

# 3D-printing Course



# What is a 3D-printer

Additive manufacturing –  
depositing layer by layer to form  
a three-dimensional object.

FDM - thermoplastics deposited  
as thin strands, layer by layer

Filament led to the Extruder

Gears control the feed movement of filament

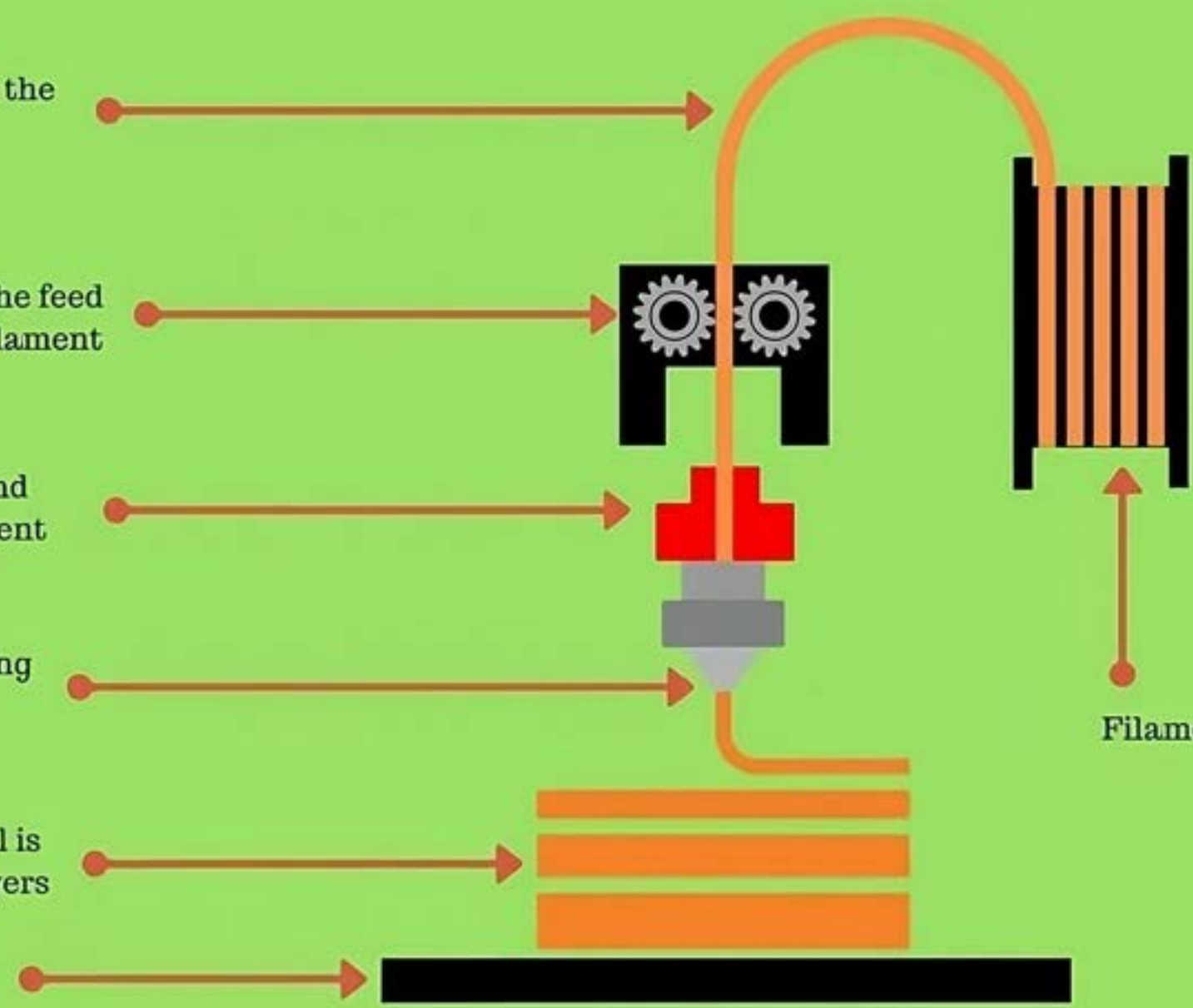
Heater heats and melts the filament

Nozzle extruding the material

Melted material is deposited in layers

FDM Print Bed

Filament Spool



# Material/filament

Material	Build plate	UV	Strength	Heat	Moisture	Support
PLA (bioplastic)	Smooth	UV breakdown	Less Elastic	Low heat resistance	Becomes Brittle and takes damage	Easy to remove
PETG (PET same as bottles)	Powder coated	-	Elastic	High Heat resistance	Resists water damage but can become stringy	Hard to remove

# Common material use cases

## PLA

- Prototypes
- Figurines

## PETG

- Mechanical Parts
- Outdoor

# Prusa

- Two types
- MK4: 25×21×21 cm
- XL: 36×36×36 cm

Prusa MK4



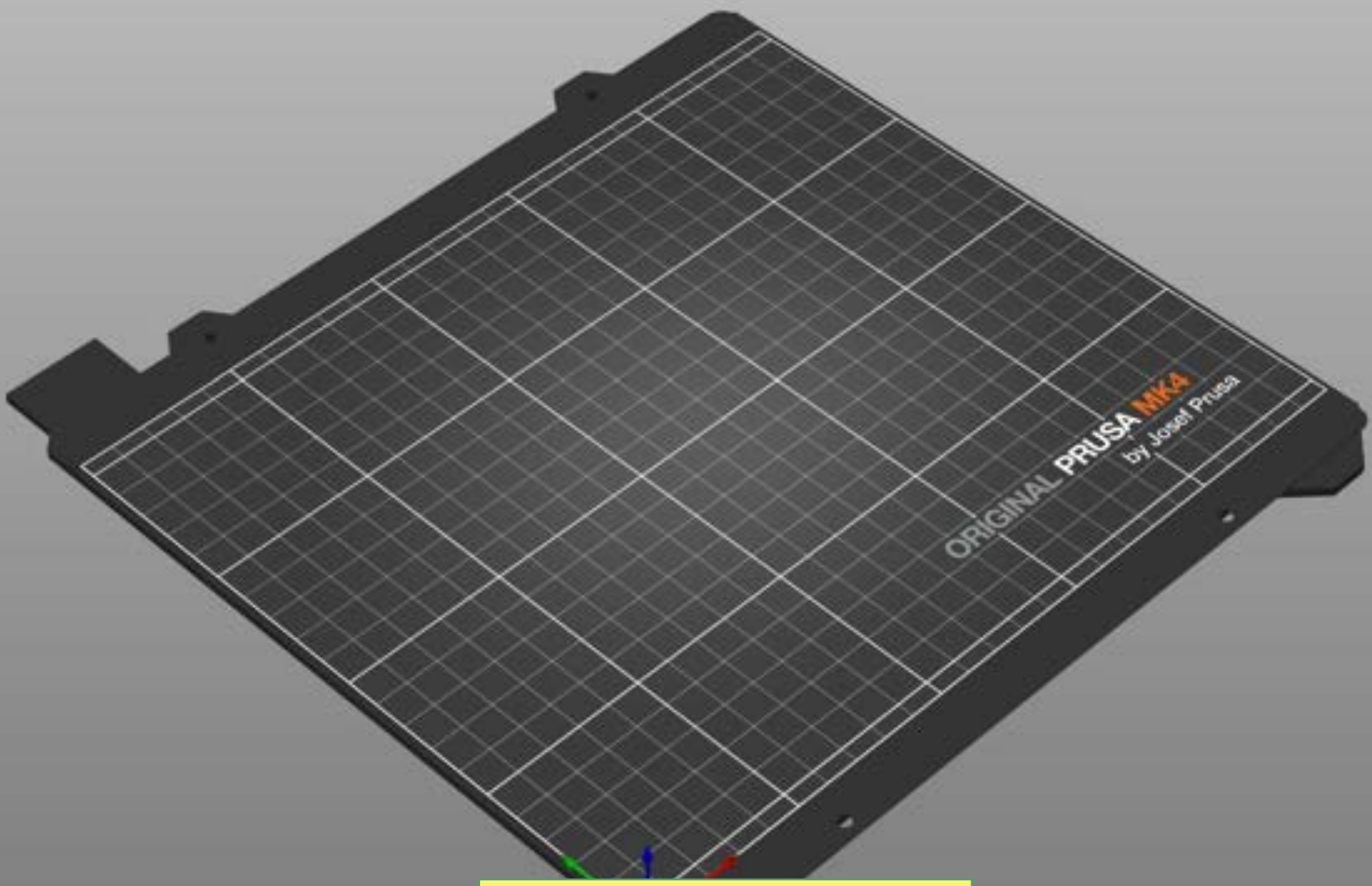
Prusa XL



# What is a Slicer?

Takes a model file (STL/3MF) and transforms it to GCODE - instructions for the printer.

There are many parameters and settings, the default settings are recommended and work in most cases.



Print settings:

0.20mm SPEED

Filament: Generic PLA

Printer: Original Prusa MK4 Input Shaper 0.4 nozzle

Supports: None

Infill: 15% Brim:

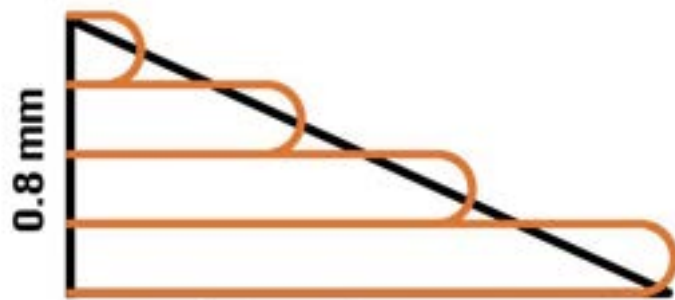
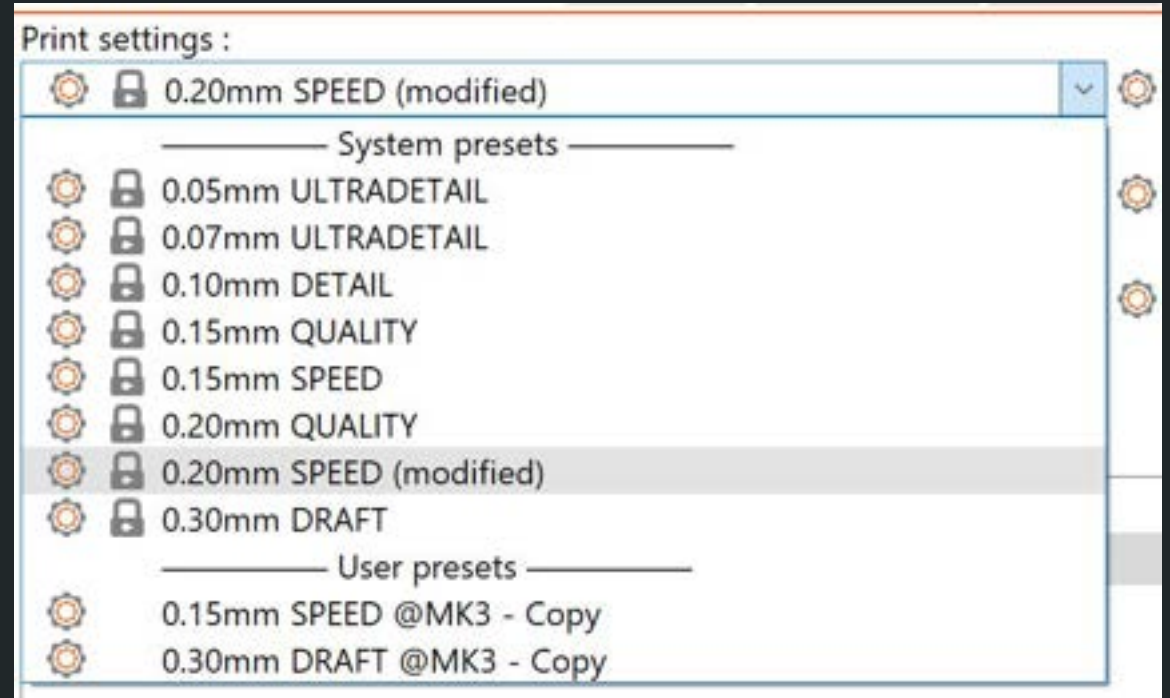
Name	Editing
------	---------

Slice now

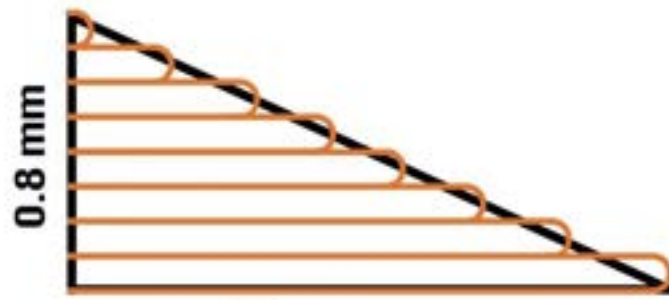
PrusaSlicer



# Layer height



0.2 mm layers



0.1 mm layers

# Infill

- Higher infill = higher strength
- Infill between 15-25% or 100% (mechanical)
- Different patterns – Different properties

## INFILL DENSITY



### Decoration: 0-15%

Objects that are only supposed to look nice don't have to be stable. Here you can save time and money with a low density.



### Standard: 15-50%

The most commonly used infill density is 20-25%. A compromise between stability, speed and filament consumption.



### Robust: 50-100%

Objects with a mechanical purpose must be sturdy. A density of 100% means a solid part.

## INFILL PATTERN

### Low Strength

Application: Figures, decorative objects.

Density: 0-15%

Patterns:



Lines



Zig Zag

### Medium Strength

Application: Standard objects

Density: 15-50%

Patterns:



Grid



Triangles



Tri-Hexagonal



Honeycomb

### High Strength

Application: Robust objects

Density: 50-100%

Patterns:



Cubic



Cubic  
Subdivision



Octet



Gyroid

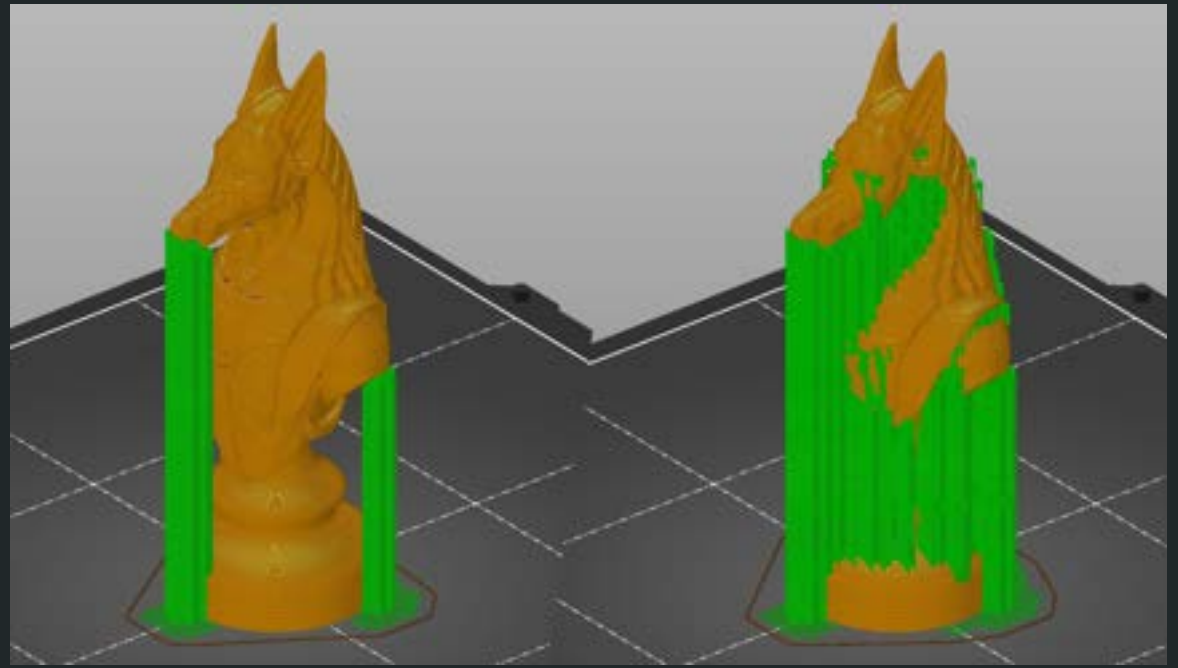
# Bridging



- Possible to print in air. (Between points)
- Can create deformations
- Avoid bridging if possible (orientate model in slicer)

# Supports

- Supports which are later removed
- Used when overhangs are unavoidable in the model
- Automatically generated by the slicer



Print settings :

0.20mm SPEED (modified)

Filament : Prusament PLA

Printer : Original Prusa i3 MK3

Supports: None

Infill: 10%

Name	Support type
Body2	Support on build plate only
	For support enforcers only
	Everywhere

# Preview

- A breakdown of the print
- Analysing before printing
- Each feature represented as a color
- Time and filament used per feature

Feature type	Time	Percentage	Used filament
Perimeter	10m	8.3%	1.04 m 3.11 g
External perimeter	16m	13.5%	1.43 m 4.27 g
Overhang perimeter	6s	0.1%	0.01 m 0.02 g
Internal infill	5m	4.4%	0.61 m 1.82 g
Solid infill	8m	6.9%	0.80 m 2.40 g
Top solid infill	4m	3.1%	0.30 m 0.91 g
Bridge infill	4m	3.0%	0.25 m 0.75 g
Gap fill	3m	2.8%	0.06 m 0.17 g
Skirt/Brim	21s	0.3%	0.02 m 0.07 g
Support material	59m	50.7%	2.04 m 6.09 g
Support material interface	8m	6.7%	0.16 m 0.47 g
Custom	12s	0.2%	0.02 m 0.06 g

Estimated printing times [Normal mode]:

First layer: 5m  
Total: 1h57m

Show stealth mode

# Warping

## Symptoms

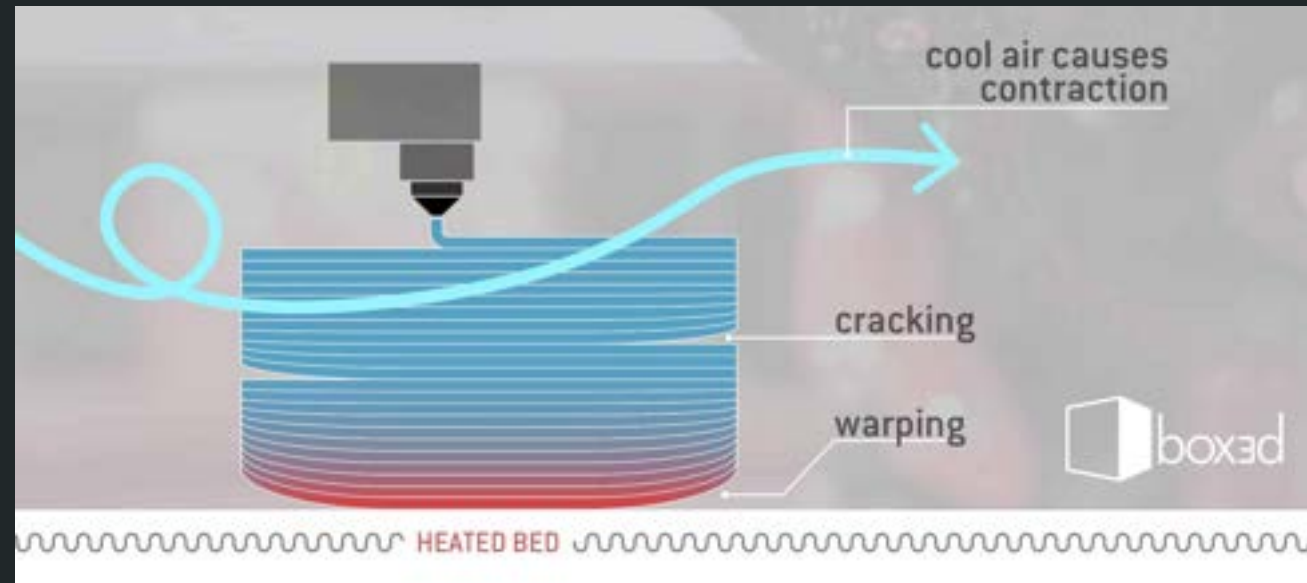
- Corners lift from bed
- Best Case: bottom is twisted
- Worst Case: Print detaches from plate

## Cause

- The plastic expands and contracts with temperature changes.
- If adhesion between bed and print is bad the print will detach

## Solutions

- Clean build plate with “T-Röd”(denatured alcohol)
- Rounded corners



# To think about when designing and orienting model in software

- Avoid bridging over large gaps
- Complex shapes may need support
- Design with the printing process in mind
- Layer orientation
- You need less infill and support than you think
- Tolerances can vary (!)

# Export G-CODE

---

- Export to USB. There is one per printer.
- The filename contains print information. Double-check before starting print.