

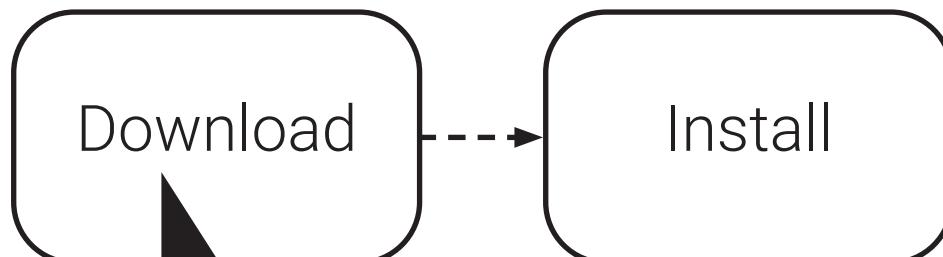
3D print self check

Guide to check your own 3d
models

This guide shows you how to perform a self check on your 3d model. The sample will uses Creality machines as example since they are hugely popular.

Please note your model will still be reviewed and checked after the completion of a self check.

Prerequisite



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

INTRODUCTION & DOWNLOAD

PrusaSlicer (formerly known as Slic3r Prusa Edition or Slic3r PE) is our own in-house developed slicer software based on the open-source project Slic3r. PrusaSlicer is an open-source, feature-rich, frequently updated tool that contains everything you need to export the perfect print files for your Original Prusa 3D printer.

PrusaSlicer is available for Windows, Mac and Linux

Windows Mac Linux

Get PrusaSlicer 2.3.3 now!

Windows PrusaSlicer Standalone

This is our new (experimental) installer. Sample objects are an optional download and it features a new auto-updater for PrusaSlicer. Please report any bugs to [GitHub](https://github.com).

Windows Drivers & Apps Package Full Installer

Mac Drivers & Apps Package Full Installer

Linux Drivers & Apps Package Full Installer

The Linux AppImage can also be installed on ChromeOS machines, check our [Chromebook installation guide](#).

Full release log and the latest unstable builds on [GitHub](#)

Select Components

Which components should be installed?

Select the components you want to install; clear the components you do not want to install. Click Next when you are ready to continue.

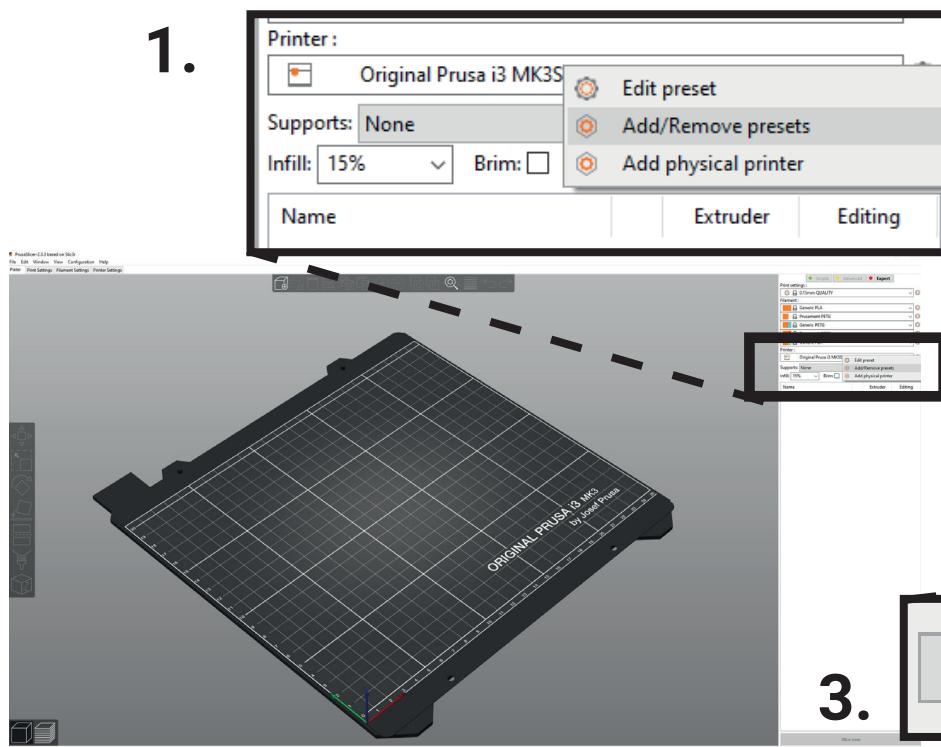
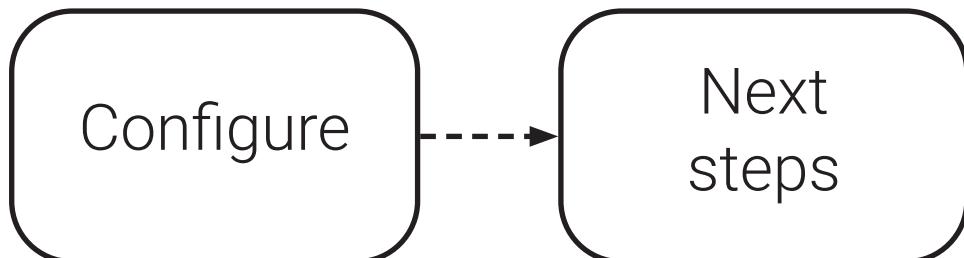
All printers

Component	Size
PrusaSlicer 2.3.0	70.8 MB
Windows drivers	3.9 MB
Utilities	53.9 MB
Test objects (MK2)	415.2 MB
Test objects (MK2 MMU1)	493.0 MB
Test objects (MK3 / MK3S)	184.0 MB
Test objects (MK3 + MMU2)	113.5 MB
Test objects (MK3S + MMU2S)	112.3 MB
Test objects (MK2.5 + MMU2)	73.5 MB
Test objects (MK2.5S + MMU2S)	76.4 MB

Current selection requires at least 1,432.1 MB of disk space.

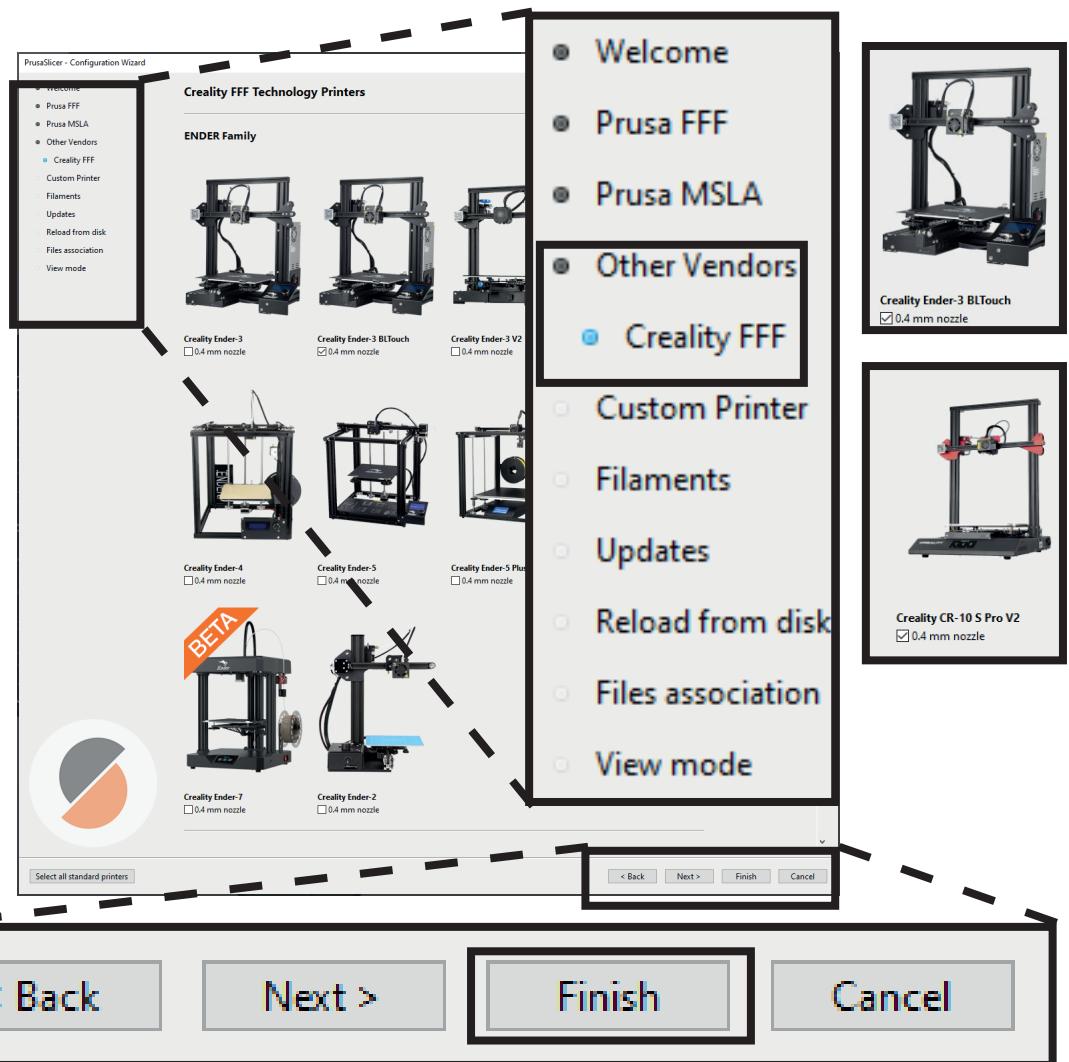
Next > Cancel

Prerequisite



Make sure you have the following configuration installed

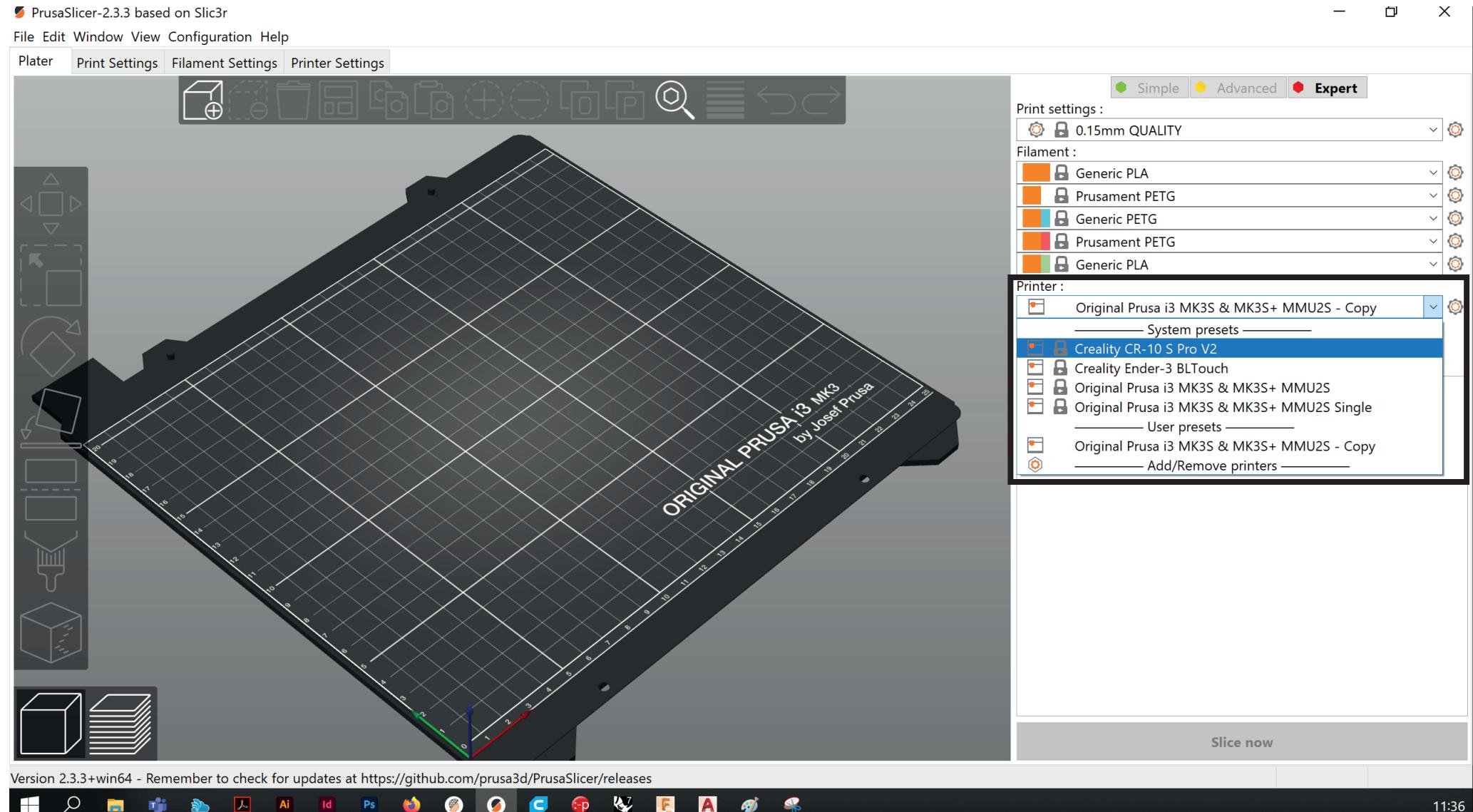
2.



Steps

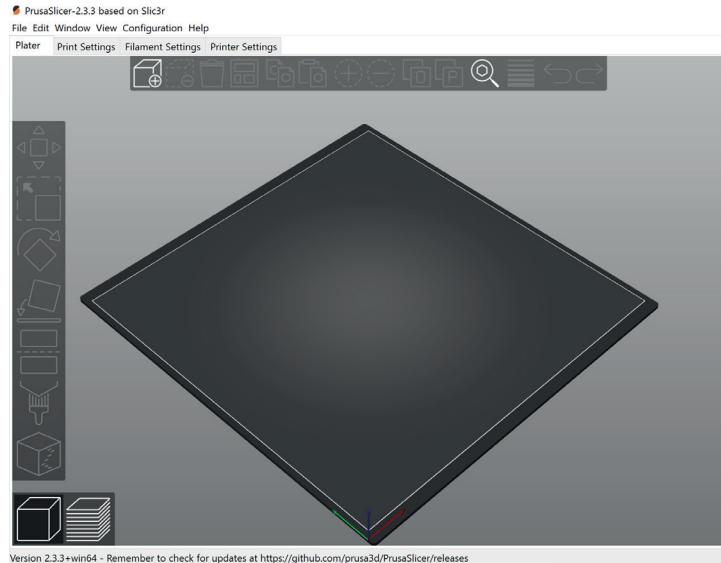
- 1. Select correct printer profile**
- 2. Select filament**
- 3. Load model**
- 4. Check settings**
- 5. Slicing to G code**
- 6. Export G code**

Select correct printer profile



Select the correct printer profile as shown

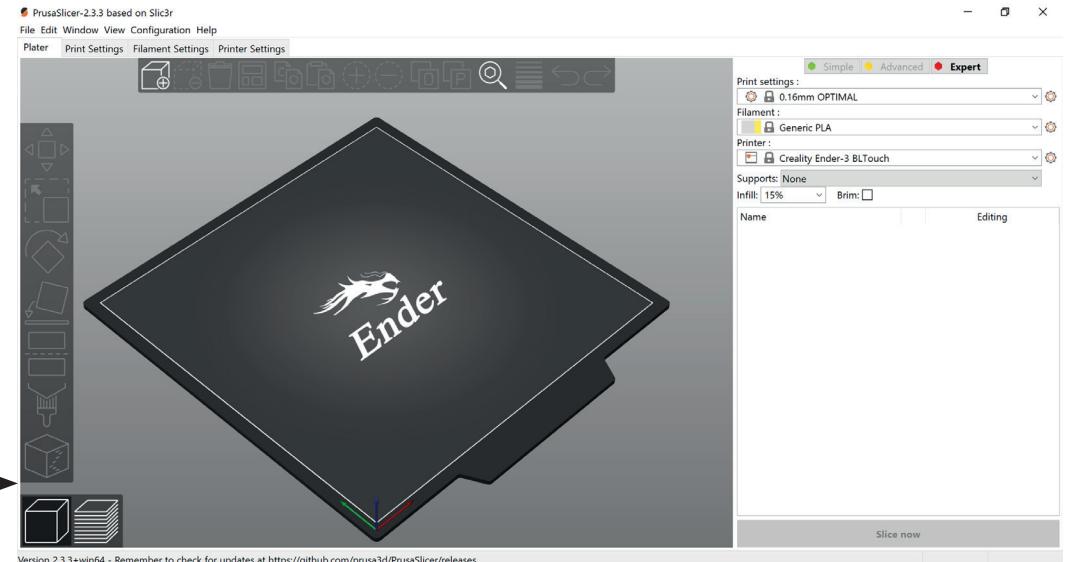
Select correct printer profile



Version 2.3.3+win64 - Remember to check for updates at <https://github.com/prusa3d/PrusaSlicer/releases>

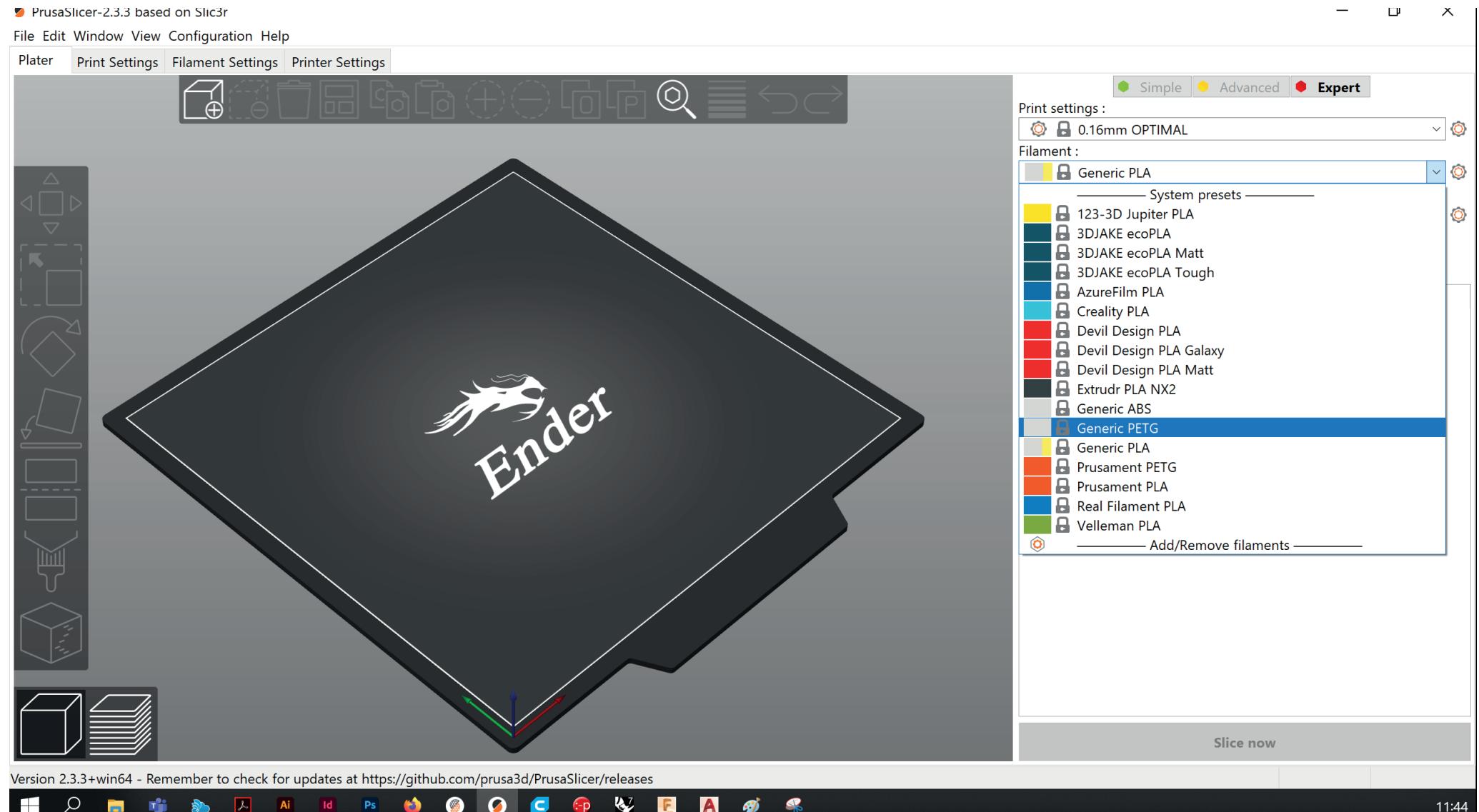
Once a printer profile is selected, the preview will change to reflect it

← CR 10 style machine



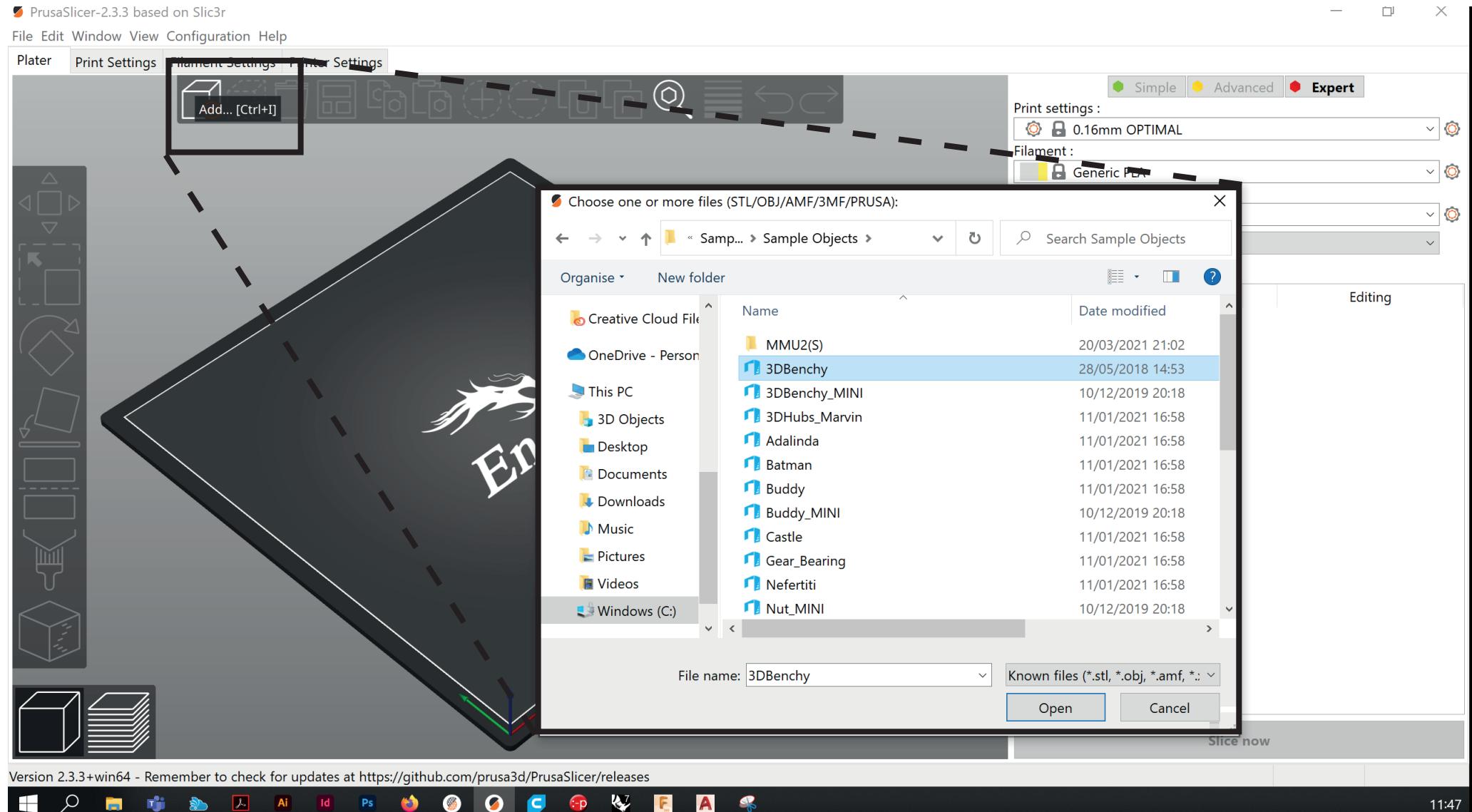
Ender 3 style machine →

Select filament



Select the filament you are using. Use Generic settings if the brand is not there.

Load model



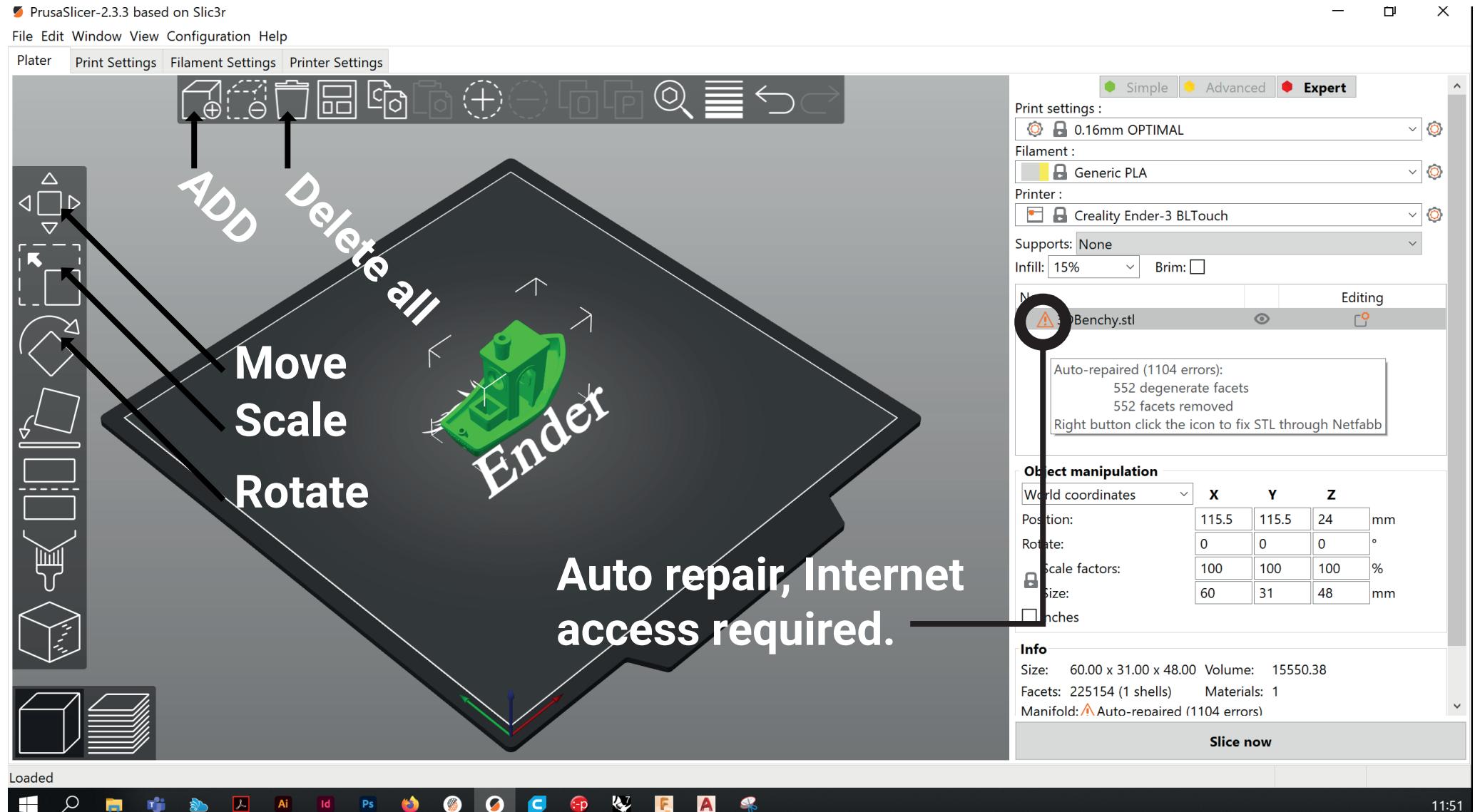
Version 2.3.3+win64 - Remember to check for updates at <https://github.com/prusa3d/PrusaSlicer/releases>



11:47

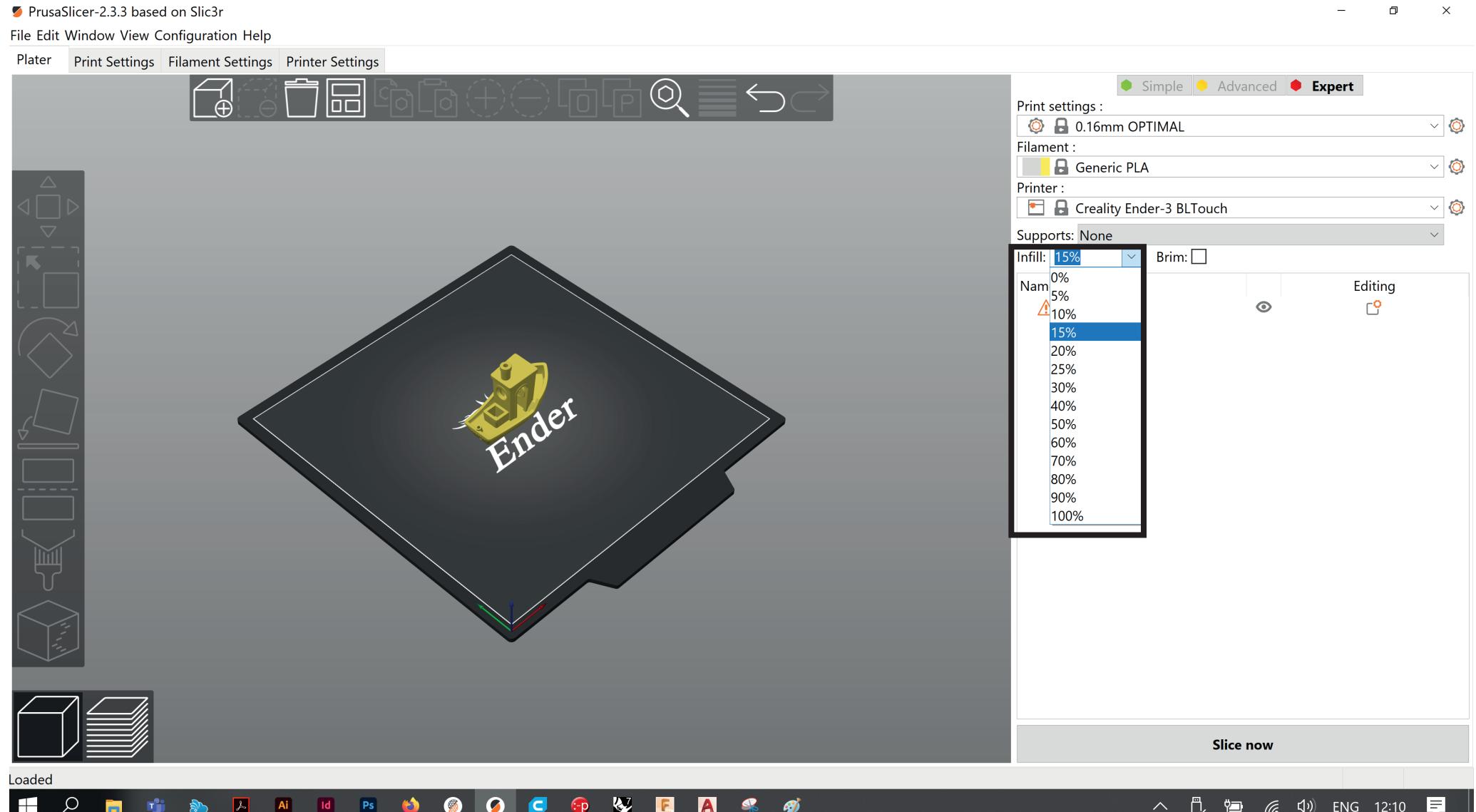
Use add bottom to load model

Load model



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

Check settings



Check this for infill density

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

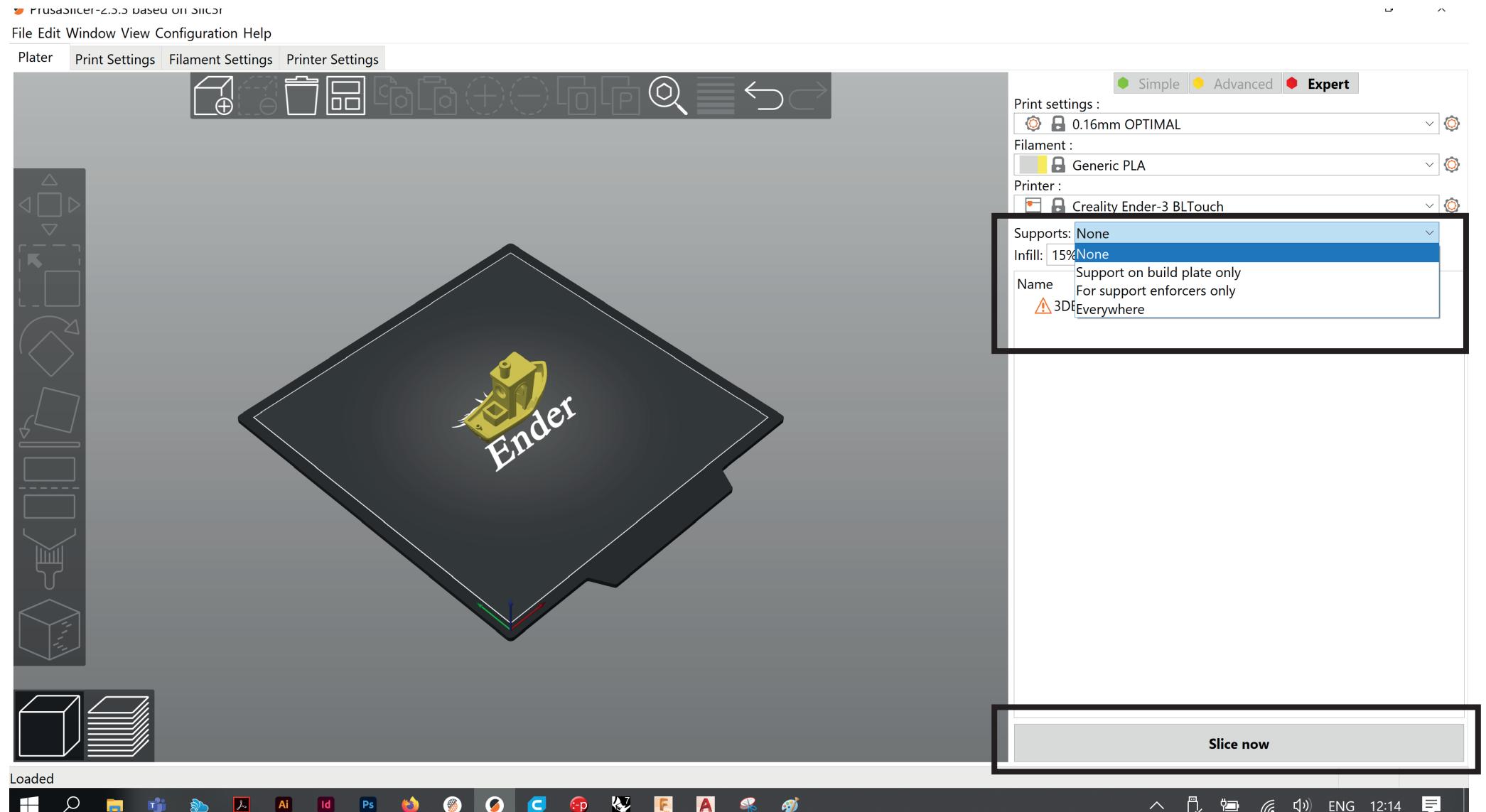
Bridging angle: 0 °

Only retract when crossing

Simple Advanced Expert

Check this for infill patterns

Check settings



Check this for support. Press slice now for next step

Slicing to G code

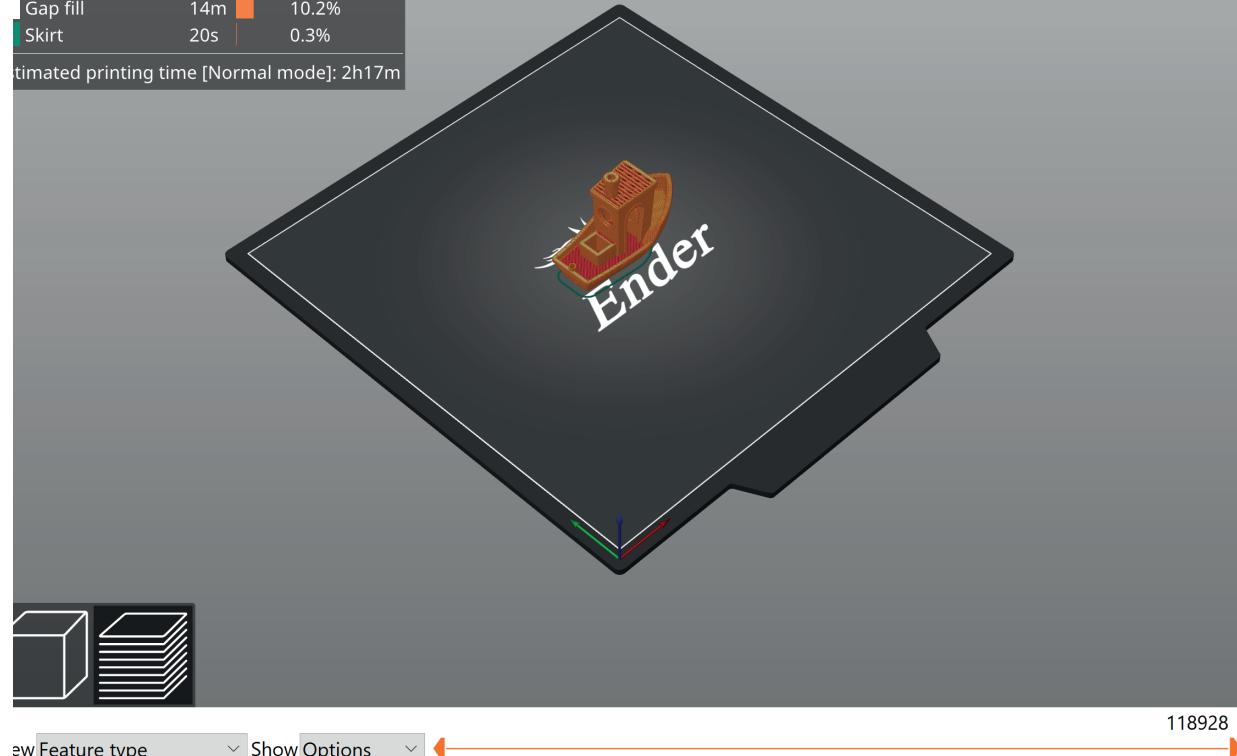
PrusaSlicer-2.3.3 based on Slic3r

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



Printing complete...

Gcode preview

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

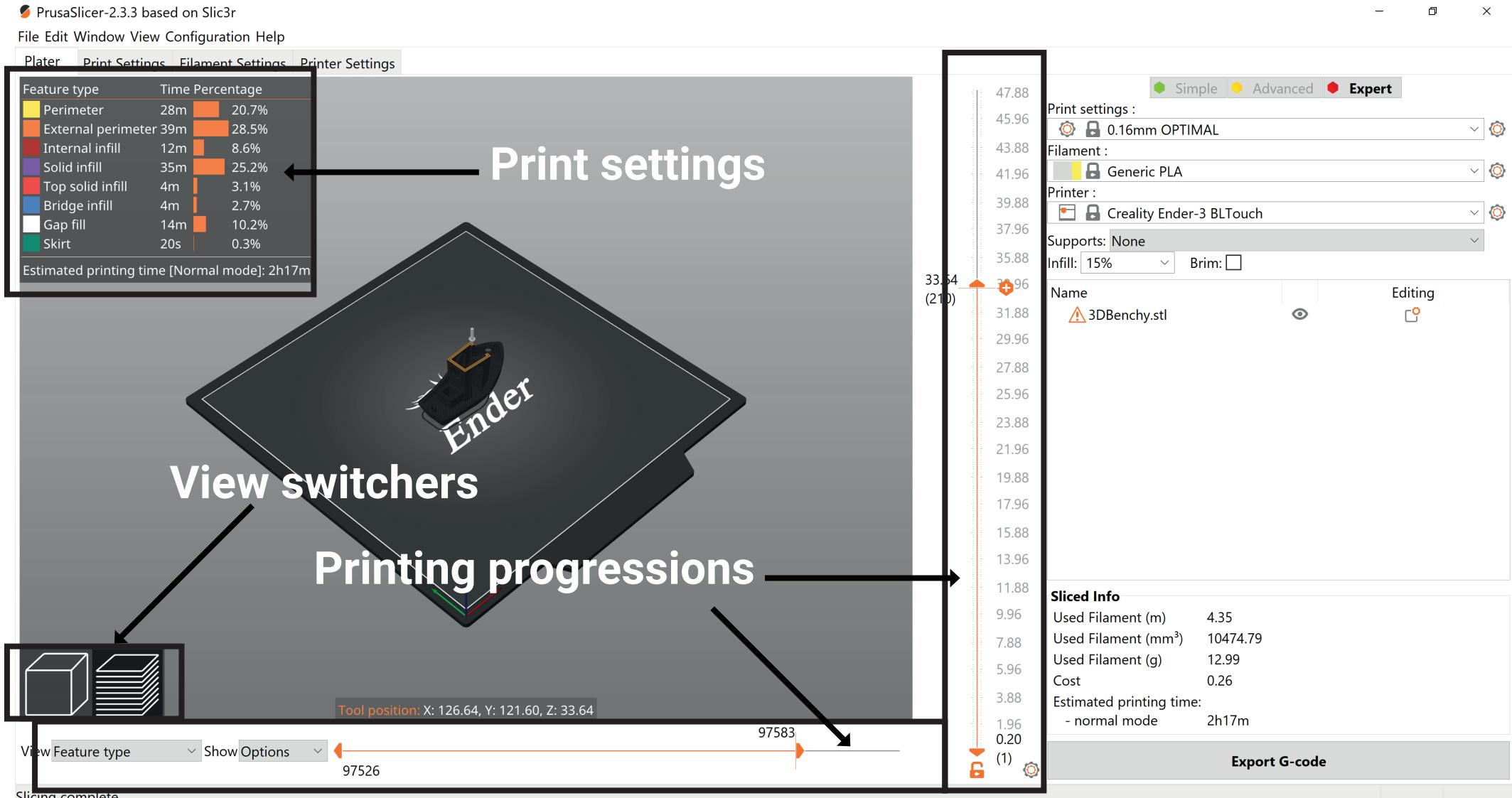
Sliced Info

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

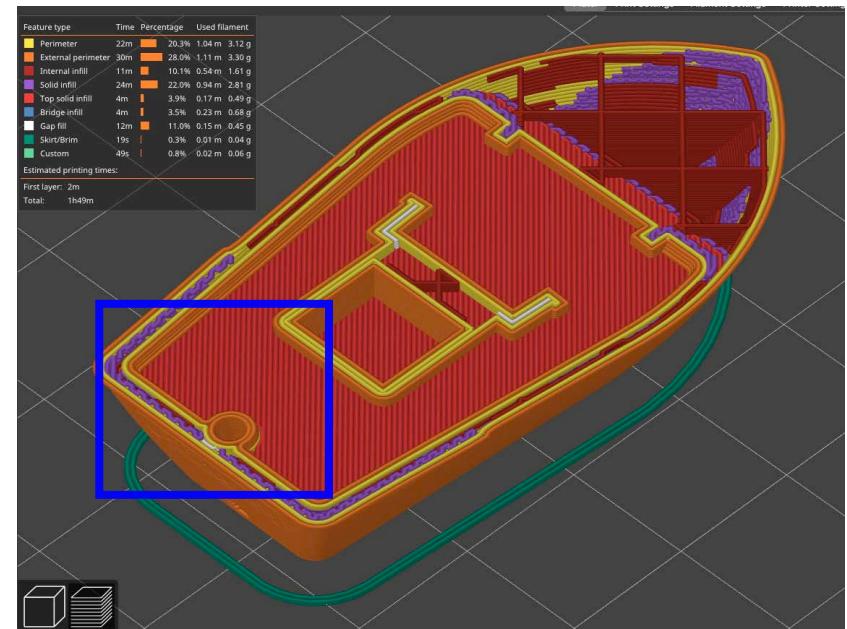
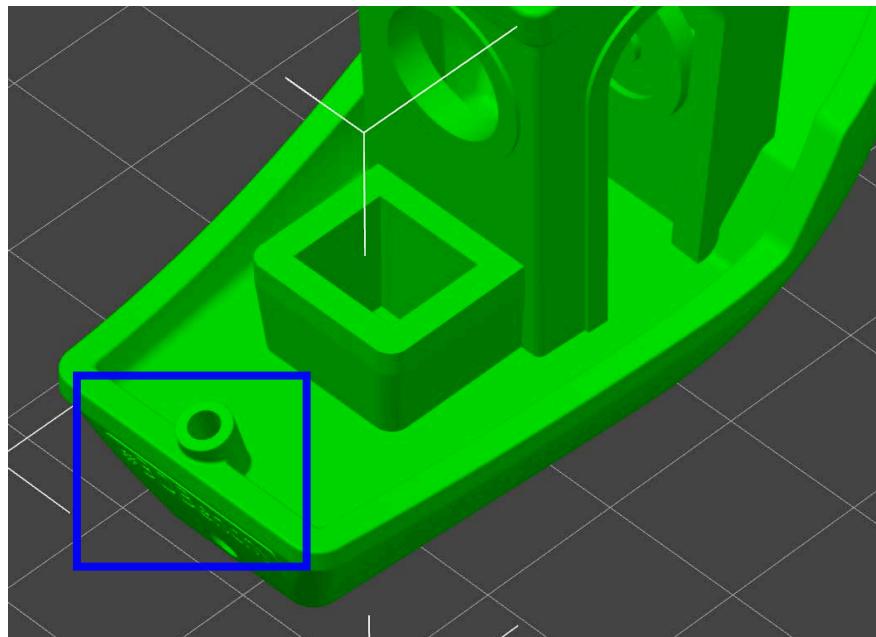
Slicing to G code



Check these settings to review your G code

On the left you see a model compared with the slice path on the right. In the highlighted part you see that some details are barely thick enough to be printed.

We will recommend you not to have any parts in your model smaller than 5mm.

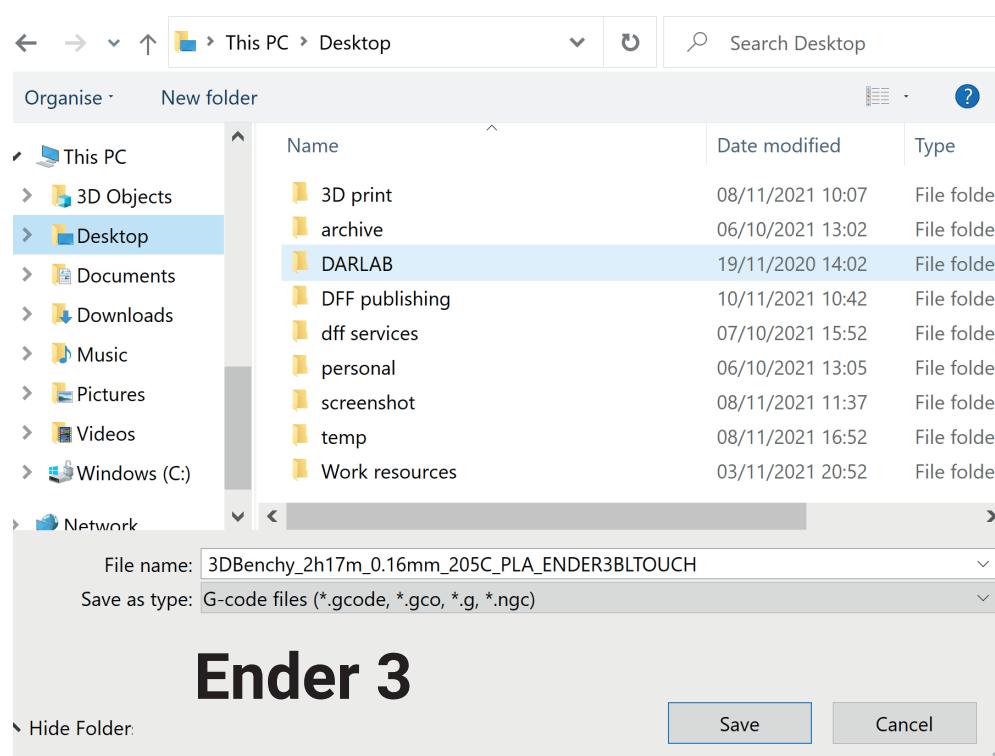


Export G code

PrusaSlicer-2.3.3 based on Slic3r

Edit Window View Configuration Help

Save G-code file as:



Ender 3

Save

Cancel

Hide Folder



Tool position: X: 126.64, Y: 121.60, Z: 33.64

Feature type

Show Options

97526

97583

ing complete...

CR10

Save

Cancel

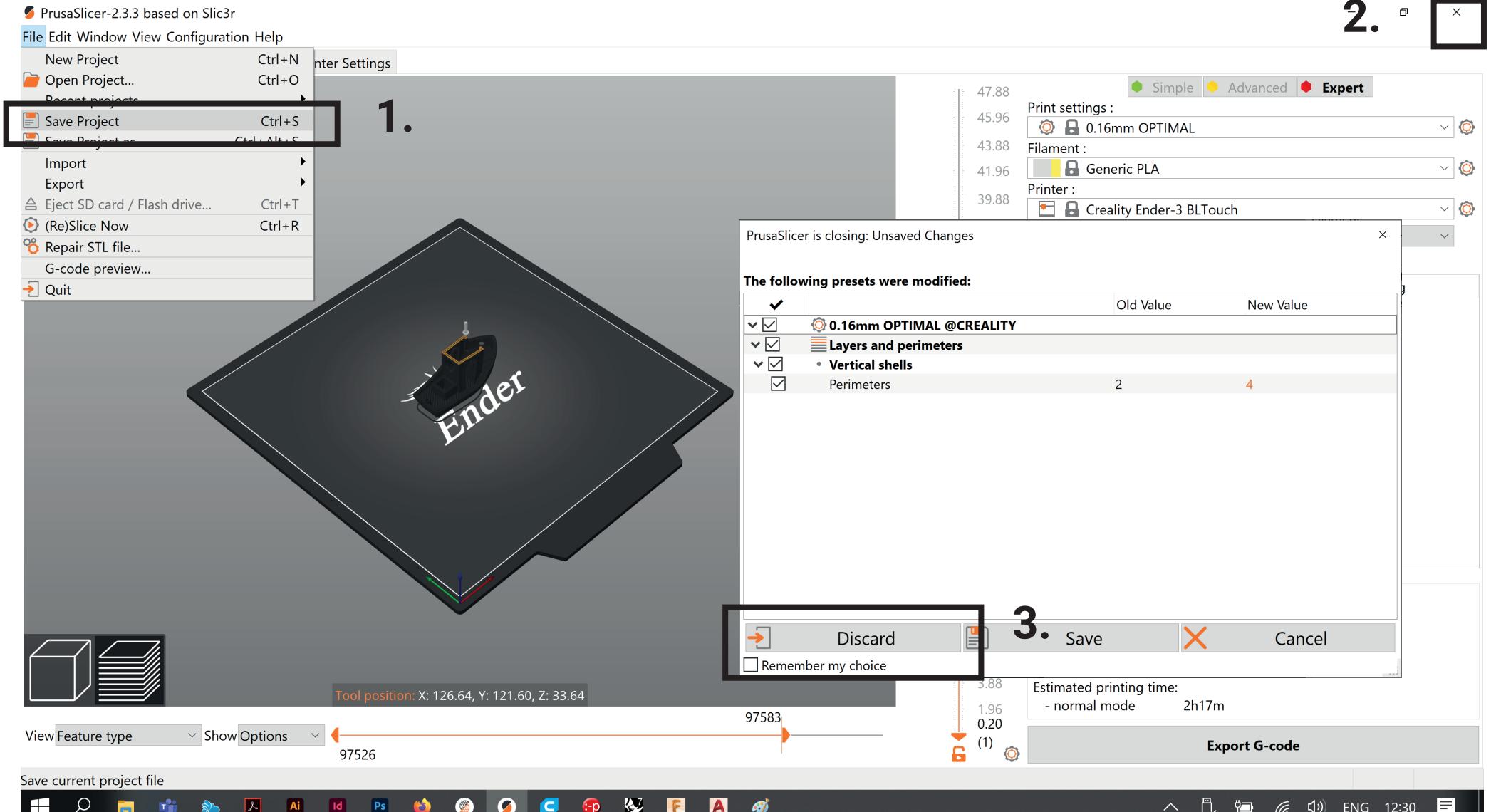
Hide Folder

7.88	Used Filament (mm³)	10474.79
5.96	Used Filament (g)	12.99
Cost	0.26	
Estimated printing time:		21.17h
Normal mode		

Export G-code

Press export G code to export file. For CR10, avoid special character in file names. **Use cost estimate to get rough figure.**

Aftermath



Save your file afterwards then close. Choose discard when warning shows up as you have already saved.

Contact us if you have any question

Digital Fabrication Facilities

Level 7 Keyworth

t:+44 (0)20 7815 7464

dffarchitecture@lsbu.ac.uk

Mon-Fri: 9am-5pm