

**Fusion3**

# **EDGE 3D Printer**

**INTERNAL:**

**INSTALLING & USING PRUSASLICER**

Revision 10/31/2022

# CONFIGURING & USING PRUSASLICER WITH EDGE

How to install, configure, and use Prusaslicer with EDGE

## DOWNLOAD & INSTALL

1. Download using this link:  
[https://cdn.prusa3d.com/downloads/drivers/PrusaSlicer\\_Win\\_standalone\\_2.5.0.exe](https://cdn.prusa3d.com/downloads/drivers/PrusaSlicer_Win_standalone_2.5.0.exe)
2. During the install process:
  - a. On the "Select Components" window, de-select everything except "PrusaSlicer X.X.X" (should be the first item).
  - b. On the "Completing Setup" screen, de-select both "Open tutorials" and "Open Prusaprinters.org".
3. The first time you run PrusaSlicer, it will open a configuration wizard. Click "Cancel".
4. On the top right side of the software, click "expert" mode.

## LOAD FUSION3 PROFILES

1. Download the Fusion3 configuration folder from [HERE](#) (updated 10/31/22).
2. Open PrusaSlicer. Go to *Help > Show configuration folder*. It will open a windows explorer window.
3. Close PrusaSlicer.
4. Copy and paste the contents of the Fusion3 config folder into this location. When asked if you want to replace existing files, select "**Yes to all**".

**IMPORTANT: If you have custom profiles for other printers, or use PrusaSlicer with other printers, do not over-write all the folders. Instead, copy the contents of each folder into the appropriate place so that you don't lose any of your existing profiles.**

5. Re-open PrusaSlicer. Confirm that you see the following:
  - a. Printer settings: F3 EDGE 0.4, F3 EDGE 0.6, F3 EDGE 0.8
  - b. Filament settings: Should have profiles for all our basics in the format of "Material\_NozzleSize\_QualityLevel"
  - c. Print settings: EDGE 0.4 STANDARD, EDGE 0.4 DRAFT, EDGE 0.4 FAST, EDGE 0.4 FINE, EDGE 0.6 STANDARD, EDGE 0.8 STANDARD

# SLICING WITH PRUSASLICER

**Note:** This is not a comprehensive guide for how to use PrusaSlicer.

PrusaSlicer breaks print settings into 3 profiles: Printer, Filament, and Print Settings. This has some advantages and some disadvantages over a single profile setup such as Simplify3D.

Due to how PrusaSlicer segments out print quality settings and material settings, we have to make a separate Filament profile for different quality levels AND different nozzle sizes. For instance, in the material list drop-down generic ABS has 5 Filament profiles:

- ABS\_0.4\_Standard
- ABS\_0.4\_Draft-Fast (these two are close enough they can share settings)
- ABS\_0.4\_Fine
- ABS\_0.6\_Standard
- ABS\_0.8\_Standard

When slicing with PrusaSlicer, it's very important that you always make sure two things match:

1. **Printer** selected (nozzle size) matches the nozzle size for your **Print Settings** preset and your **Filament** preset.
2. The **quality level** (standard, fine, fast, or draft) in your **Print Settings** preset matches the quality level in your **Filament** preset.

If these things don't match you will get bad output from the slicer.

The current batch of profiles are set up using PrusaSlicer's "dependencies", so you only see the applicable material profiles for the nozzle size you have selected. You still need to match quality presets and material profiles manually as described.

## Examples

This is ok:

- Print settings: EDGE 0.4 STANDARD
- Filament: PLA\_0.4\_Standard
- Printer: F3 EDGE 0.4

This is NOT ok:

- Print settings: EDGE 0.4 **STANDARD**
- Filament: PLA\_0.4\_ **Fast**
- Printer: F3 EDGE 0.4

This is NOT ok:

- Print settings: EDGE 0.4 **DRAFT**
- Filament: PC-PBT\_0.6\_Draft-Fast
- Printer: F3 EDGE 0.4

## FURTHER NOTES ON PROFILES

Much like we do for Simplify3D, for most materials we publish a single profile for that entire family. For specific materials, you may find you need to make small changes to this family profile to achieve optimum results.

Some popular materials that require very specific settings, such as Atomic's Nuclear Nylon PA-CF, have their own profile.

Not all materials are going to have all quality presets (standard, draft, fast, fine) available. Sometimes, this is because the material simply won't achieve the performance needed for that preset (PC can only be melted so fast, for example).

As of today (10/31/22), all profiles have been tested. The 0.6 and 0.8 profiles have less run time on them, so if you encounter issues or settings that need to be changed, please contact us so we can make those updates.

## Adding New Profiles

We are happy to continue to add profiles for new materials to PrusaSlicer! Please reach out to us about this if you encounter either scenario:

- If you print with a material in a family that already has a profile (ex: ABS) but it doesn't work well and requires custom settings.
- If you want to use a material that requires a custom profile that doesn't exist at all.