

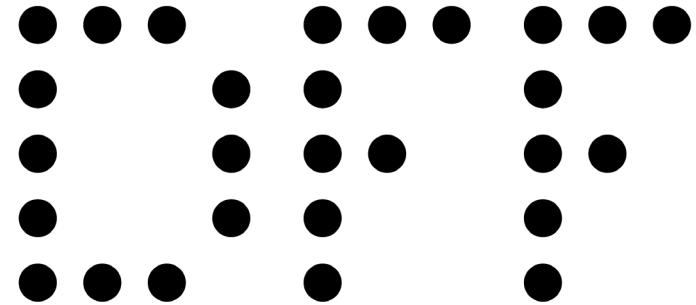
1.4

3D

print

self-check

3D printing self check for exported models



Digital Fabrication Facilities
for Architecture



Contents

INTRO

SETUP

MODEL SIMULATION

EXPORT

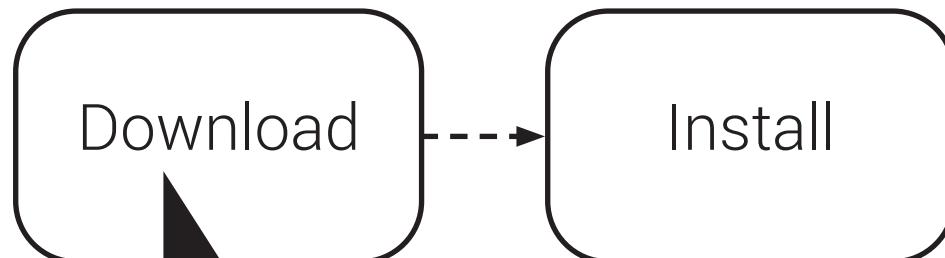
Q: What does this do?

A:

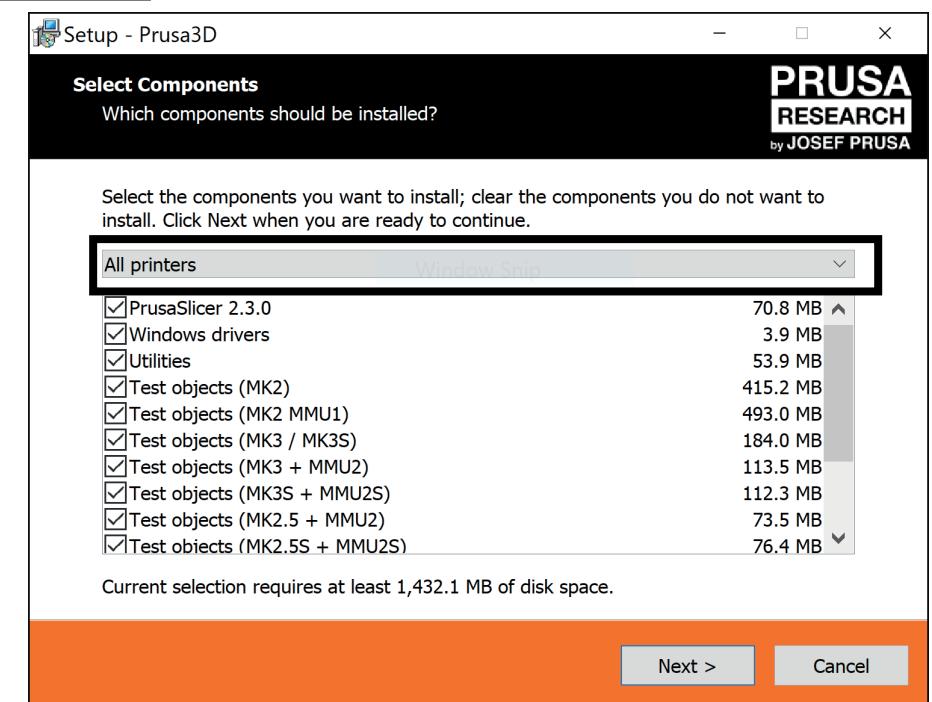
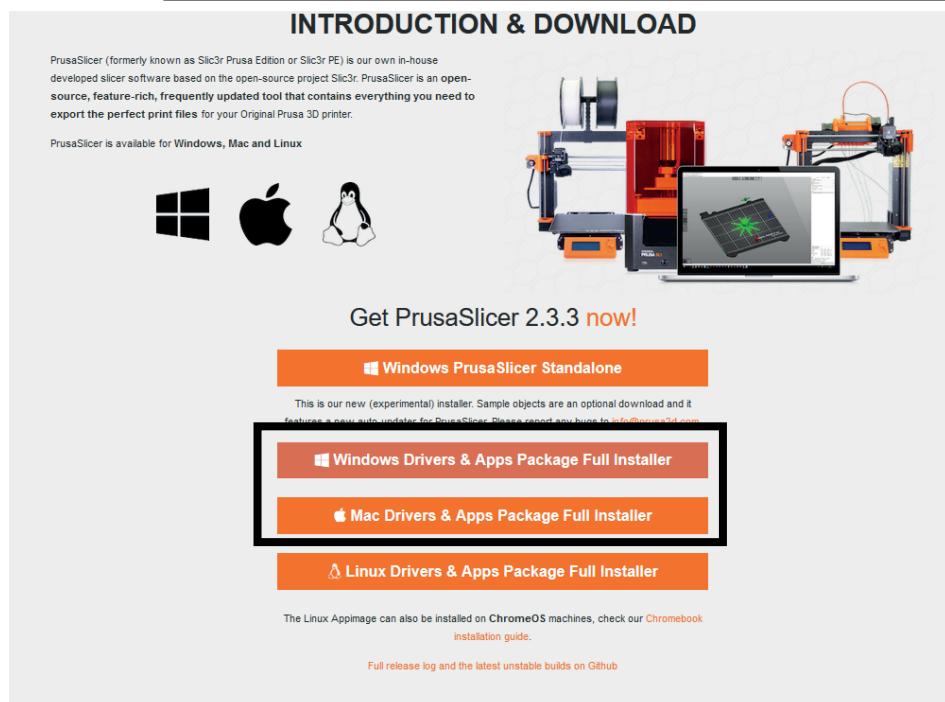
This guide helps you to get check your exported STL models and pick up problems and perform some fixes which may help prepare the model. Please note there is no guarantee that the model will be printable based on the simulation and the more works are usually required if the model is poorly designed .

Please note the document shows you the principle and in practice details may vary.

Prerequisite



<https://www.prusa3d.com/prusaslicer/>



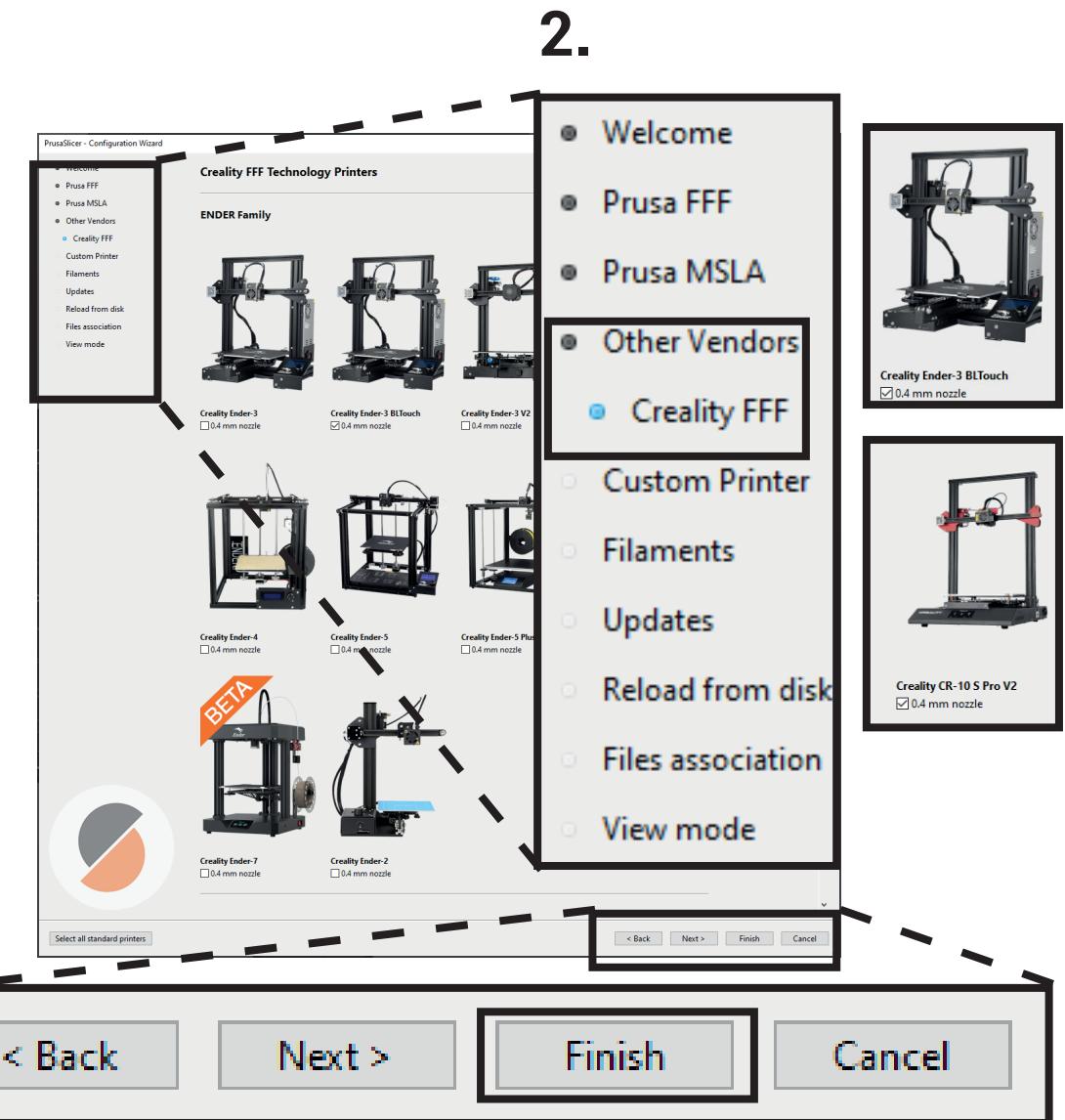
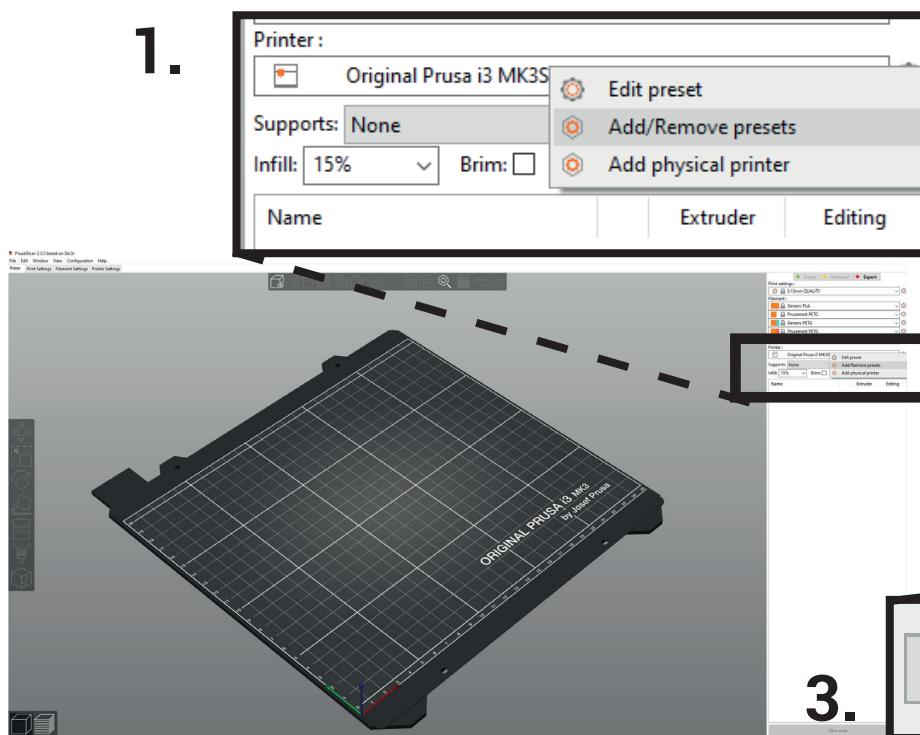
Steps

- 1. Select a printer profile**
- 2. Select a filament**
- 3. Load a model**
- 4. Check settings**
- 5. Slicing to G code**
- 6. Export**

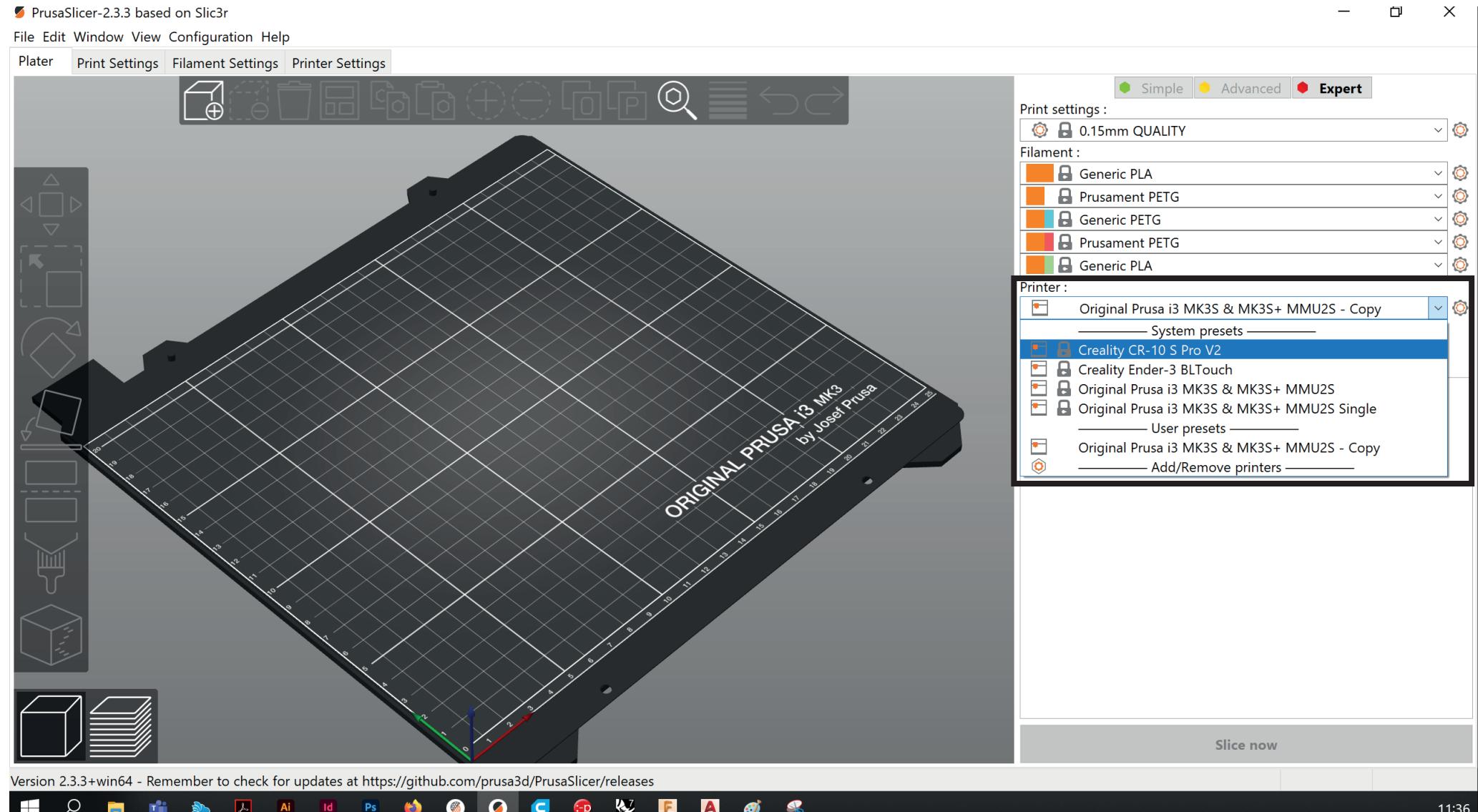
Configure

Next steps

Choose any models of 3d printers preset

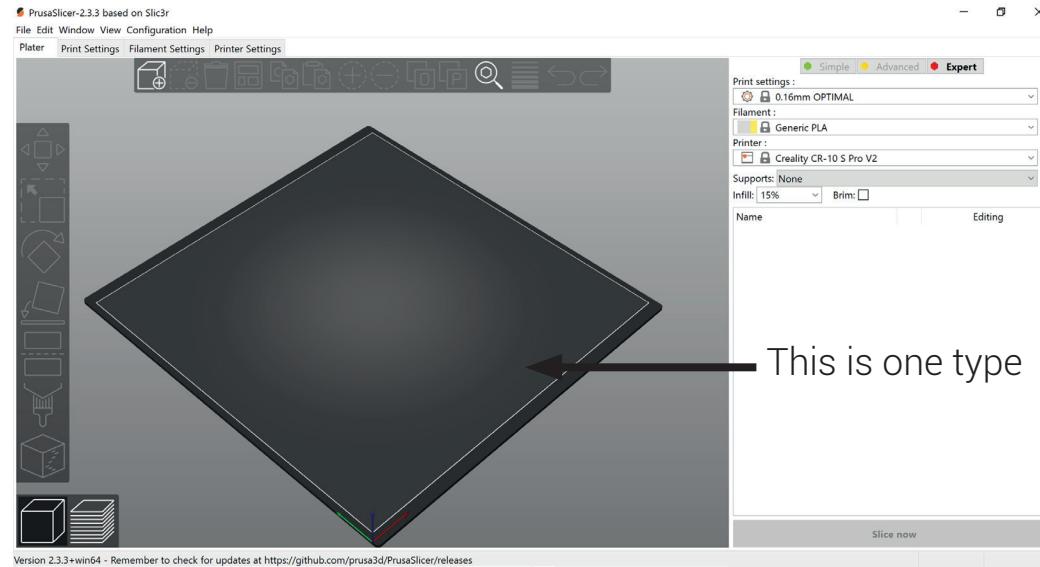


Select a printer profile



Select a preset profile.

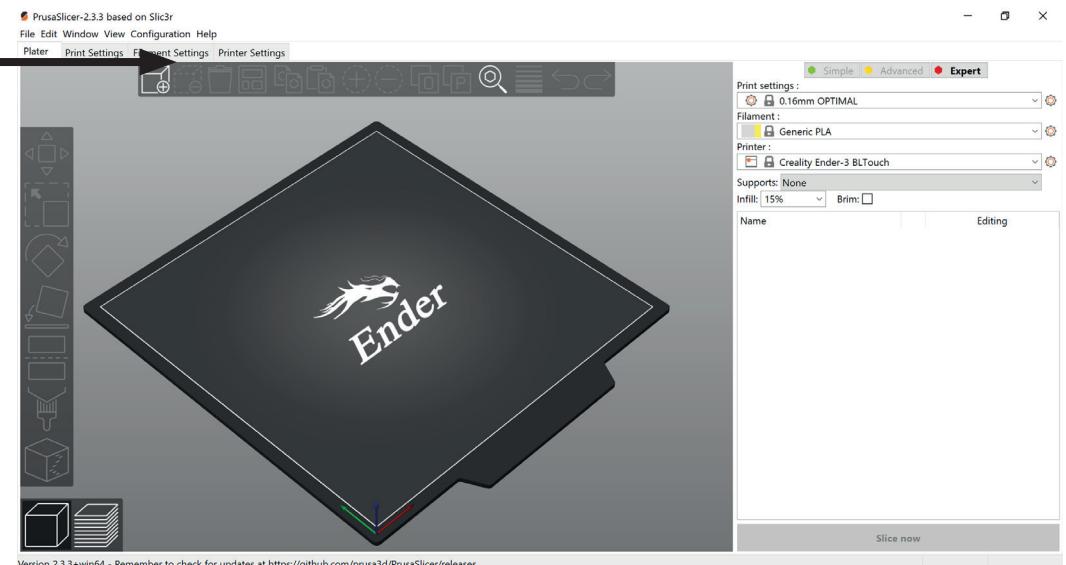
Select a printer profile



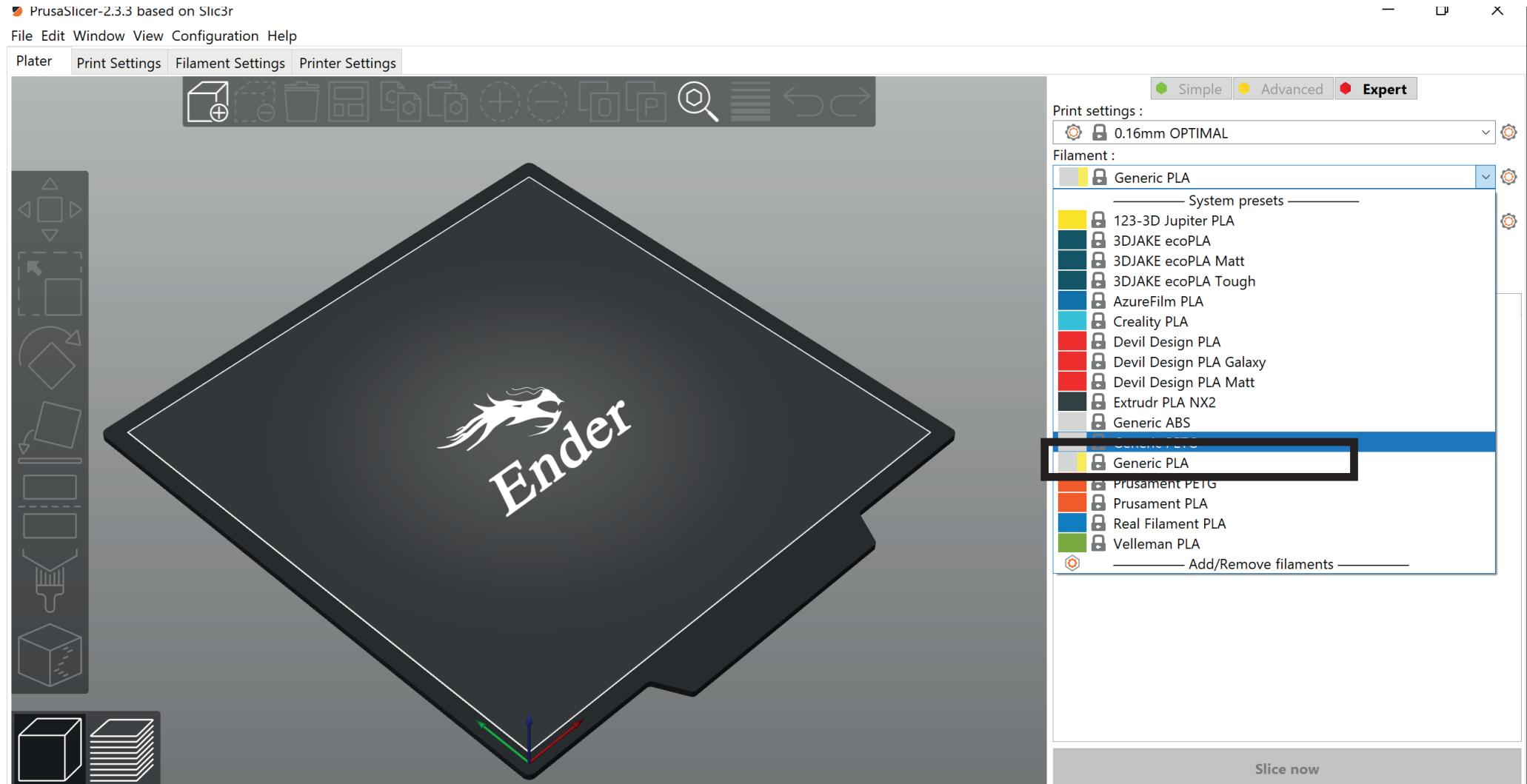
This is another type

Different profile may affect printing time estimation but it doesn't matter as final printing time can only be determined by DFF staff member due to other technical settings.

Once a printer profile is selected, the preview will change to reflect it. The only thing you need to consider is the size of the model and bed. Please check this separately with DFF staff.

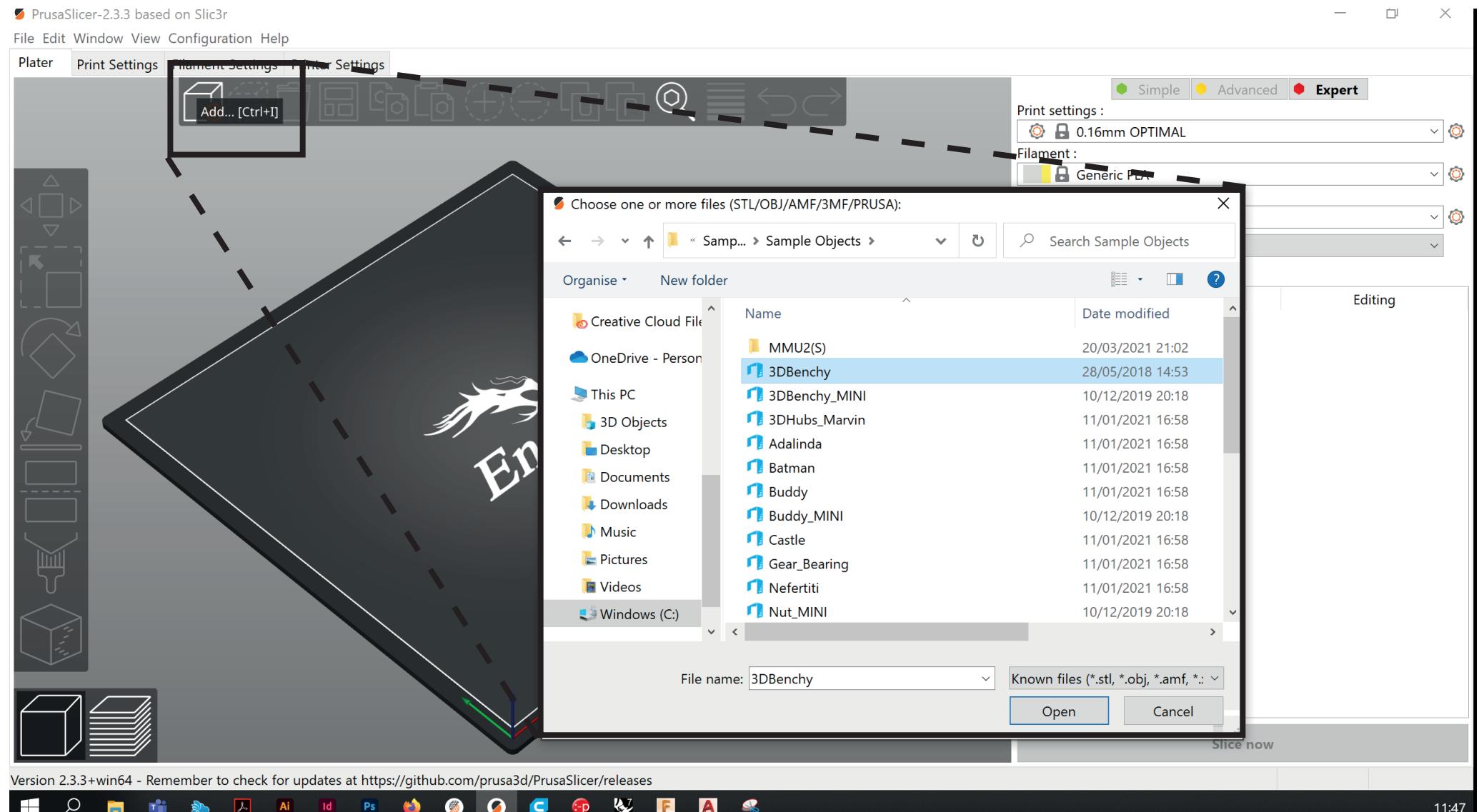


Select a filament



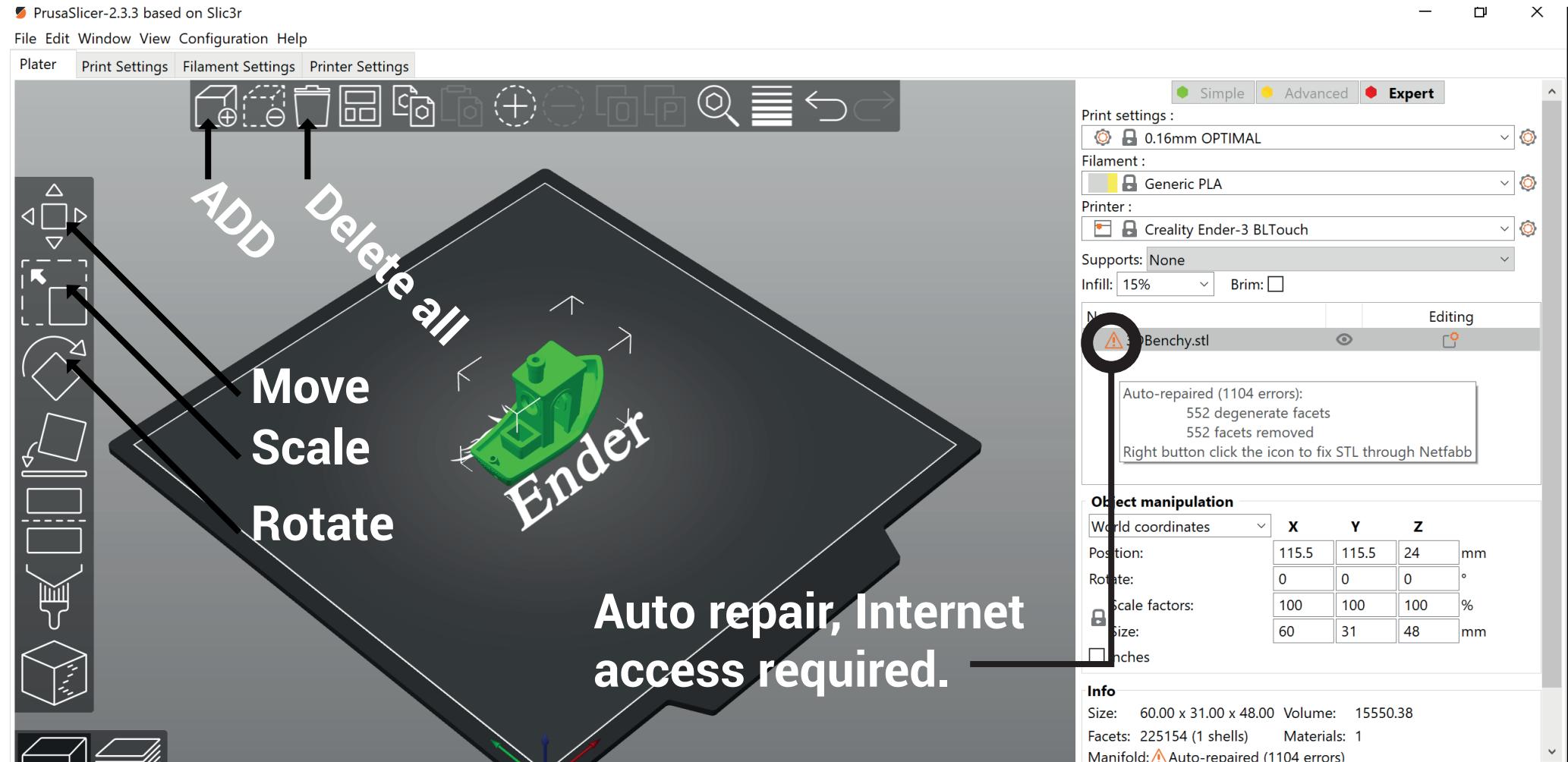
Select the filament you are using. Select PLA and use generic settings if the brand is not there to avoid complications.

Load model



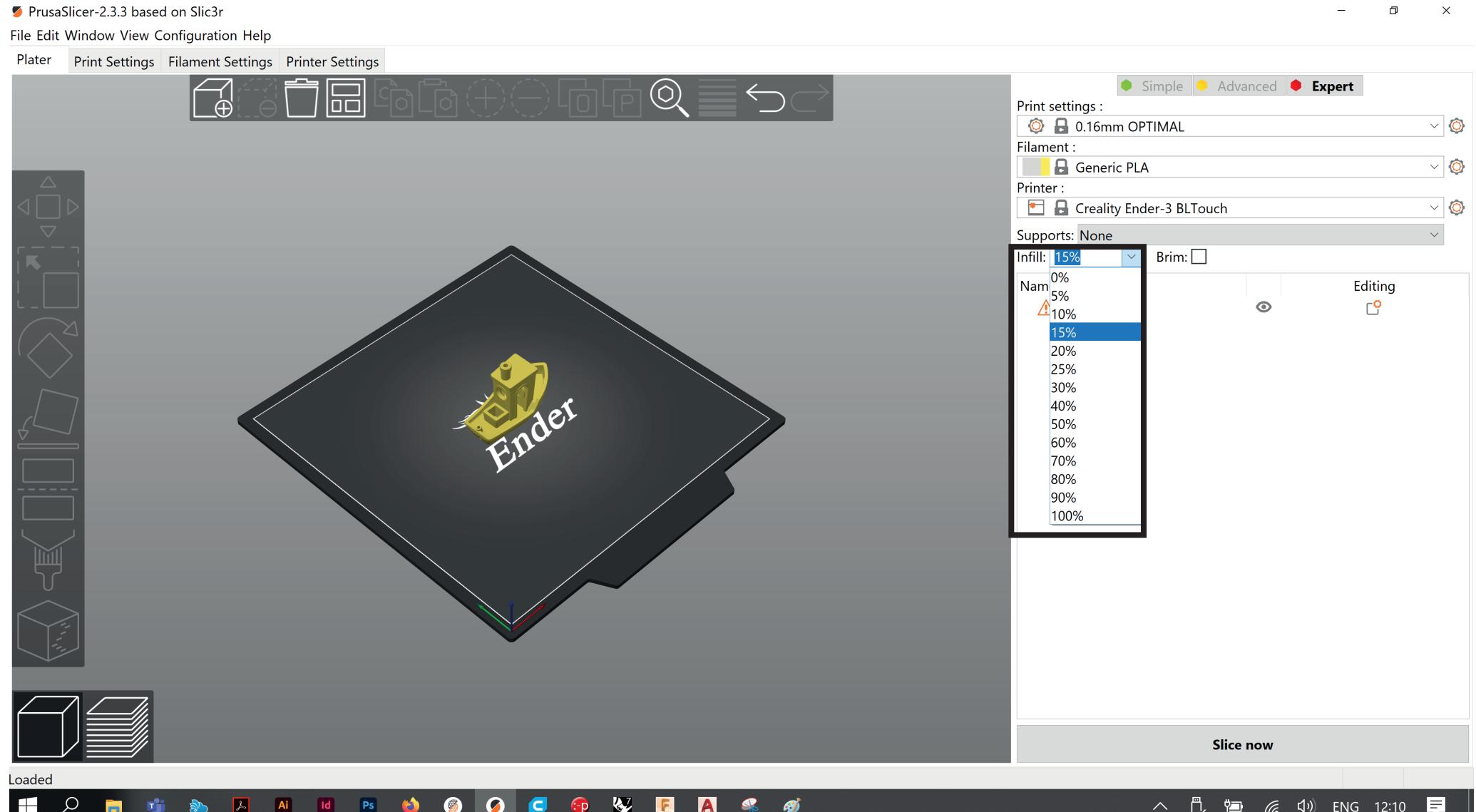
Use add bottom to load model.

Load model



Commonly used settings identified here. If you hover above a setting, you can see explanations.

Check settings



Check this for infill density, the less the quicker it is but also less strong.

Model simulation / Adjust settings

Check settings

PrusaSlicer-2.3.3 based on Slic3r

File Edit Window View Configuration Help

Plater Print Settings Filament Settings Printer Settings

0.16mm OPTIMAL @CREALITY

Layers and perimeters

Infill

Fill density: 15%

Fill pattern: Grid (highlighted)

Length of the infill anchor:

Maximum length of the infill anchor:

Top fill pattern:

Bottom fill pattern:

Ironing

Enable ironing:

Ironing Type:

Flow rate:

Spacing between ironing passes:

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

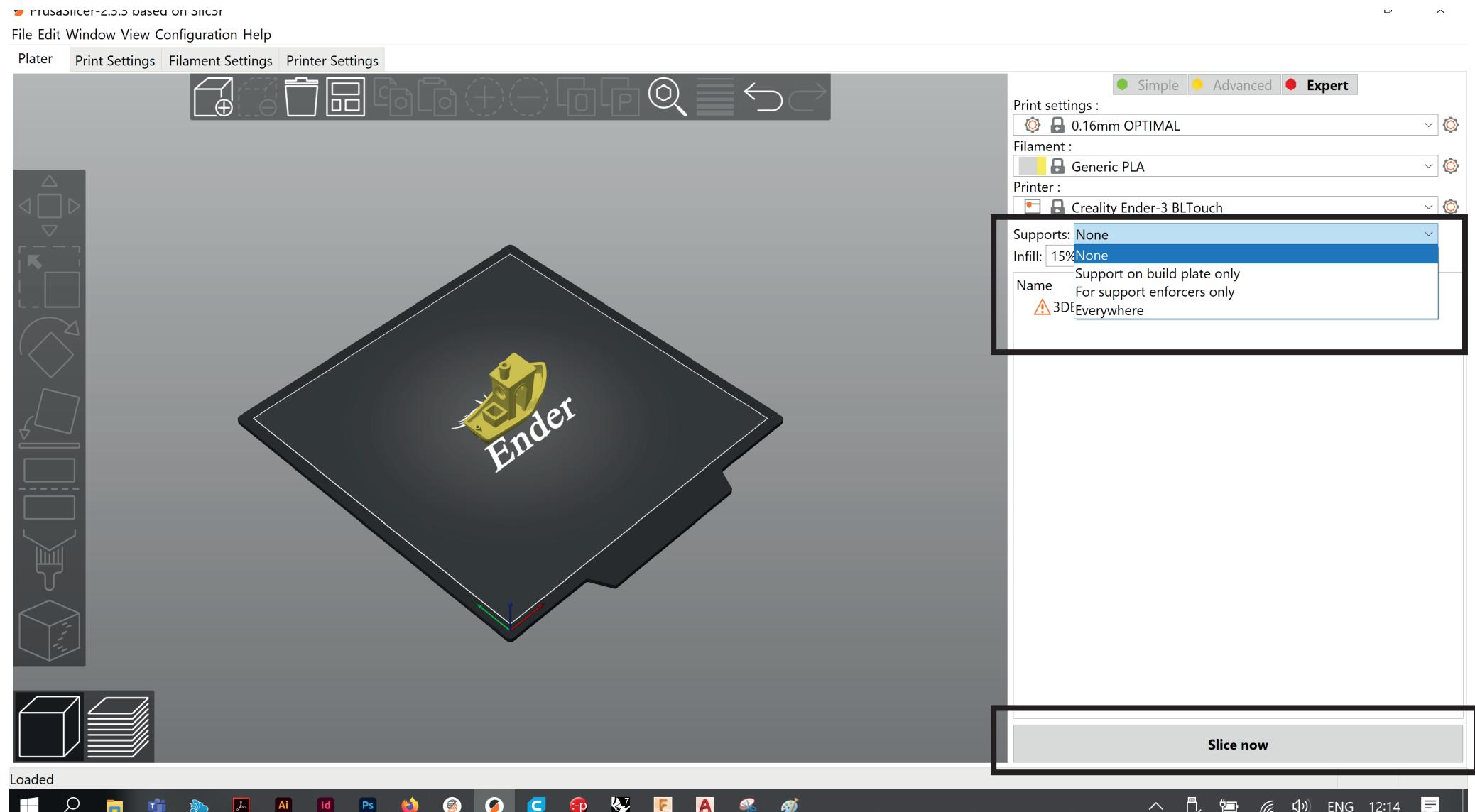
Bridging angle: 0 °

Only retract when crossing

Simple Advanced Expert

Check this for infill patterns. It may affect appearance or be used as a feature.

Check settings



Check this for support. Press slice now for next step

Slicing to G code

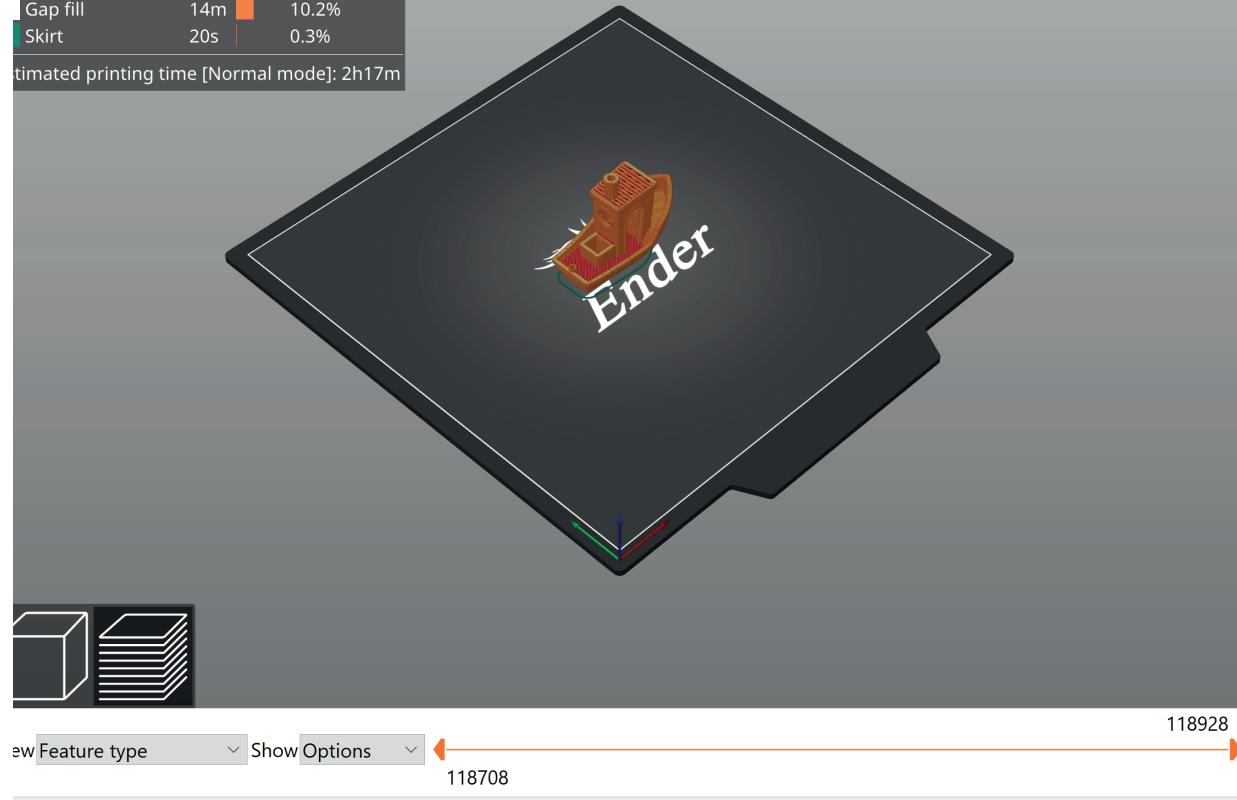
PrusaSlicer-2.3.3 based on Slic3r

Edit Window View Configuration Help

Printer Print Settings Filament Settings Printer Settings

Feature type	Time	Percentage
Perimeter	28m	20.7%
External perimeter	39m	28.5%
Internal infill	12m	8.6%
Solid infill	35m	25.2%
Top solid infill	4m	3.1%
Bridge infill	4m	2.7%
Gap fill	14m	10.2%
Skirt	20s	0.3%

Estimated printing time [Normal mode]: 2h17m



Print settings : Simple Advanced Expert

Print settings : 0.16mm OPTIMAL

Filament : Generic PLA

Printer : Creality Ender-3 BLTouch

Supports: None

Infill: 15% Brim:

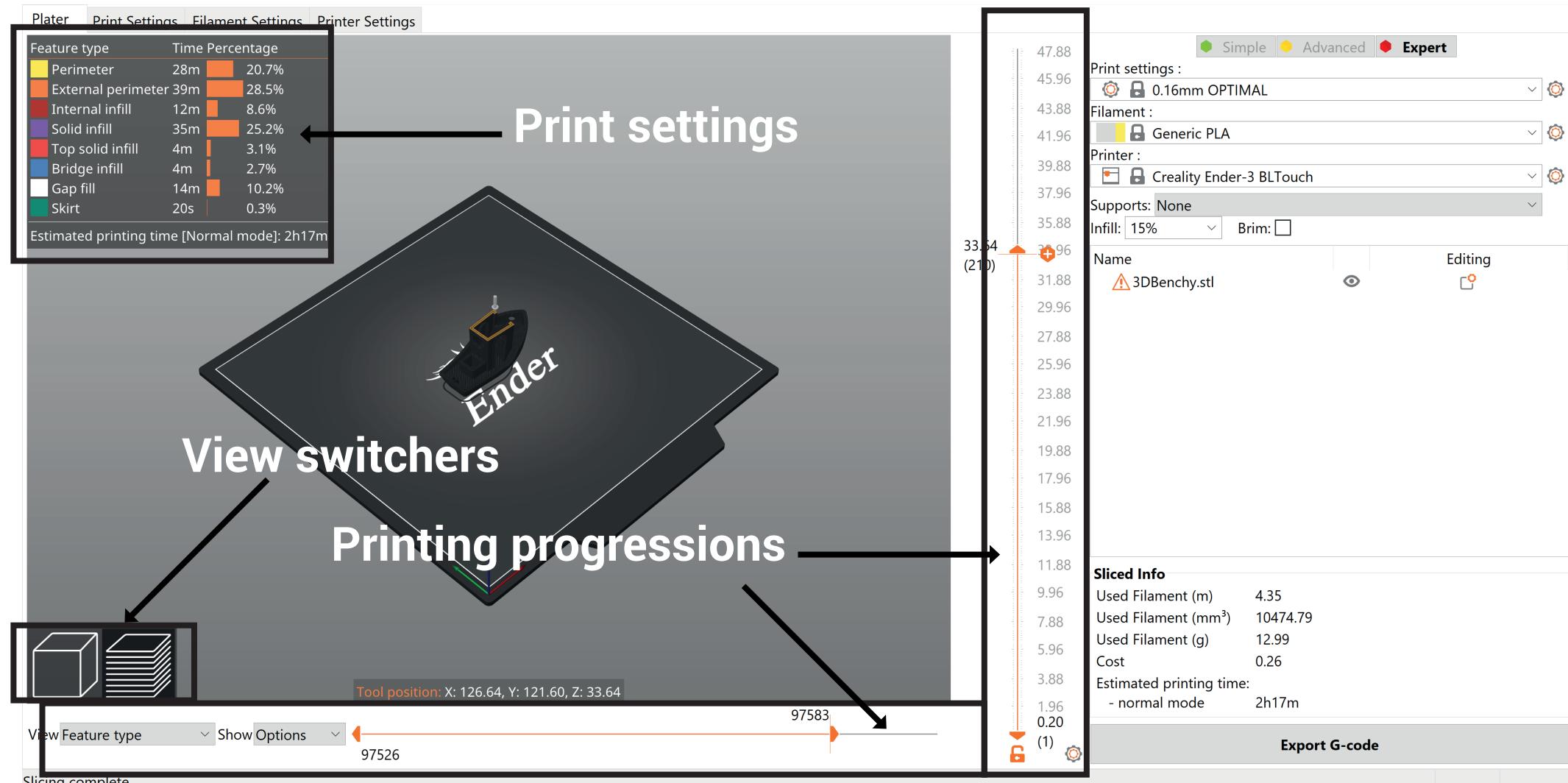
Name: 3DBenchy.stl Editing

Sliced Info

- Used Filament (m) 4.35
- Used Filament (mm³) 10474.79
- Used Filament (g) 12.99
- Cost 0.26
- Estimated printing time:
 - normal mode 2h17m

Export G-code

Slicing to G code



Important: Check these settings to review your G code if your model has too many floating pieces you may need to redesign or split the model to minimise support. Otherwise your print will be more likely to fail.

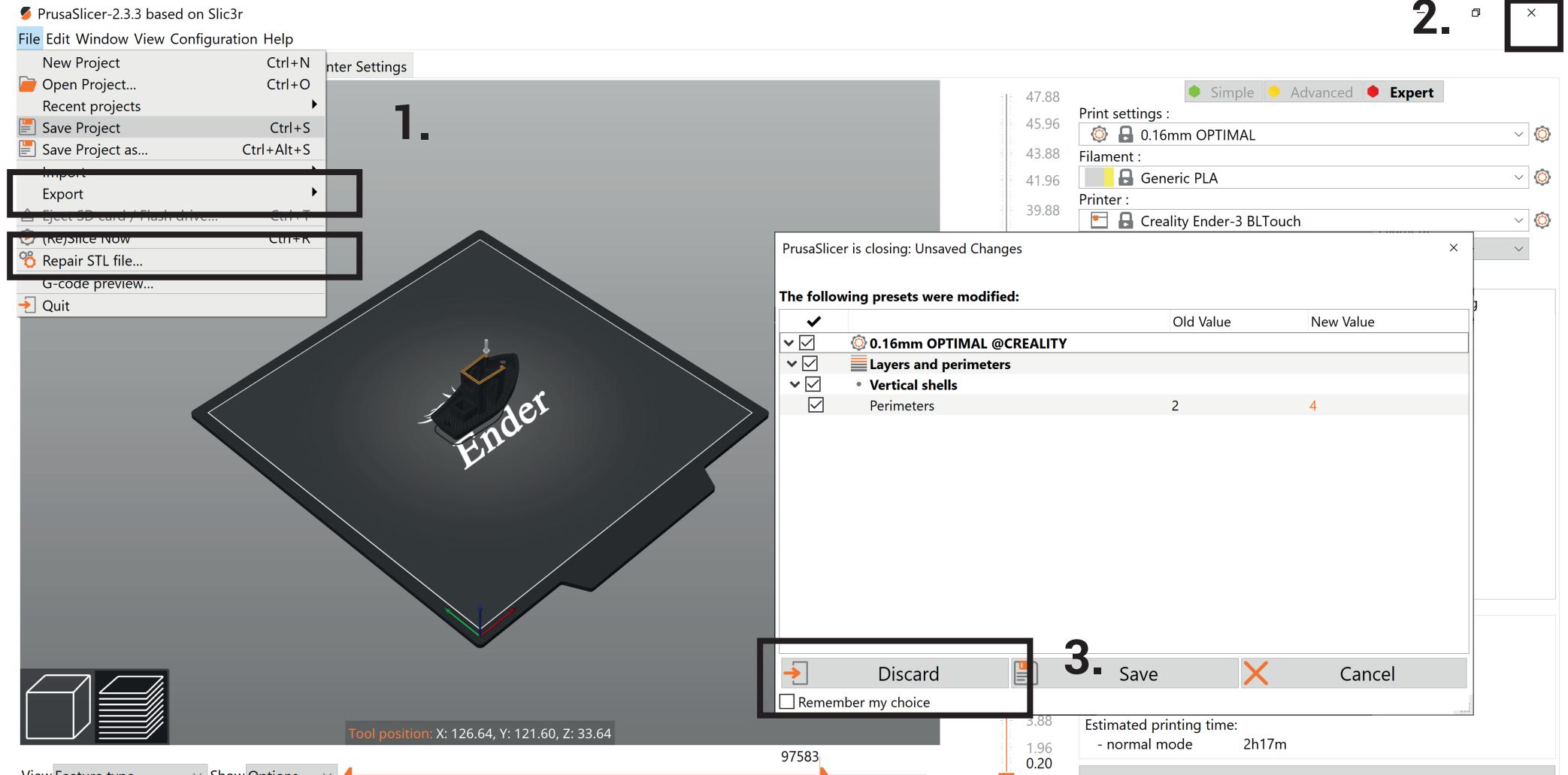
Export repaired file

The screenshot shows the PrusaSlicer software interface with the following details:

- File Explorer:** Shows the desktop folder structure. The "Desktop" folder is selected.
- Save G-code file as:** Dialog box:
 - File name: 3DBenchy_2h17m_0.16mm_205C_PLA_ENDER3BLTOUCH
 - Save as type: G-code files (*.gcode, *.gco, *.g, *.ngc)
 - Buttons: Save, Cancel
- Print Settings:**
 - Simple tab selected.
 - Print settings: 0.16mm OPTIMAL
 - Filament: Generic PLA
 - Printer: Creality Ender-3 BLTouch
 - Supports: None
 - Infill: 15%
 - Brim:
- Model Preview:** A 3D model of the DBenchy test object is shown with a vertical build plate scale.
- Save as (Stereolithography STL):** Dialog box:
 - File name: 3DBenchy
 - Save as type: StereolithographySTL (*.stl)
 - Buttons: Save, Cancel
- Bottom Status Bar:**
 - Tool position: X: 126.64, Y: 121.60, Z: 33.64
 - Feature type: Feature type dropdown
 - Show Options dropdown
 - 97526 (highlighted with an orange arrow)
 - 97583 (highlighted with an orange arrow)
 - Operation complete...
 - Please use auto repair by right click the warning sign and export as STL.
- Bottom Summary:**

Used Filament (mm ³)	10474.79
Used Filament (g)	12.99
Cost	0.26
Estimated printing time:	
- normal mode	2h17m

Aftermath



You can also find the repair STL and export options shown above. Choose discard when warning shows up as you have already exported the repair STL.