School of Engineering
Institute of Materials
Laboratory of Macromolecular
and Organic Materials (LMOM)

Polymer Science (MSE-360)

3D Printing

Semester 2025/26

Assistant:

Giada Romano (LAFT, Neuchâtel)

giada.romano@epfl.ch

Objective

- Understand the basics of 3D printing through Fusion Deposition Modelling (FDM) concept
- Learn how the printing parameters affect the final mechanical properties of the printing object
- Design and realize a beam with the best mechanical properties

Preparation Before the Lab Session

The lab session lasts three hours. To make the most of it, prepare yourselves in advance! Read this protocol and follow the instructions carefully so you will have more time to enjoy printing and analyzing as many parts as possible.

- Download the Prusa Slicer software as described below. Explore the software to see where the printing parameters can be adjusted.
- Watch this video explaining the basics of FDM - 3D printing technique : DLL-MAT Impression 3D par dépose de polymères fondus (FDM) - EPFL - Ecole polytechnique fédérale de Lausanne
- The goal is to print as many beams as possible. To do this, optimize your time and remember that mechanical testing and printing can be done in parallel.
- Think about which parameters you want to adjust to optimize the beam's mechanical properties and why, so as you will have more time to test and verify your assumptions.

Introduction

Additive manufacturing (AM), commonly referred to as **3D printing**, encompasses a set of computer-controlled technologies that build 3D objects layer by layer (Figure 1). This approach enables the fabrication of complex geometries that are often inaccessible or prohibitively expensive with conventional subtractive techniques (e.g. milling, turning, drilling) or forming processes (e.g. injection molding, extrusion).

According to the ISO/ASTM classification (ISO/TC 261 and ASTM F42), additive manufacturing is divided into seven categories: vat photopolymerization, material jetting, binder jetting, material extrusion, powder bed fusion, sheet lamination, directed energy deposition.

Among them, **material extrusion** is the most widespread for polymers. It relies on the controlled deposition of a thermoplastic filament, which is softened or molten and extruded through a nozzle. The most widely adopted process in this category is **Fused Deposition Modeling** (FDM), also known as **Fused Filament Fabrication** (FFF).

Polymers printed by FDM must meet specific rheological and thermal requirements: the material must soften and flow under heat, solidify rapidly upon cooling, and adhere both to the substrate and to previously deposited layers. Consequently, only a limited subset of thermoplastics (e.g. PLA, ABS, PETG, polyamides, polycarbonate) are commonly used, sometimes requiring pre-processing (e.g. drying of hygroscopic polymers).

The goal of this TP is to get an overview of AM processes, familiarization with the FDM technique (principle, process parameters, influence on part quality), and experimental optimization of the bending performance of a printed beam by adjusting materials and process parameters.

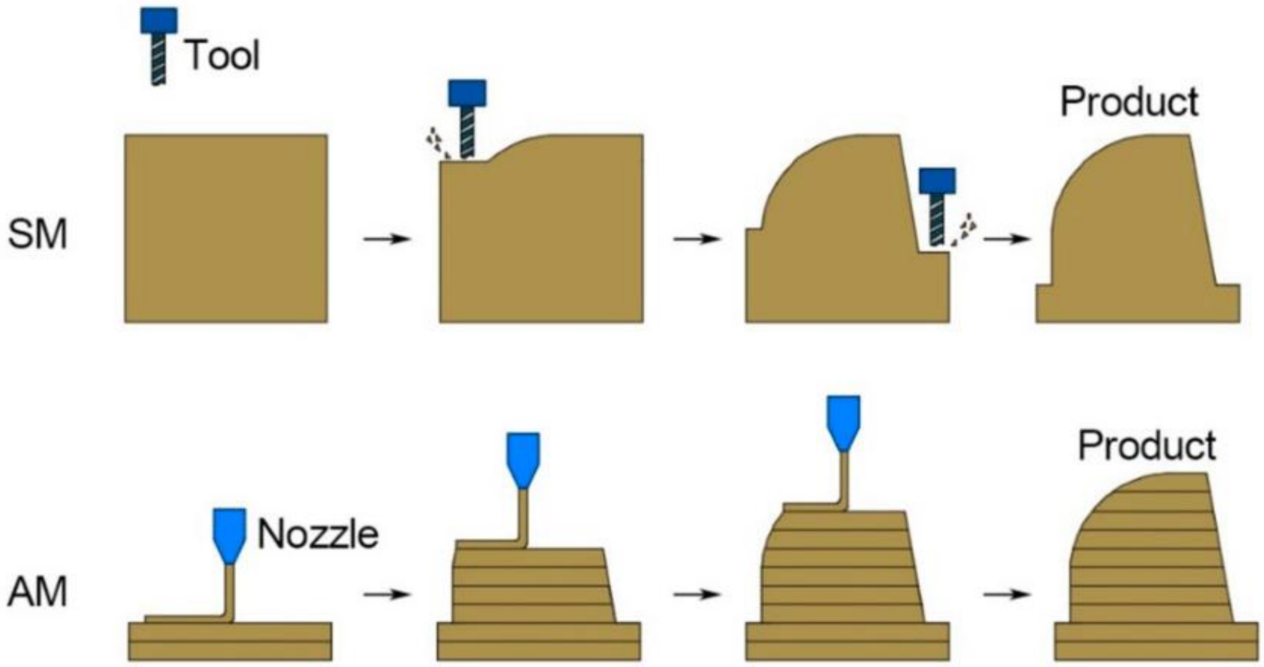


Fig. 1. Milling versus additive manufacturing [ref].

Fused Deposition Modeling (FDM)

Fused Deposition Modeling (FDM) is perhaps the best-known 3D printing technology to the general public. Its popularity stems from the low cost of machines and materials, the ease of use, and extensive media coverage. Despite its reputation as a “modern” technology, FDM is actually relatively old: the first patent was filed in 1989 by Stratasys. When the patent expired in 2009, the simultaneous democratization of affordable microcontrollers (e.g. Arduino boards) triggered a dramatic drop in cost. Since then, open-source communities have enabled the development of reliable 3D printers that can be built for only a few hundred Swiss francs.

The principle of FDM is straightforward (Figure 2). A **thermoplastic filament** is melted and extruded through a heated nozzle (typically 0.4–0.5 mm in diameter). The molten filament is deposited in thin, adjacent lines that rapidly solidify upon cooling. By repeating this process, layer by layer, the final three-dimensional part is constructed. Commercial filaments are produced by extrusion under controlled conditions to ensure a continuous, homogeneous cylindrical shape with standard diameter of either 1.75 mm or 2.85/3.00 mm.

To fabricate a part, three digital steps are required:

- a CAD model of the created geometry (or downloaded).
- the model is saved as an STL file, which encodes the part's surface geometry.
- the STL is processed by a slicer software, which generates G-code printing instructions to be executed by the printer.

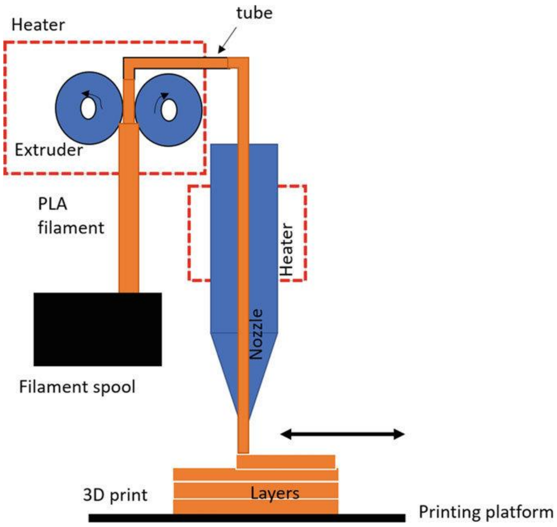


Fig. 2. Principles of fused deposition modeling (FDM)

STL files

The STL ("Standard Triangle Language" or "Standard Tessellation Language") format describes the external geometry of an object by approximating its surfaces with a mesh of triangles (Figure 3). STL files are unitless (millimeters are assumed by default but must always be checked). They contain geometrical surface data, but don't contain any information no material, color, or scale. High resolution (many small triangles) increases file size significantly. Because STL only stores surface geometry and quickly grows in size for higher accuracy, it is best used as an intermediate format prior to slicing rather than for long-term storage of CAD models.

Slicing

"Slicing" converts a CAD geometry (STL) into a stack of layers and generates a **G-code** file, i.e. the machine-readable set of instructions for the printer (Figure 3). In this TP, you will use **PrusaSlicer**.

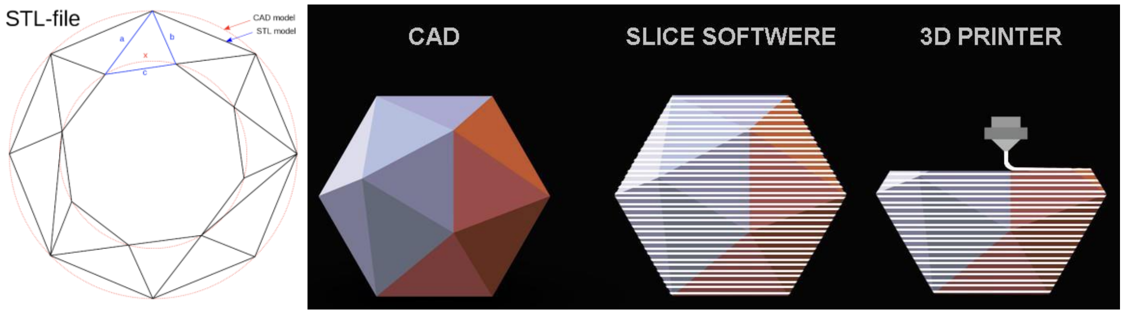


Fig. 3. Left: a CAD representation of a torus (shown as two concentric red circles) and an STL approximation of the same shape (composed of triangular planes). Source: Wikipedia.org. Right: Slicing of a CAD model into layers.

G-code

G-code stands for "Geometric code" and describes toolpaths and process parameters. Each command specifies the type of motion (e.g. G01 = linear move), the spatial coordinates (X, Y, Z), extrusion length (E), and feed rate (F). For example:

```
G01 X247.951560 Y11.817060 Z-1.000000 F400.000000
```

FDM Parameters

FDM part quality and mechanical performance are strongly dependent on process parameters. These can be grouped into:

Machine parameters:

- **deposition orientation**: the build orientation determines how layers are stacked relative to the loading direction. Different orientations lead to distinct cross-sectional structures at identical part locations. The deposition angle θ represents the inclination of the deposited filaments with respect to the reference axis.
- **layer thickness (resolution):** This is the height of each deposited layer. It depends on nozzle size and process settings. Thicker layers reduce printing time but may decrease interlayer bonding quality and surface finish.
- **deposition style (infill pattern):** The raster pattern fills the interior of the part according to its cross-section outline. Three typical styles may be distinguished: long-raster (0°), long-short raster (0°/90° alternating), staggered raster (+45°/-45° alternating). The raster angle corresponds to the angle between the infill lines and the x-axis. Raster style affects anisotropy, stiffness, and failure modes.
- **raster width:** the physical width of individual extruded filaments. Wider rasters increase deposition overlap but may lead to inhomogeneous cooling and internal stress.
- **raster gap:** the spacing between adjacent rasters in the same layer. A positive gap creates voids and reduces density, while zero or negative gaps improve consolidation at the expense of printing accuracy.

- extrusion temperature and platform temperature,
- extrusion and head speed,
- flow rate, air gap.

Material parameters:

- thermal properties (T_g, crystallinity, heat capacity, thermal conductivity)
- physical properties (density, crystallinity)
- chemical (polarity, moisture sensitivity, degradation behavior)
- mechanical (modulus, strength, toughness)

For this TP, we will focus on beam fabrication and study the influence of geometry and raster orientation on bending performance.

Experimental Part

The experiment is about 3D-printing of beams and their mechanical testing in 3-point bending. Your goal is to optimize their bending resistance, for which you have approximately 3 hours. The remaining time will allow you to begin the report preparation.

Procedure

1. Preparation:
    - work in pairs with 1 PC and 1 Prusa FDM printer.
    - install **PrusaSlicer** (free, www.prusa3d.com/prusaslicer/).
    - review the appendix and answer the preparation questions before starting.
2. Design and Printing
    - start from the STL geometries provided on Moodle.
    - adjust parameters in PrusaSlicer: beam cross-section, infill density and pattern, wall thickness, orientation, etc.
    - ensure part **mass is below 2.5 g** (check in PrusaSlicer) and beam **length is at least 10cm**.
    - export G-code, transfer to SD card, and print (10–20 minutes per beam).
    - print several iterations if possible.
3. Mechanical Testing
    - perform a **3-point bending test** using the manual setup (load cell + dial gauge).
    - measure **deflection vs. applied load** and the **breaking force.**
    - carefully note beam geometry, process parameters, and test results.

Mechanical Characterization

For each printed beam, proceed as follows:

1. *deflection measurement* (Figure 6)
    - switch on the dynamometer and zero the force reading.
    - lift the loading module, place the beam in position, and adjust the height of the dial gauge so that the probe gently touches the bottom surface of the beam.
    - carefully lower the loading module onto the beam
    - record the applied load (force) and the corresponding deflection (displacement).
2. *fracture test* (breaking force)
    - lower the dial gauge as far as possible to prevent damage during fracture.
    - set the dynamometer to "PEAK" mode (press and hold the key for several seconds until "max" appears)
    - grasp the handle firmly and apply load gradually until the beam breaks.
    - note the maximum force displayed.

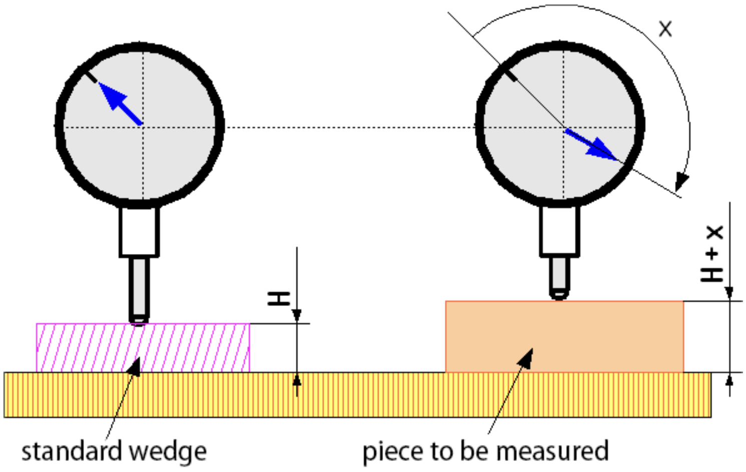
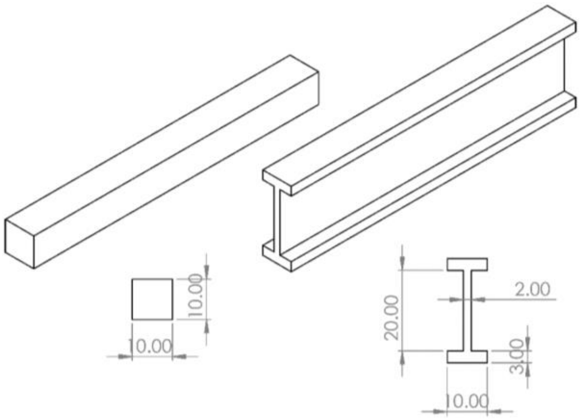


Fig. 6. Measuring principle using a dial gauge. The zero point is established using a gauge block. Source: Wikipedia.

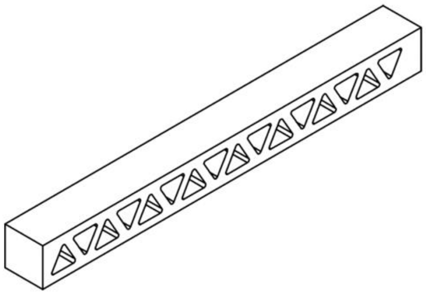
Preparation Questions

Before the lab, answer the following using the appendix and literature:

1. Introduce FDM for 3D printing and highlight its main properties such as functioning principle, materials used, resolution (µm), advantages, and disadvantages.
2. Which polymer families can (or cannot) be processed by FDM? Why?
3. Considering ABS as a printing material: what property explains why ABS is difficult to print?
4. Compute the second moment of area and cross-sectional area for the two proposed beam geometries. Which is more resistant to bending? Consider this factor to justify the best geometry to print.



5. Analyze the “unusual” beam geometry shown below. Is it suitable for FDM? Why or why not?



Concluding Questions

After the lab, answer the following:

1. Discuss how printing parameters, such as filling pattern, infill percentage, raster angle (or filament orientation) and speed affect the final properties of the part.
2. Describe the physical phenomena experienced by the polymer during FDM. How do they affect final mechanical properties?
3. How can you speed up the printing process? Which parameters can you modify, and how will they affect the quality of the beam?
4. How could you modify the material to further enhance the mechanical properties of the printed part?
5. What should you consider when printing with a different material?

Report Guidelines:

Submit a pair report (in English) within 14 days after your lab session. Your report should be a structured analysis, not just a list of results. The reader should be guided through the experiment you ran, giving context and justification for the work you have done. Think about the story you want to tell and follow the structure given below.

3. **Introduction**: context, objectives, challenges of the lab.

4. **Theoretical background**: explain FDM, its benefits/limitations, and its impact on material properties. Briefly compare with traditional manufacturing and with other AM techniques (if known).
5. **Methodology**: describe how you prepared, designed, and carried out the experiments. Include answers to the *Preparation Questions* to justify your choices in terms of geometry of your parts' structure. You do not need to write the answers as a list, rather you should include them in the story you want to tell. They will guide you very smoothly to complete this section.
6. **Results and discussion**: present bending and force test results, possibly photographs of the printed beams, critique accuracy, analyse parameter choices vs. expectations, and identify the most influential parameters. In this part, you should include answers to the *Concluding Questions*. Again, consider integrating the answer in the paragraph as part of the story you are telling. There is no need to list them. The answers should be naturally addressed by writing your comments about the results.
7. **Conclusion**: summarize insights gained and broader relevance.
8. **References**: cite sources when used.

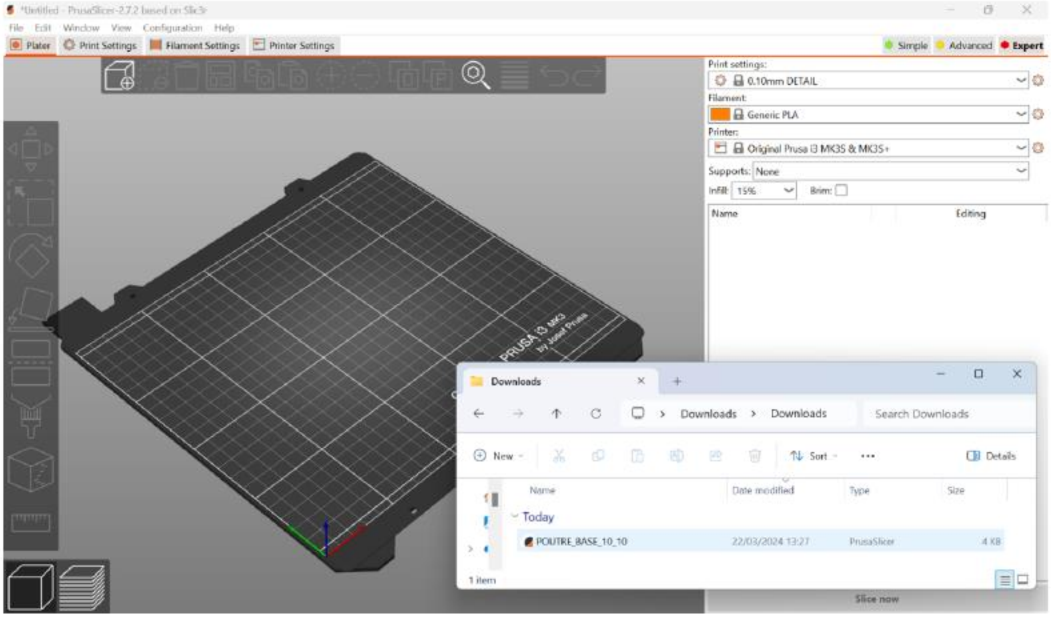
Appendix

Guidelines to use PrusaSlicer:

The software PrusaSlicer allows you to modify and print beams. Here you can find some easy steps to guide you through the software.

1. Download and install PrusaSlicer (free) from www.prusa3d.com/prusaslicer/.

   The printer model is "Original Prusa i3 MK3S & MK3S+". Please, be sure to select the correct one.
2. Import your STL file into the slicer.

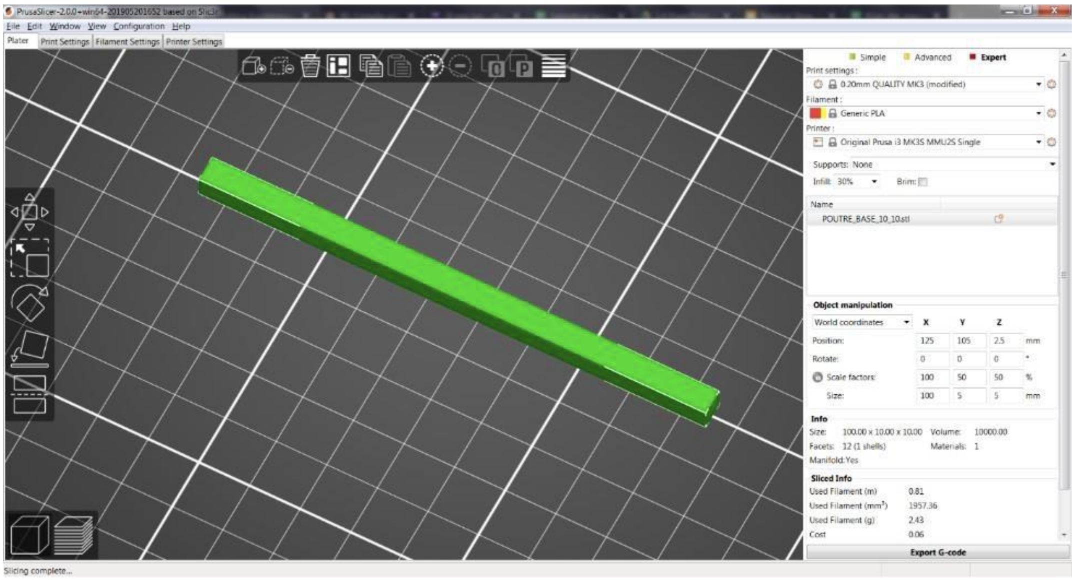


*Untitled - PrusaSlicer-2.7.2 based on Slic3r

Print settings: 0.10mm DETAIL
Filament: Generic PLA
Printer: Original Prusa i3 MK3S & MK3S+
Supports: None
Infill: 15%
Brim: ☐

Name	Date modified	Type	Size
POUTRE_BASE_10_10	22/03/2024 13:27	PrusaSlicer	4 KB

3. Adjust parameters to optimize the beam:
    a. material, section, and orientation: in the “Plater” tab, unlock “scale factors” to modify section dimensions independently of length.
    b. wall and layer thickness: “print settings” > “Layers and Perimeters”
    c. infill: “Print settings” > “Infill”



Print settings: 0.20mm QUALITY MK3 (modified)
Filament: Generic PLA
Printer: Original Prusa i3 MK3S MMU2S Single
Supports: None
Infill: 30%
Brim: ☐

Name: POUTRE_BASE_10_10.stl

Object manipulation

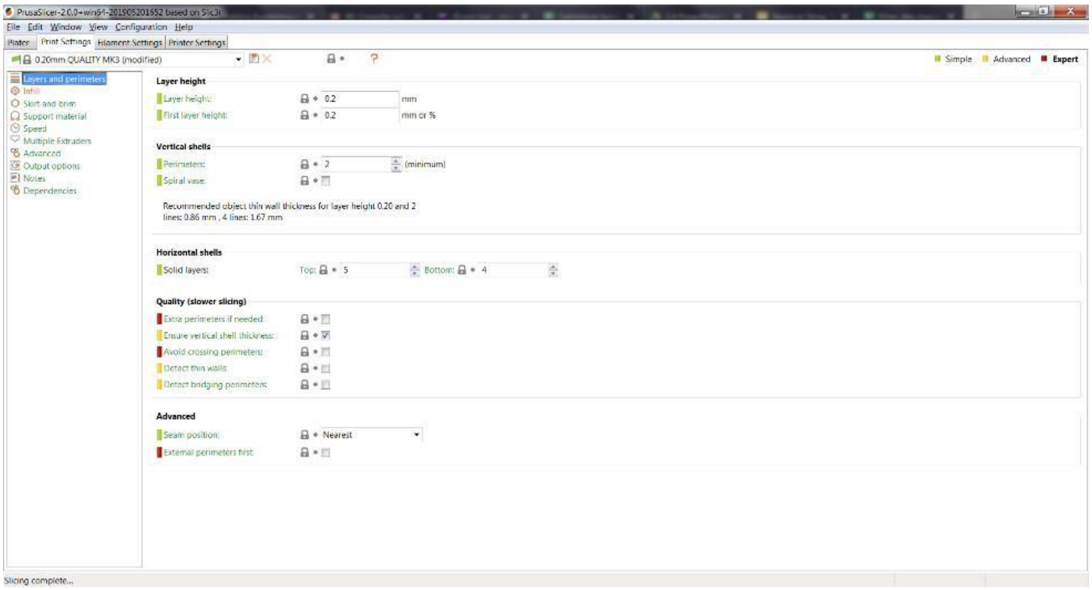
World coordinates

	X	Y	Z	
Position:	125	105	2.5	mm
Rotate:	0	0	0	°
Scale factors:	100	50	50	%
Size:	100	5	5	mm

Info
Size: 100.00 x 10.00 x 10.00
Volume: 10000.00
Facets: 12 (1 shells)
Materials: 1
Manifold: Yes

Sliced Info

Used Filament (m)	0.81
Used Filament (mm³)	1957.36
Used Filament (g)	2.43
Cost	0.06



0.20mm QUALITY MK3 (modified)

Layer height
Layer height: 0.2 mm
First layer height: 0.2 mm or %

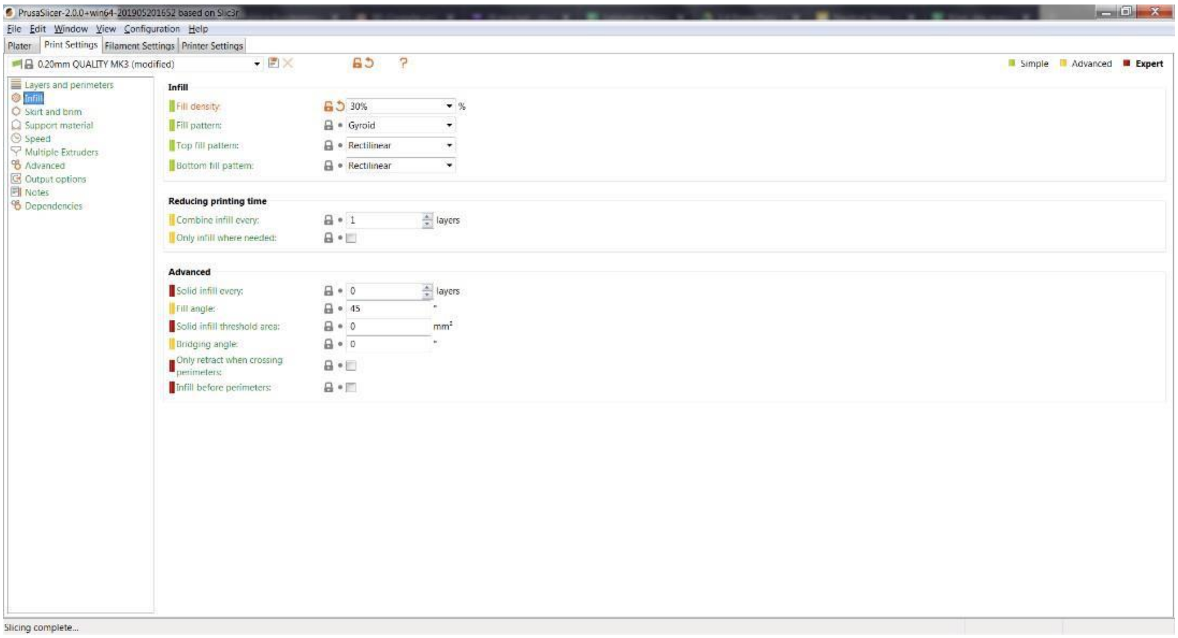
Vertical shells
Perimeters: 2 (minimum)
Spiral vase: ☐

Recommended object thin wall thickness for layer height 0.20 and 2 lines: 0.86 mm , 4 lines: 1.67 mm

Horizontal shells
Solid layers: Top: 5 Bottom: 4

Quality (slower slicing)
Extra perimeters if needed: ☐
Ensure vertical shell thickness: ☑
Avoid crossing perimeters: ☐
Detect thin walls: ☐
Detect bridging perimeters: ☐

Advanced
Seam position: Nearest
External perimeters first: ☐



0.20mm QUALITY MK3 (modified)

Infill

Fill density: 30%
Fill pattern: Gyroid
Top fill pattern: Rectilinear
Bottom fill pattern: Rectilinear

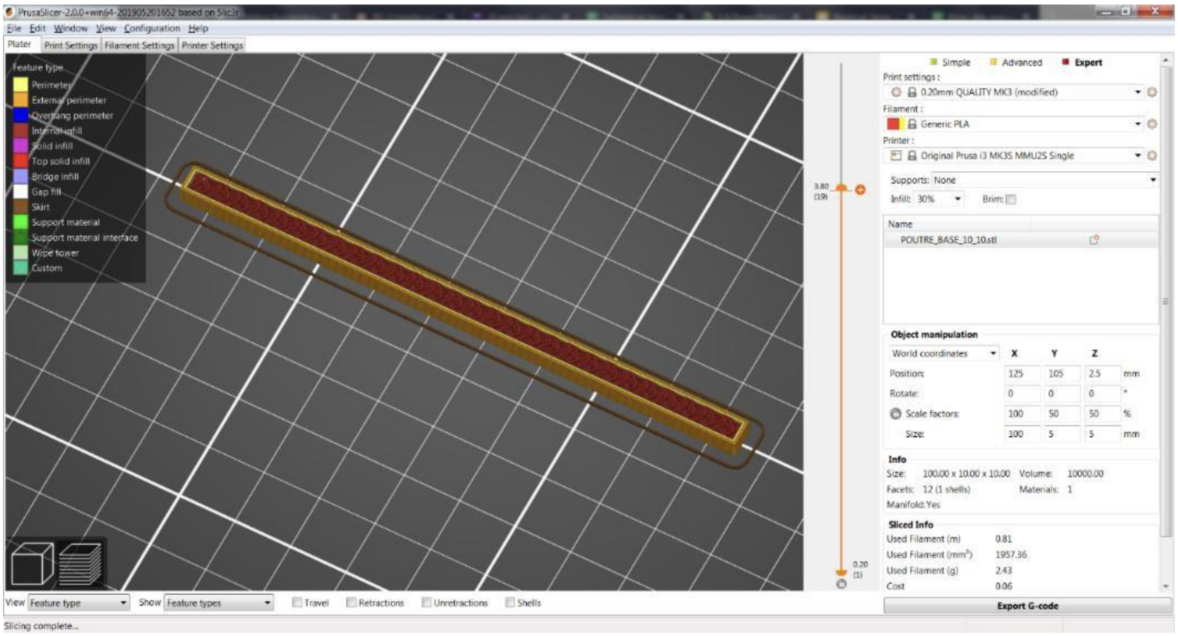
Reducing printing time

Combine infill every: 1 layers
Only infill where needed: ☐

Advanced

Solid infill every: 0 layers
Fill angle: 45 °
Solid infill threshold area: 0 mm²
Bridging angle: 0 °
Only retract when crossing perimeters: ☐
Infill before perimeters: ☐

4. After each modification, click “Slice now” (bottom right). Use the Preview mode to visualize layer-by-layer deposition. Check the estimated filament weight (bottom right); beams must remain below 2.5 g to obtain low printing time.



Print settings: 0.20mm QUALITY MK3 (modified)
Filament: Generic PLA
Printer: Original Prusa i3 MK3S MMU2S Single
Supports: None
Infill: 30% Brim: ☐

Name
POUTRE_BASE_10_10.stl

Object manipulation

World coordinates	X	Y	Z	
Position:	125	105	2.5	mm
Rotate:	0	0	0	°
Scale factors:	100	50	50	%
Size:	100	5	5	mm

Info

Size: 100.00 x 10.00 x 10.00 Volume: 10000.00
Facets: 12 (1 shells) Materials: 1
Manifold: Yes

Sliced Info

Used Filament (m)	0.81
Used Filament (mm³)	1957.36
Used Filament (g)	2.43
Cost	0.06

5. Once satisfied, click “Export G-code” in the bottom right corner, and save the file to the 3D printer’s SD card.
6. Print the beam and perform the mechanical test as described above.

Additional Documents

- W. Wu, P. Geng, G. Li, D. Zhao, H. Zhang, and J. Zhao. Influence of Layer Thickness and Raster Angle on the Mechanical Properties of 3D-Printed PEEK and a Comparative Mechanical Study between PEEK and ABS. *Materials* **2015**, *8*, 5834–5846.
- Avinson. How Long Does 3D Printing Take?. 3 Space, Sept. 20, **2018**. https://3space.com/how-long-does-3d-printing-take/ (accessed Sept. 30, 2025).
- 9T Labs. Revolutionizing Motorsports with Additive Fusion Technology™. https://www.9tlabs.com/use-cases/automotive-brackets (accessed Sept. 30, 2025).
- 9T Labs. Reshaping Aircraft Structural Parts Manufacturing. https://www.9tlabs.com/use-cases/door-hinge (accessed Sept. 30, 2025).
- EPFL. Using 3D printing to make prosthetics for Colombia war victims. https://actu.epfl.ch/news/using-3d-printing-to-make-prosthetics-for-colomb-6/ (accessed Sept. 30, 2025).
- MIT News. 3-D printing offers new approach to making buildings. April 26, **2017**. https://news.mit.edu/2017/3-d-printing-buildings-0426 (accessed Sept. 30, 2025).
- Sculpteo. Let's 3D print for museum conservation! https://www.sculpteo.com/blog/2018/09/14/lets-3d-print-for-museum-conservation/ (accessed Sept. 30, 2025).

Supplementary documents discussing the factors influencing 3D printing quality (e.g. materials selection, filling pattern, orientation, etc.) are available on the Moodle page.