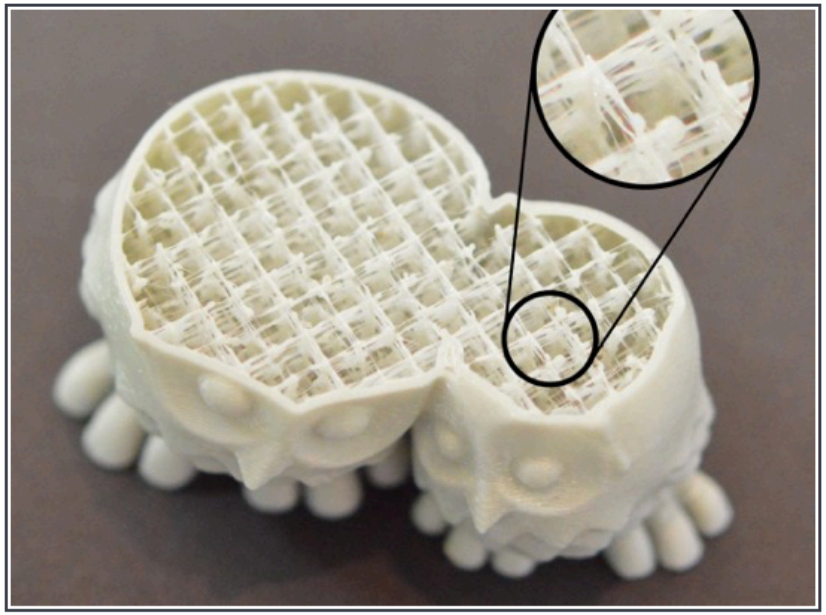
Warping

Warping of large parts, particularly with high temperature materials such as ABS

Infill


General Settings
Infill Percentage:
40%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
General
Infill Extruder
Primary Extruder
Internal Fill Pattern
Triangular
External Fill Pattern
Rectilinear
Interior Fill Percentage
40
%
Outline Overlap
15
%
Infill Extrusion Width
100
%
Minimum Infill Length
5.00
mm
Combine Infill Every
1
layers
Include solid diaphragm every
20
layers
Internal Infill Angle Offsets
0
deg
Add Angle
Remove Angle
0
60
-60
Print every infill angle on each layer
External Infill Angle Offsets
0
deg
Add Angle
Remove Angle
45
-45

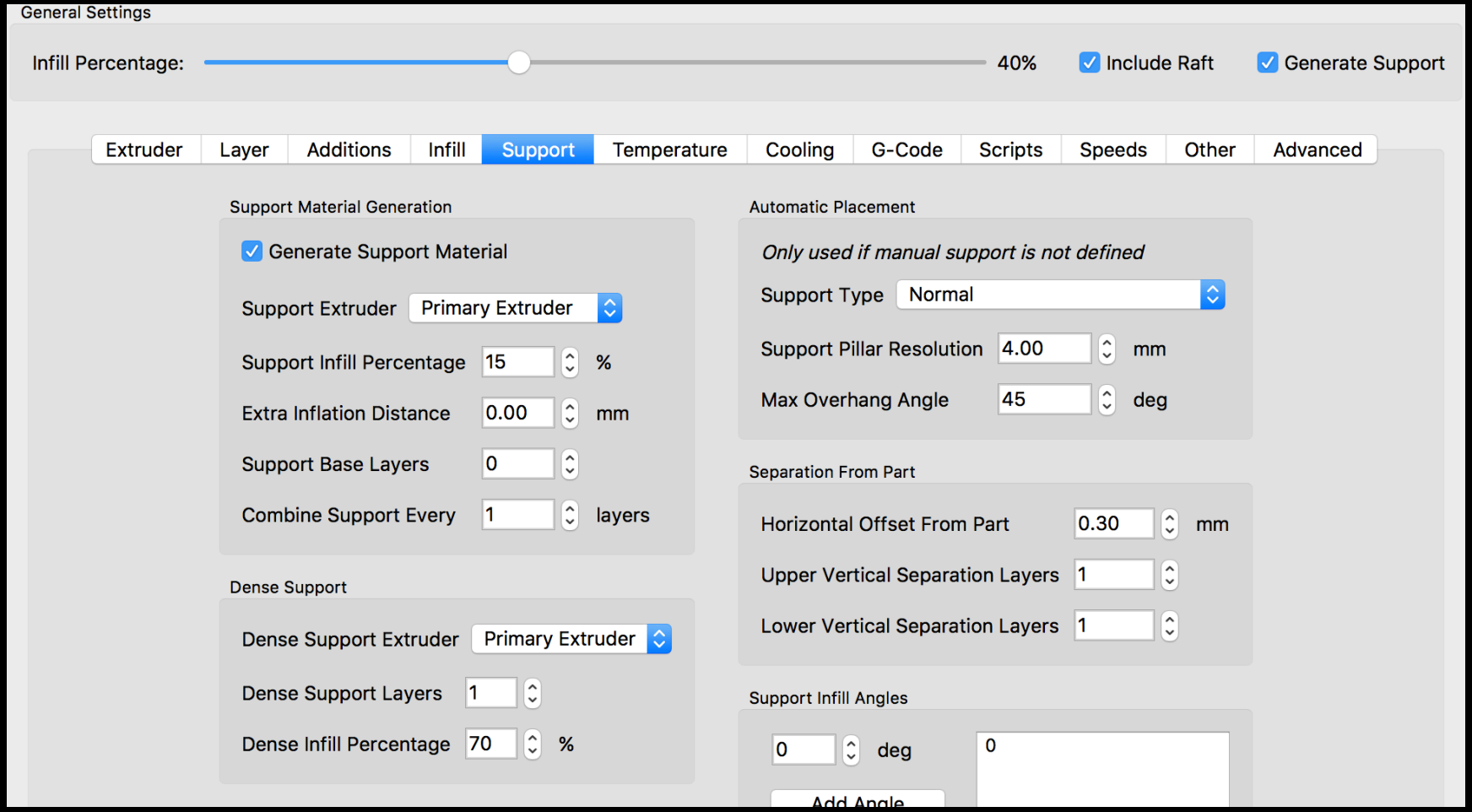
Infill



Weak Infill

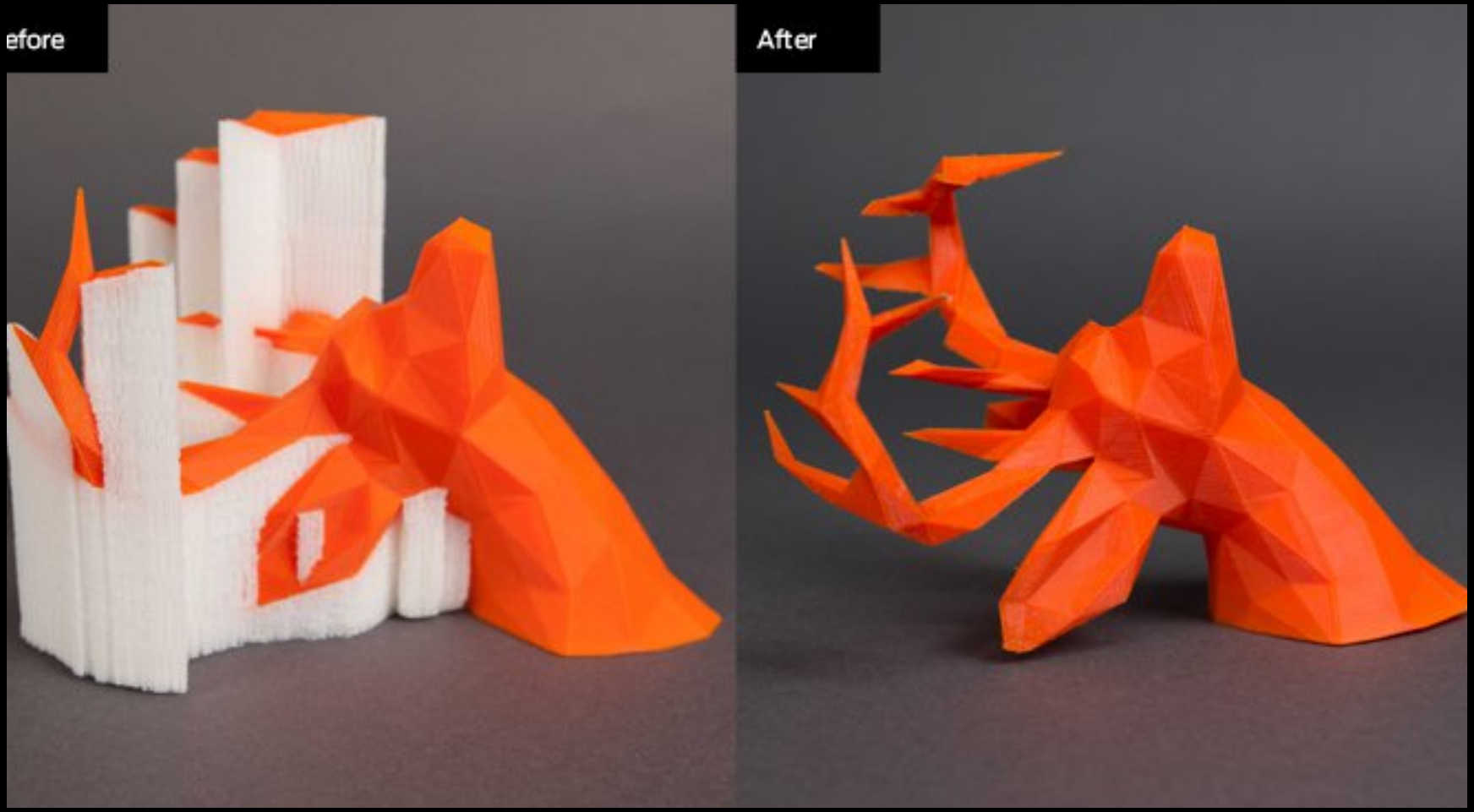
Very thin, stringy infill that creates a weak interior and does not bond together well

Support

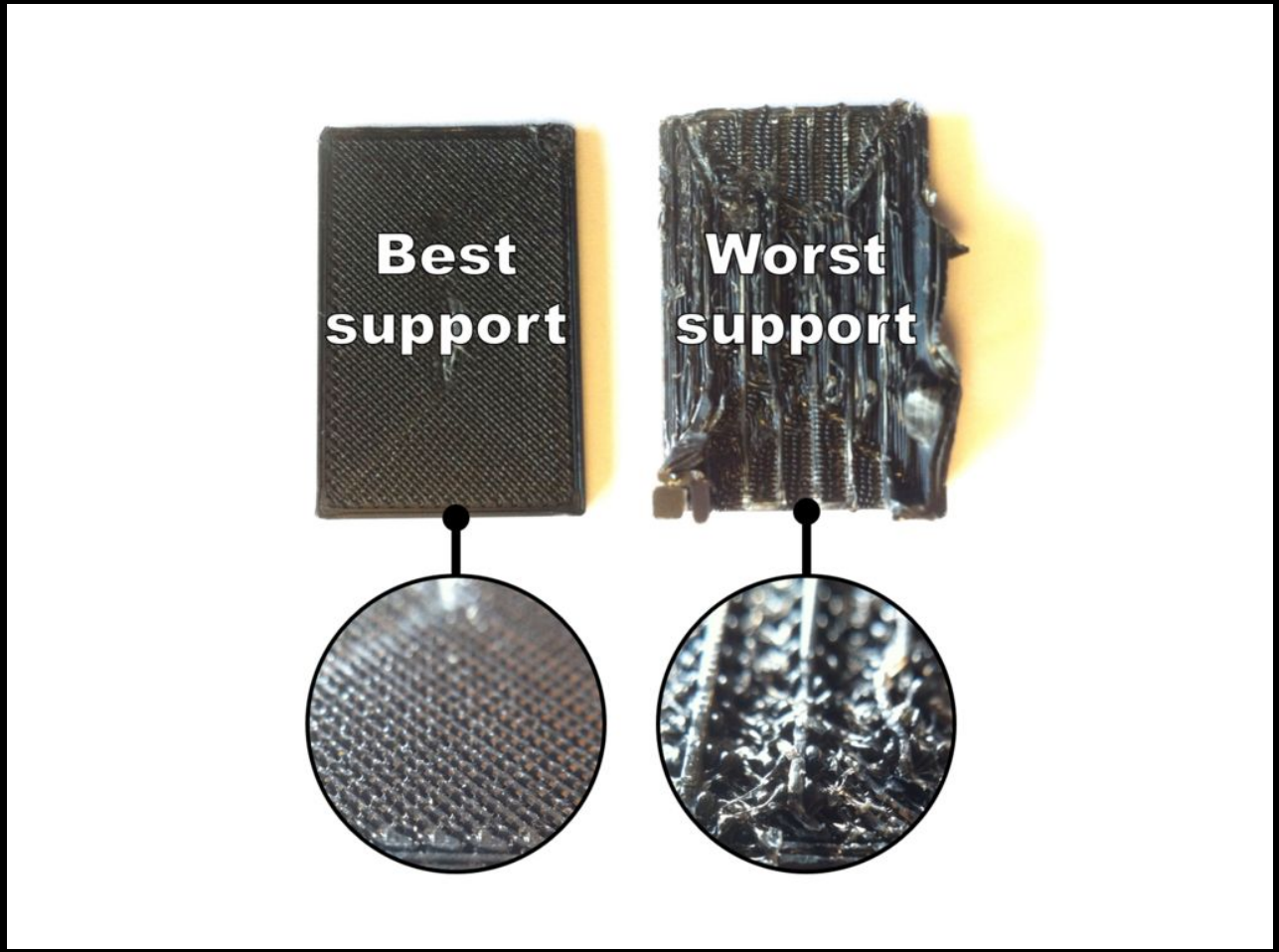

General Settings
Infill Percentage:
40%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
Support Material Generation
Generate Support Material
Support Extruder
Primary Extruder
Support Infill Percentage
15
%
Extra Inflation Distance
0.00
mm
Support Base Layers
0
Combine Support Every
1
layers
Dense Support
Dense Support Extruder
Primary Extruder
Dense Support Layers
1
Dense Infill Percentage
70
%
Automatic Placement
Only used if manual support is not defined
Support Type
Normal
Support Pillar Resolution
4.00
mm
Max Overhang Angle
45
deg
Separation From Part
Horizontal Offset From Part
0.30
mm
Upper Vertical Separation Layers
1
Lower Vertical Separation Layers
1
Support Infill Angles
0
deg
0
Add Angle

Support

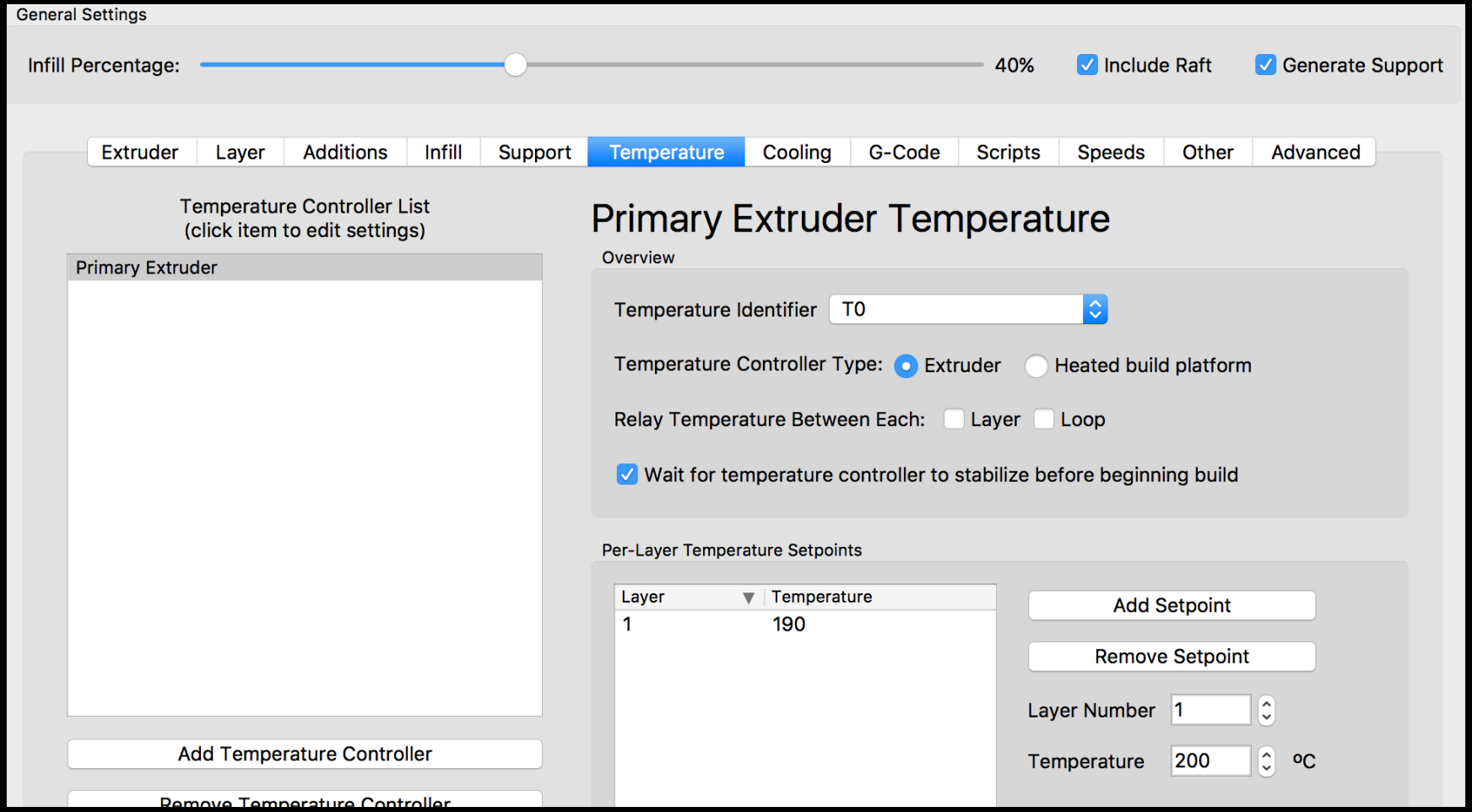
Support



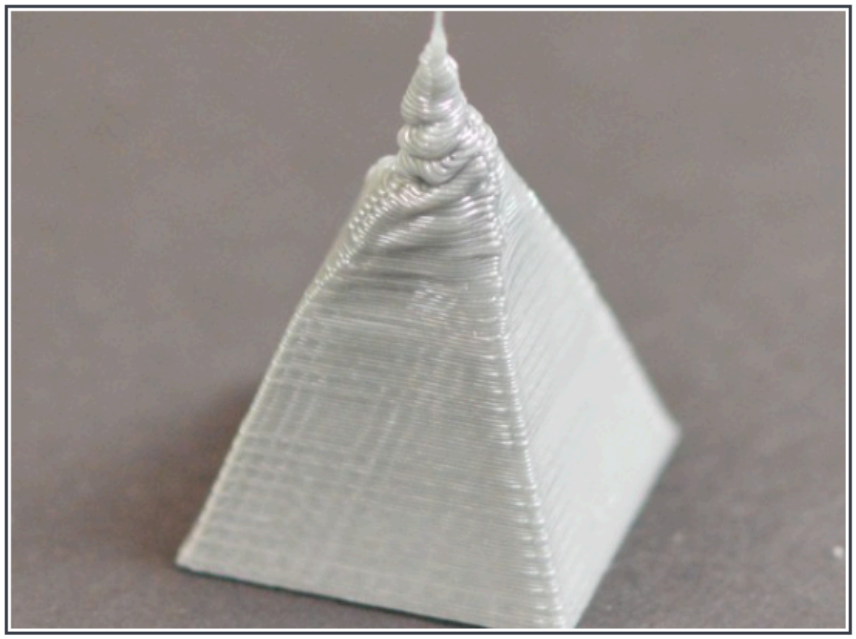
Support



Temperature



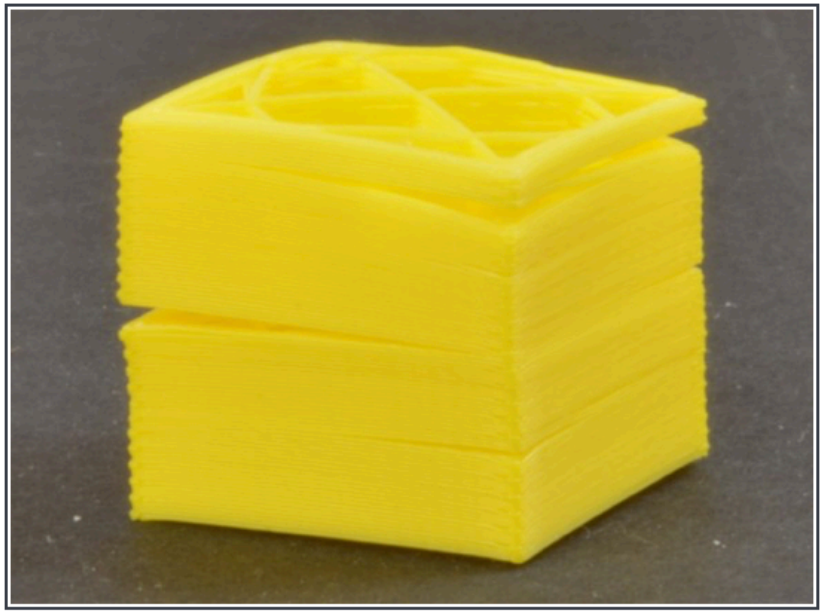
Temperature & Cooling



Overheating

Small features become overheated and deformed

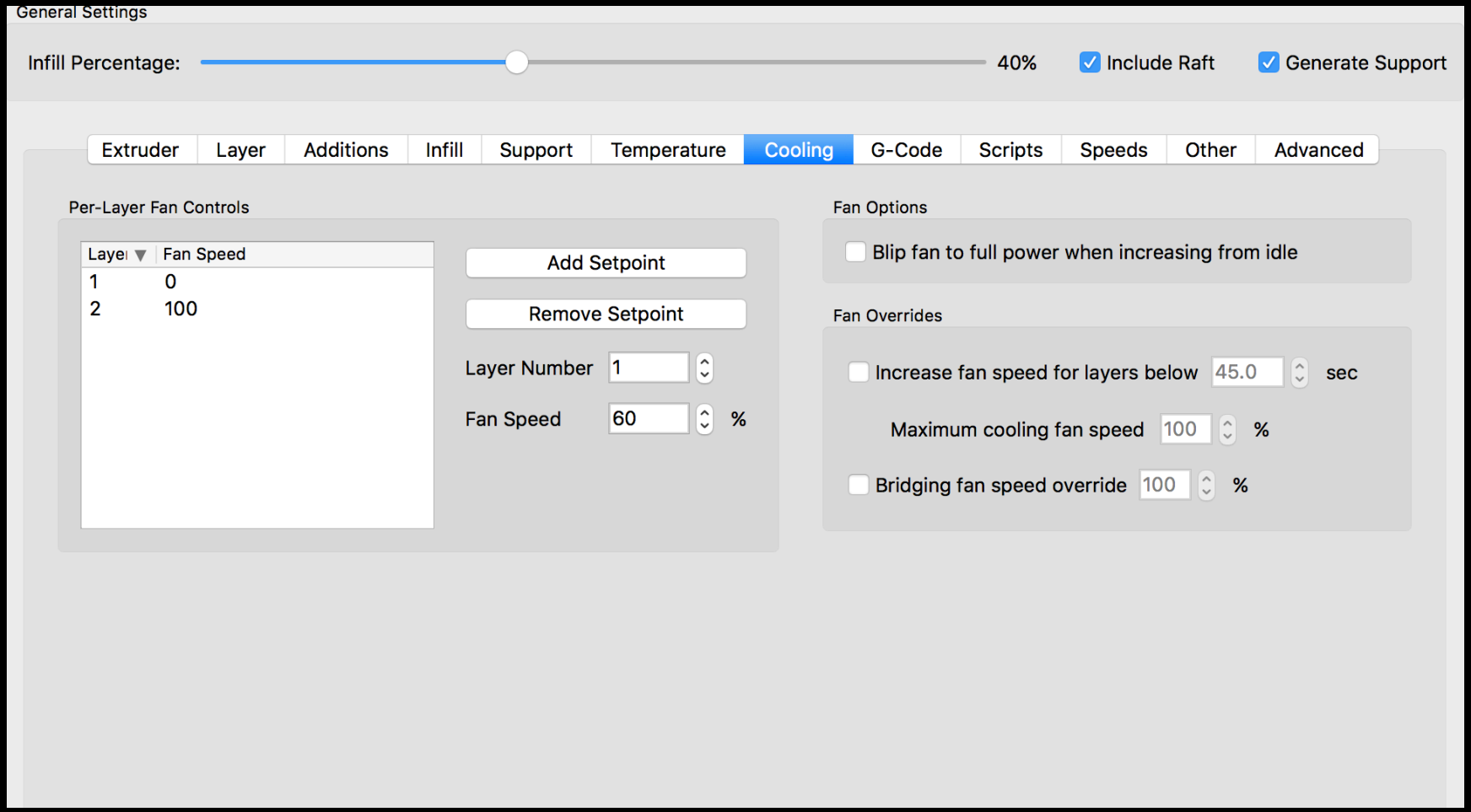
Temperature



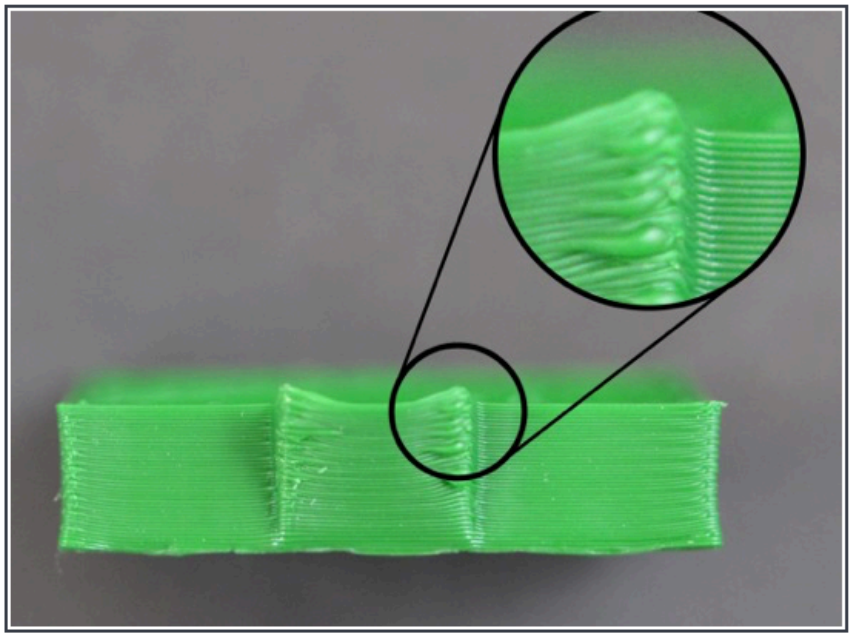
Layer Separation and Splitting

Layers are separating and splitting apart while printing

Cooling


General Settings
Infill Percentage:
40%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
Per-Layer Fan Controls
Layer
Fan Speed
1
0
2
100
Add Setpoint
Remove Setpoint
Layer Number
1
Fan Speed
60
%
Fan Options
Blip fan to full power when increasing from idle
Fan Overrides
Increase fan speed for layers below
45.0
sec
Maximum cooling fan speed
100
%
Bridging fan speed override
100
%

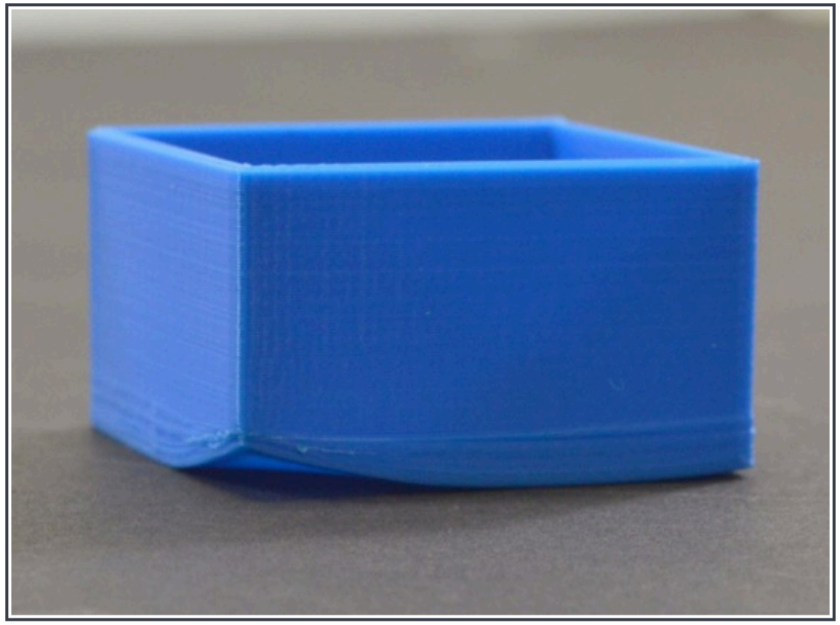
Temperature & Cooling



Curling or Rough Corners

Corners of the print tend to curl and deform after they are printed

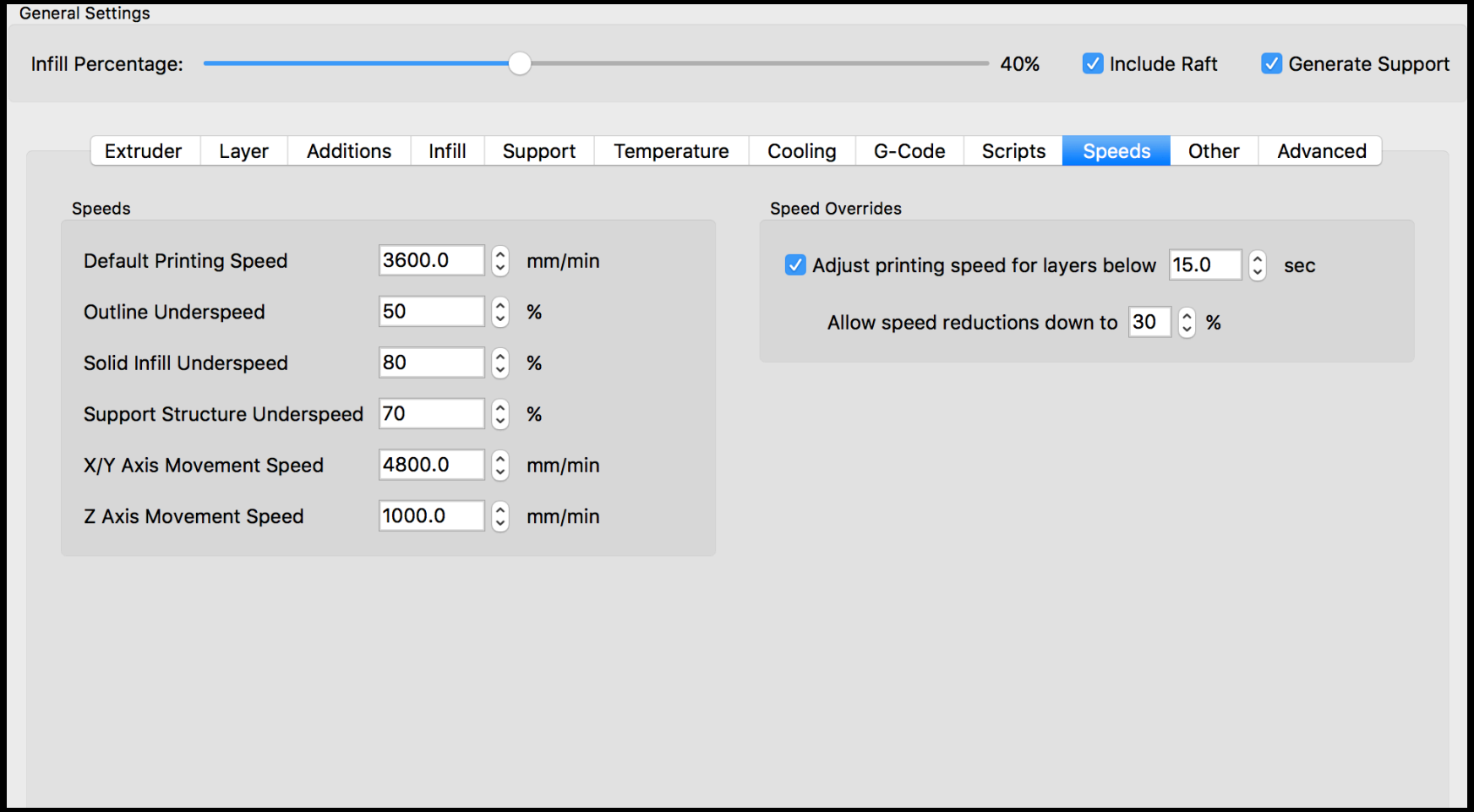
Temperature & Cooling



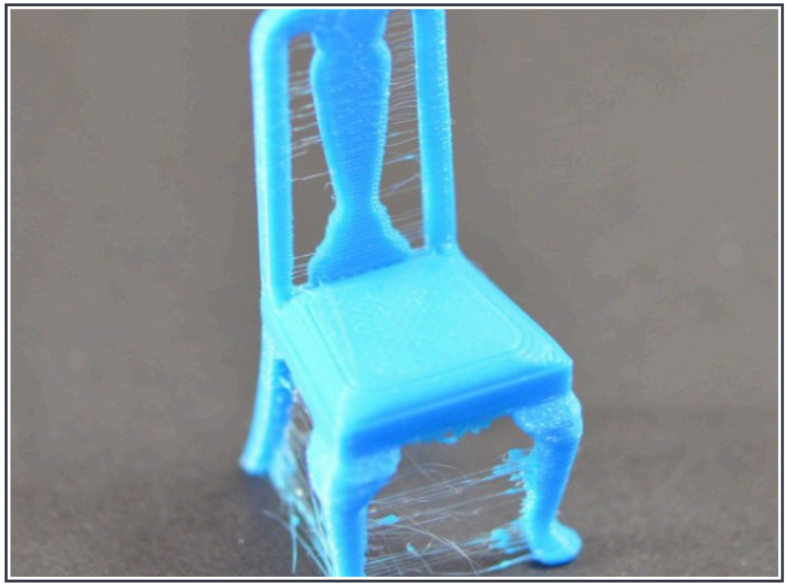
Warping

Warping of large parts, particularly with high temperature materials such as ABS

Speeds


General Settings
Infill Percentage:
40%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
Speeds
Default Printing Speed
3600.0
mm/min
Outline Underspeed
50
%
Solid Infill Underspeed
80
%
Support Structure Underspeed
70
%
X/Y Axis Movement Speed
4800.0
mm/min
Z Axis Movement Speed
1000.0
mm/min
Speed Overrides
Adjust printing speed for layers below
15.0
sec
Allow speed reductions down to
30
%

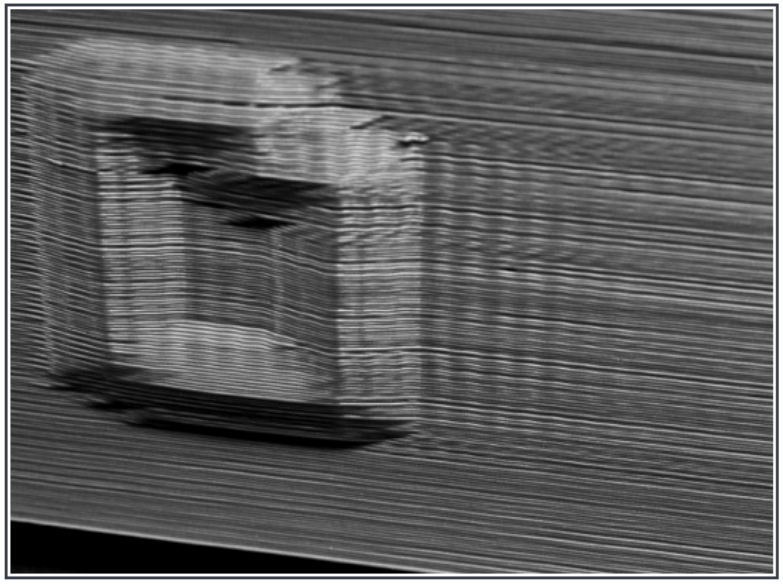
Speeds



Stringing or Oozing

Lots of strings and hairs left behind when moving between different sections of the print

Speeds



Vibrations and Ringing

Vibrations that cause oscillations on the surface of the print, otherwise known as "ringing"

The process

- Your Idea
- Creating a 3D model
- Slicing the 3D model into a Gcode file
- Selecting settings
- Uploading to the printer

Uploading

- Some softwares have inbuilt Slicers (Simplify3D, Repetier host)
- These require that you are connected to the printer while printing
- Other printers have in built computers to execute Gcode
- With these, you just flash the Gcode onto the printer and press print

Uploading

- Connect the printer via USB and open a print hosting software
- Choose your Gcode file and hit print

or

- Usually, printers take SD cards or USB drives
- Just flash the Gcode file from the slicer onto your external drive and plug it into the printer

Upload


Part 2:
The Printing

Printing

You've uploaded your Gcode, now its up to the printer to execute it.

How does it print?

Printing

- Theory: All 3D printers are Additive Manufacturing (AM) machines
- The common factor between all 3D printers is that they create a model layer by layer
- The different types of printers have different ways of constructing layers and use different materials, with different end products

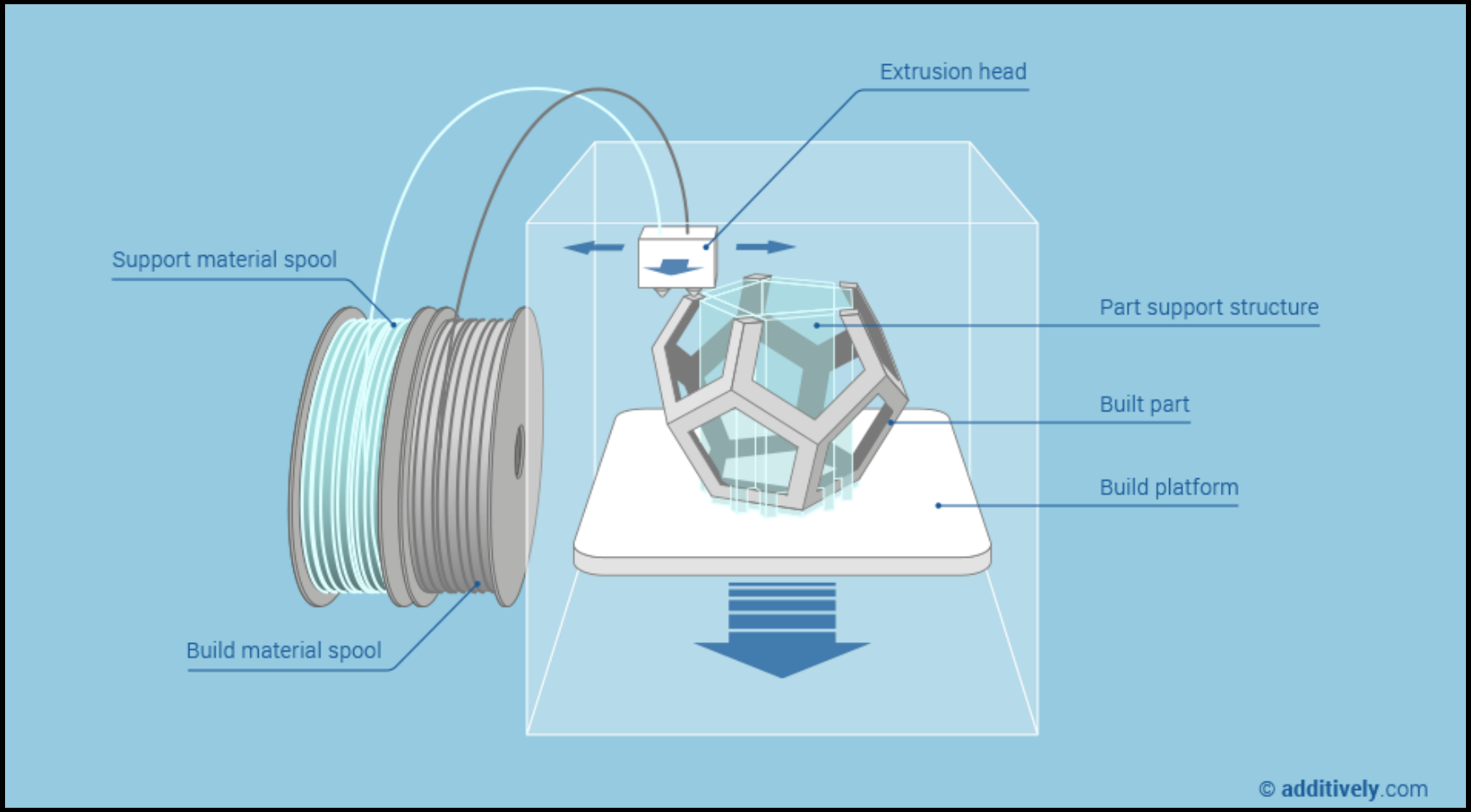
Types of Printers

- Fused deposition modeling (FDM)
- Stereolithography (SLA)
- Digital Light Processing (DLP)
- Laminated Object Manufacturing (LOM)
- Selective Laser Sintering (SLS)
- Selective Laser Melting (SLM)

Types of Printers

- Fused deposition modeling (FDM)
- Stereolithography (SLA)
- Digital Light Processing (DLP)
- Laminated Object Manufacturing (LOM)
- Selective Laser Sintering (SLS)
- Selective Laser Melting (SLM)

Fused Deposition Modeling



Fused Deposition Modeling

- https://www.youtube.com/watch?v=WHO6G67GJbM

FDM: Cartesian

- https://www.youtube.com/watch?v=ZcC7l9IH_Dw&frags=pl%2Cwn

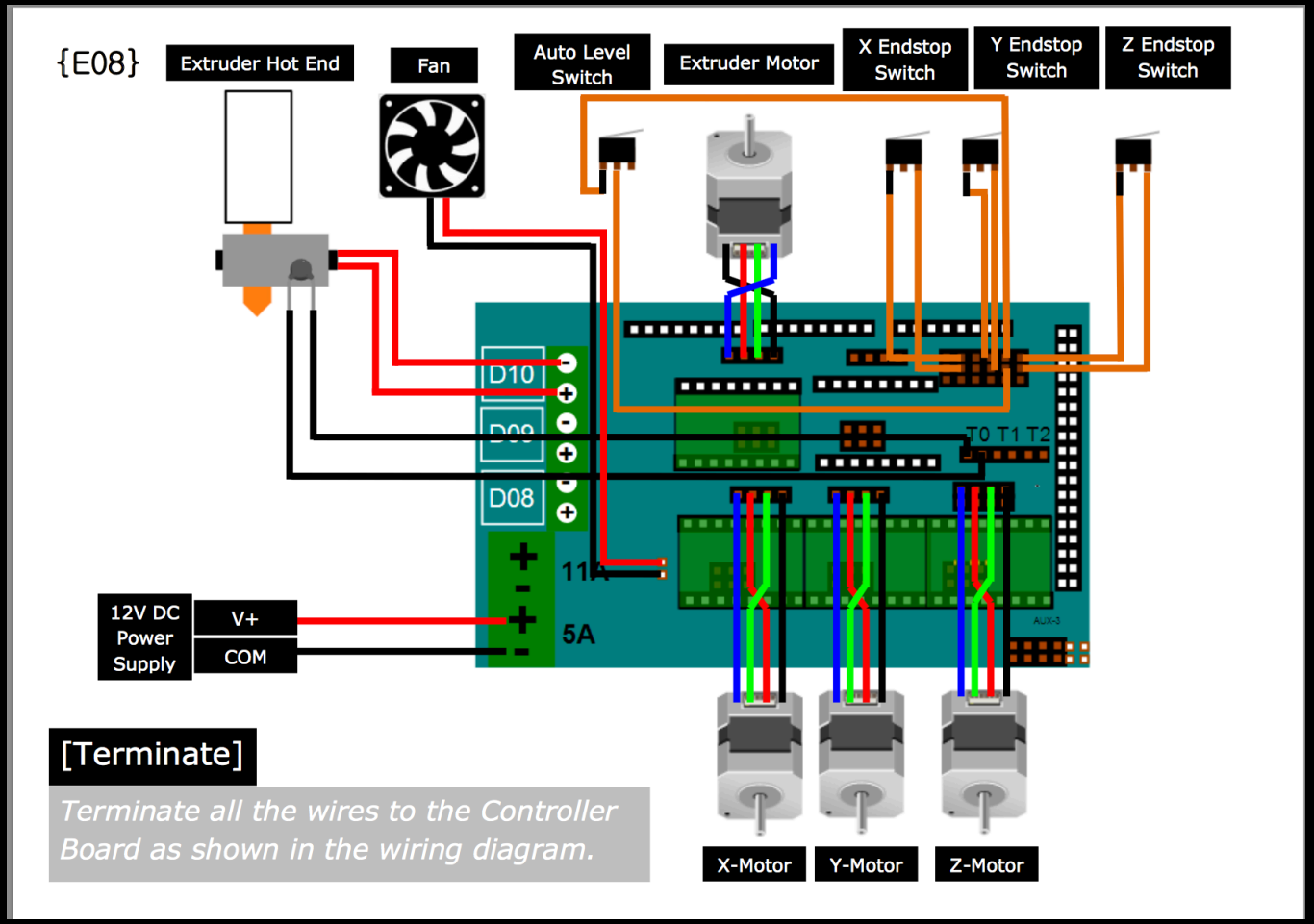
FDM: Delta

- https://www.youtube.com/watch?v=AYs6jASd_Ww&frags=pl%2Cwn

FDM: Polar

- https://www.youtube.com/watch?v=gri16vi5D78&frags=pl%2Cwn

Fused Deposition Modeling



Fused Deposition Modeling

Advantages:

- Good final thermal resistance. Can use production grade thermoplastics
- Easy to prototype
- Relatively cheap
- Can DIY

Disadvantages

- Can only do plastic

Types of Printers

- Fused deposition modeling (FDM)
- Stereolithography (SLA)
- Digital Light Processing (DLP)
- Laminated Object Manufacturing (LOM)
- Selective Laser Sintering (SLS)
- Selective Laser Melting (SLM)

Stereolithography

- One of the oldest 3D printing methods
- Involves converting liquid resin into a solid model using lasers

Stereolithography

- https://www.youtube.com/watch?v=yYGycgnYlBM

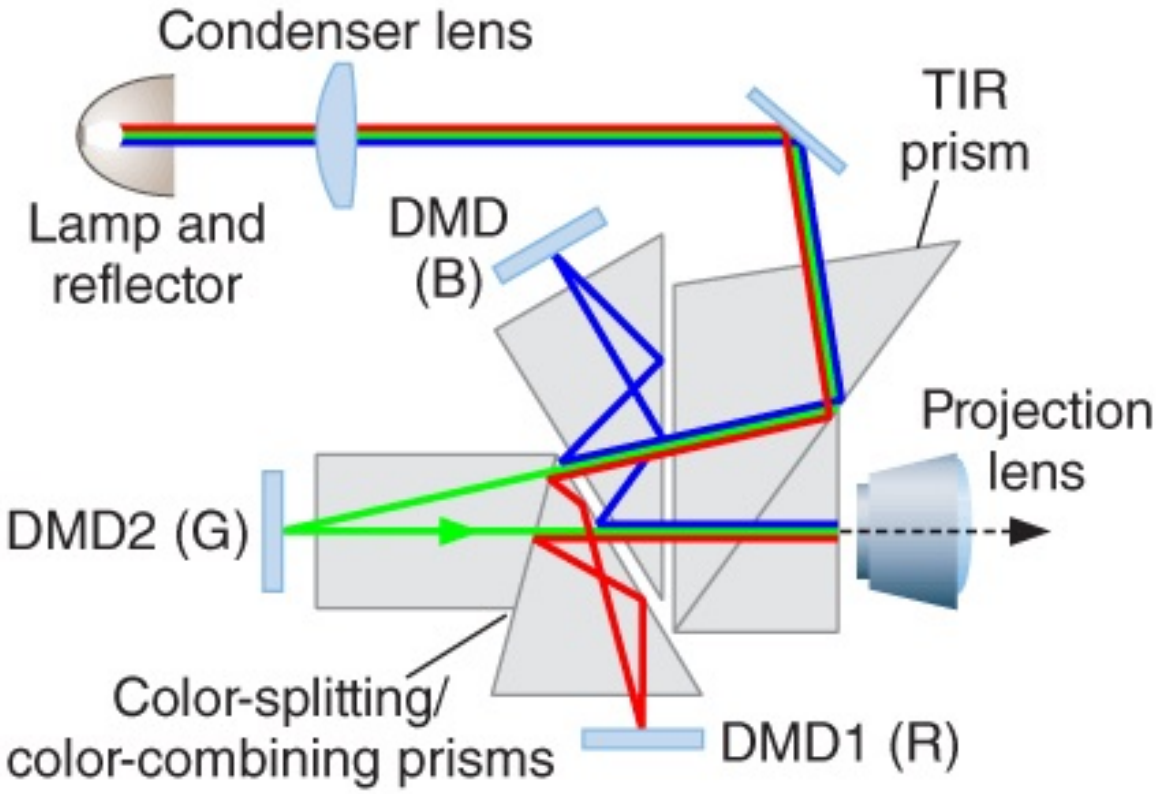
Stereolithography

- Not very commonly used
- Relatively cheap and fast
- New types can produce very strong final products for industry

Types of Printers

- Fused deposition modeling (FDM)
- Stereolithography (SLA)
- Digital Light Processing (DLP)
- Laminated Object Manufacturing (LOM)
- Selective Laser Sintering (SLS)
- Selective Laser Melting (SLM)

Digital Light Processing



Digital Light Processing

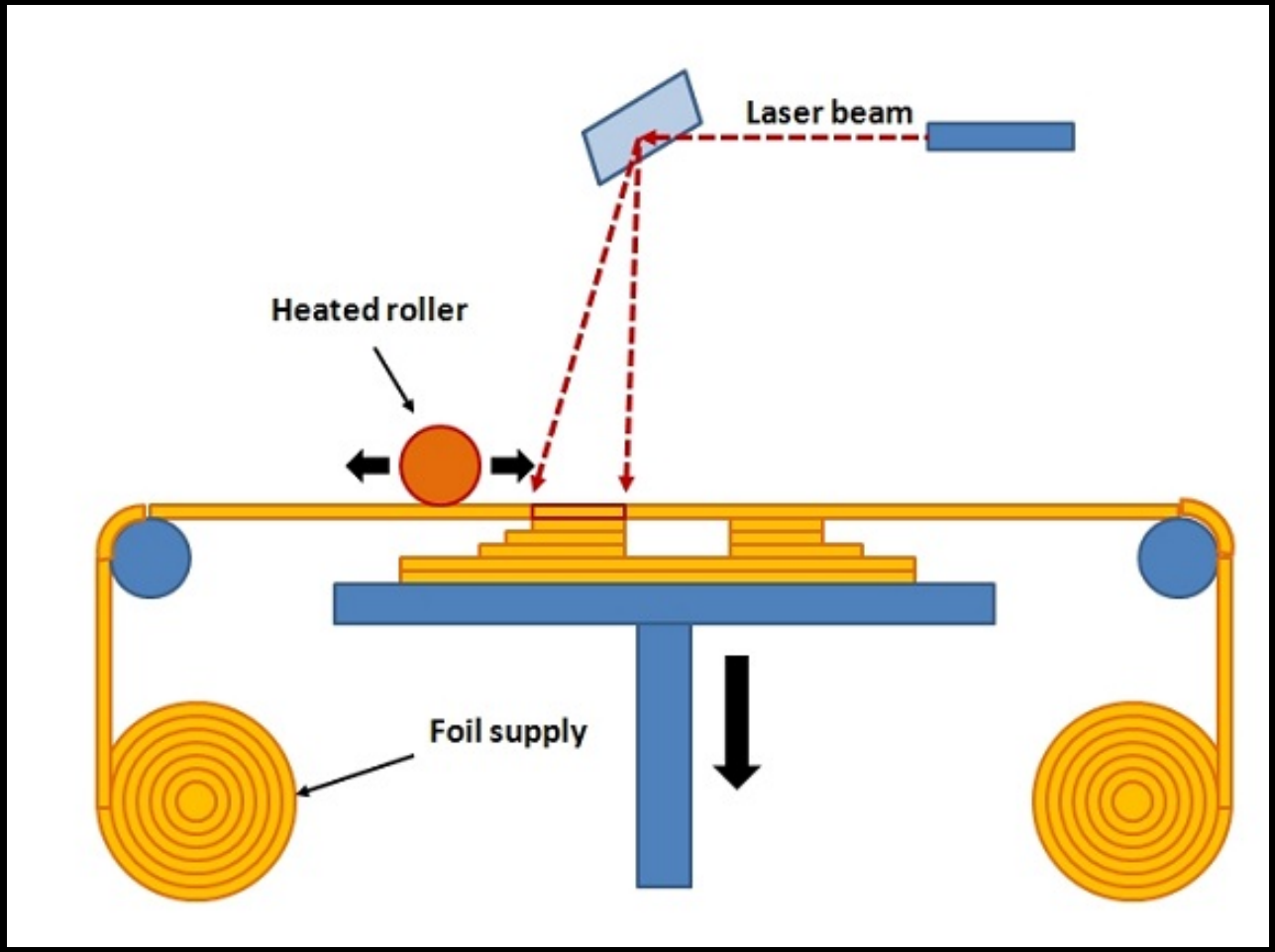
Advantages:

- Very fast speed
- Lower cost than SLA
- Robust final product
- Excellent resolution

Types of Printers

- Fused deposition modeling (FDM)
- Stereolithography (SLA)
- Digital Light Processing (DLP)
- Laminated Object Manufacturing (LOM)
- Selective Laser Sintering (SLS)
- Selective Laser Melting (SLM)

Laminated Object Manufacturing



Laminated Object Manufacturing

- https://www.youtube.com/watch?v=mwVgNqVnrew

Laminated Object Manufacturing

Laminated Object Manufaturing

Advantages:

- Extremely fast prototyping
- Relatively Cheap
- No chemical reactions needed

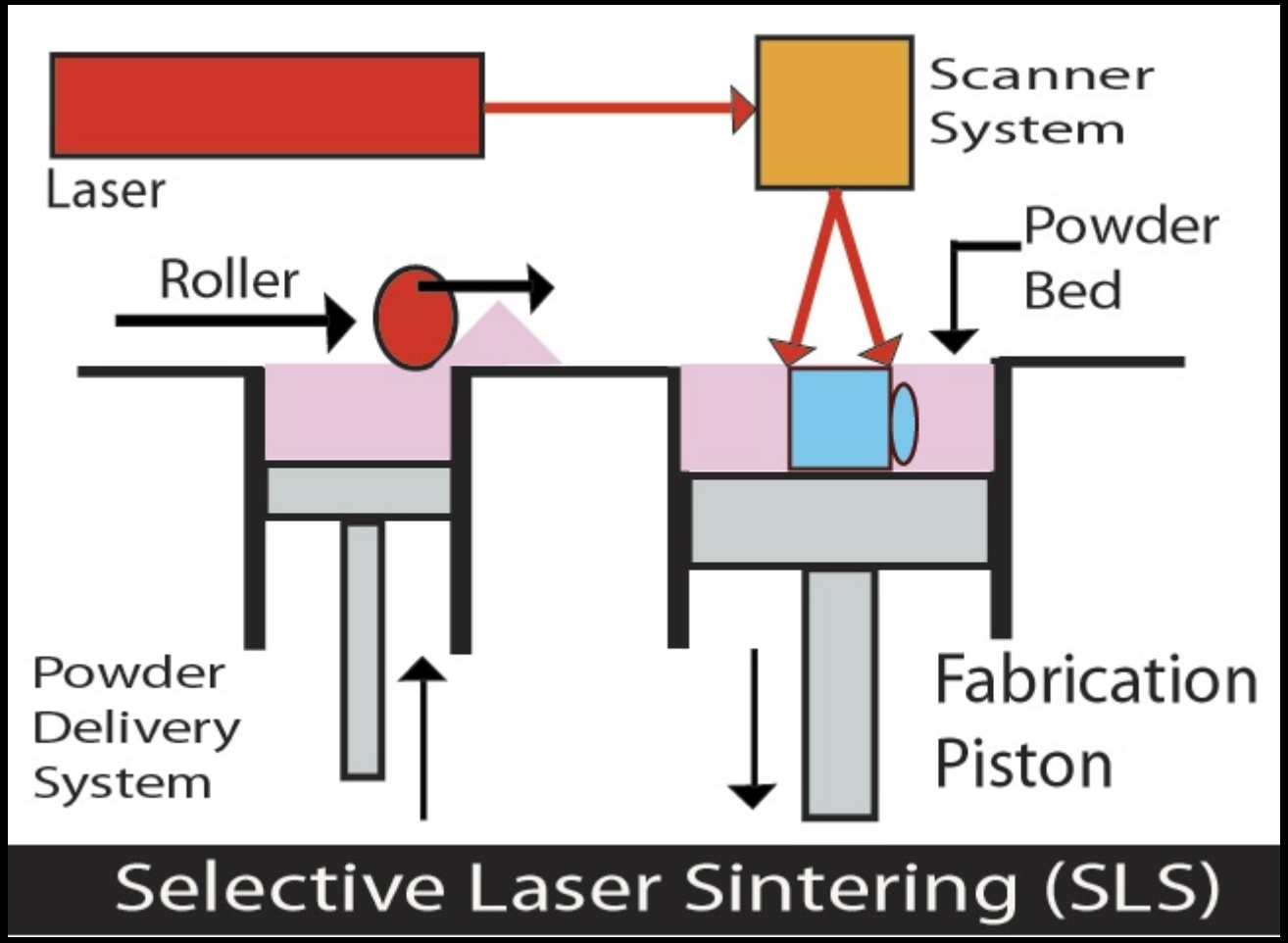
Disadvantages:

- Only one company makes it
- If you use paper, its not moisture proof
- Relatively weak product

Types of Printers

- Fused deposition modeling (FDM)
- Stereolithography (SLA)
- Digital Light Processing (DLP)
- Laminated Object Manufacturing (LOM)
- Selective Laser Sintering (SLS)
- Selective Laser Melting (SLM)

Selective Laser Sintering



Selective Laser Sintering

- https://www.youtube.com/watch?v=wD9-QEo-qDk

Selective Laser Sintering

Advantages:

- Doesn't need supports
- Can print with a wide range of materials such as metal, ceramics, glass, and nylon

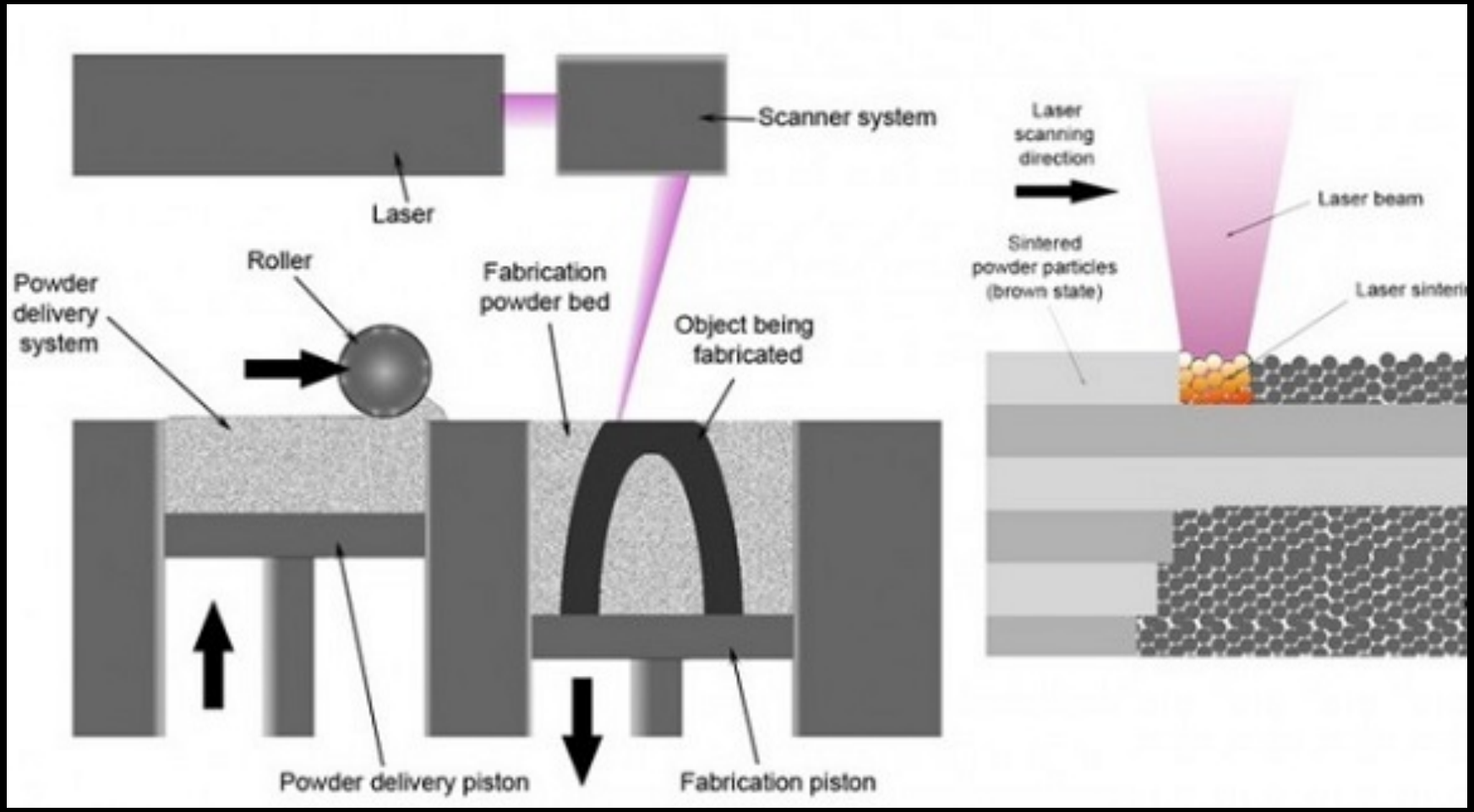
Disadvantages:

- Very expensive as it uses a high powered laser
- Used more for manufacturing than prototyping

Types of Printers

- Fused deposition modeling (FDM)
- Stereolithography (SLA)
- Digital Light Processing (DLP)
- Laminated Object Manufacturing (LOM)
- Selective Laser Sintering (SLS)
- Selective Laser Melting (SLM)

Selective Laser Melting



Selective Laser Melting

- Same as SLS, but fully melts metal layers
- Only used for pure metals and not alloys
- Much stronger final product than with SLS

CNCs

- CNCs are not strictly printers, but function in a similar way
- CNC means Computer Numerical Control
- CNCs operate by drilling or ‘lasering’ material to make cuts in it.
- Useful for the accurate machining of materials

CNCs

- CNCs also take Gcode files, although ones that are specialized for cutting
- Laser cutting is a type of CNC

Part 3:
Getting Printing Done

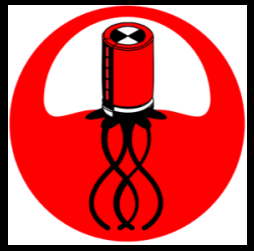
Build Your Own

- Lots of DIY options from RepRap
- Mostly FDM printers
- Can be cheaper than 10k inr if you buy cheap parts
- Mechanical parts can be 3D printed

Printing Services

- Printing services are available
- Prices depend on printing time and material cost
- Typically around 200 – 300 INR/h

Thank you!



Web: www.artscienceblr.org
Insta: artscienceblr