

Fusion3

EDGE 3D Printer

USING REACTOR WITH EDGE

Revision 5/28/2022

USING REACTOR WITH EDGE

How to configure & use Fusion3's REACTOR slicing software with your EDGE 3D printer

INTRODUCTION

Fusion3's REACTOR Is Only Supported Slicer

Unlike previous Fusion3 products, EDGE's only officially supported slicer is REACTOR.

Must Use Reactor Version > 1.2.3

If you're an existing Fusion3 customer, please ensure you update your version of REACTOR to version 1.2.3 or higher to gain access to the EDGE profiles in the software. This version will not disable your access to F400 or F410 profiles.

INSTALL & ACTIVATION

Please see our standalone documentation for how to download, install, & activate the software.

AVAILABLE PROFILES

EDGE is certified to work with all the same materials as our F410. Selecting the appropriate material and profile in REACTOR works the same way:

1. Select your material family
2. Select the brand and type of filament you're using

QUALITY PRESETS

NOTE: If you have used a Fusion3 F400 or F410 before with REACTOR, you will find that quality presets work differently for EDGE in REACTOR than they do with the presets for Fusion3's F400 & F410. For the F400 and F410, the quality presets in REACTOR primarily alter the layer height, but leave other settings (speed, temperatures, retraction) alone.

EDGE's quality presets alter the layer height, but also the print speed (and other supporting settings). This allows us to take advantage of EDGE's much larger and better performance envelope in terms of speed and print quality. For example, with ABS we have developed the following quality presets using the .4MM print nozzle:

STANDARD quality: 0.2mm layer height, 100mm/s base print speed, 30% infill

FAST quality: 0.2mm layer height, 200mm/s base print speed, 12-18% infill

DRAFT quality: 0.3mm layer height, 180-200mm/s base print speed, 10-20% infill

FINE quality: 0.1mm layer height, 50mm/s base print speed, 50% infill

What to expect from each quality preset:

STANDARD: Great print quality at great speeds.

FAST: Good print quality at very fast speeds with a .4MM print tube (when a larger print tube is not appropriate for the task) but you are looking for faster speeds and normal layer heights.

DRAFT: Decent print quality at fast speeds.

FINE: Best quality, longest print times.

IMPORTANT: Not all material profiles have these presets. For some materials, the performance envelope of the material is not large enough to allow the very high flow rates required for "fast" and "draft" settings. An example of such a material is polycarbonate. We have focused on developing these presets for the most commonly-used and highest-impact materials.

IMPORTANT: Due to how carefully tuned the "fast", "draft", and "fine" presets are, we recommend you do not make changes to them. For instance, decreasing the infill beyond the minimums listed above will cause top infill issues. Increasing infill is usually safe. The "standard" preset is safer to make changes to, if you need to do something specific.

SLICING FOR EDGE

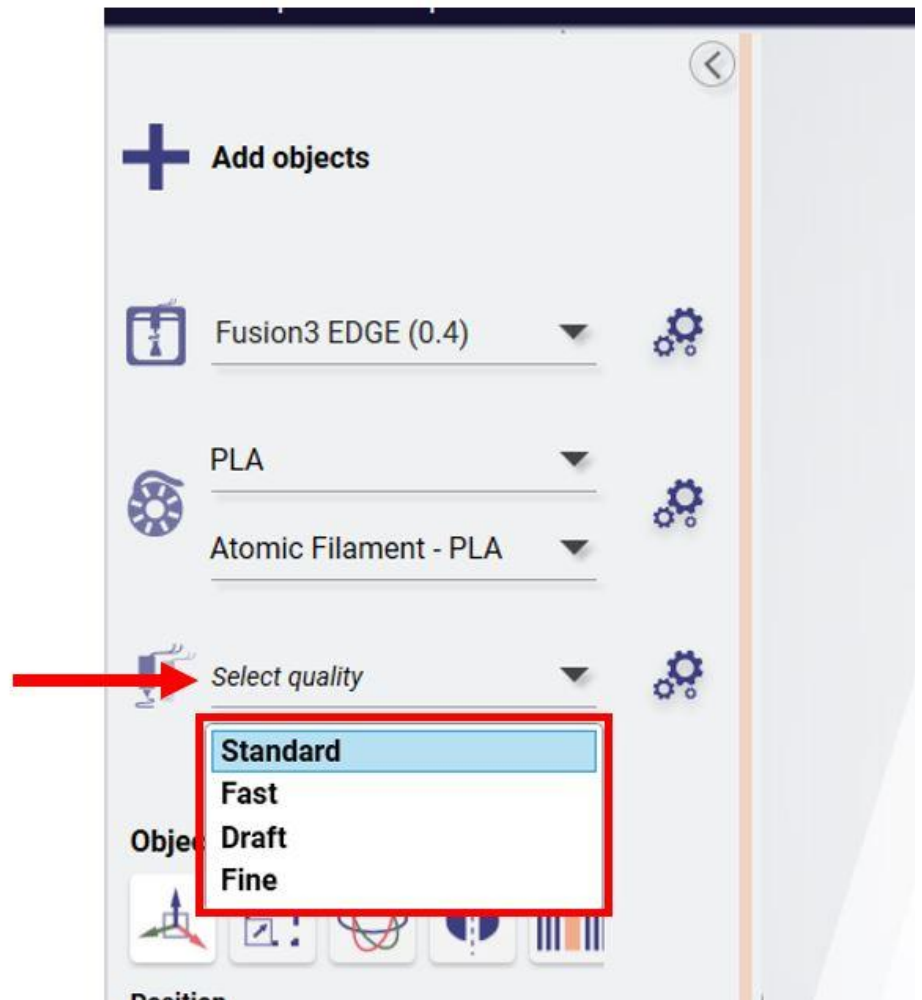
Using REACTOR to prepare your 3D printer output files is no different than normal except for two items:

1. Select EDGE as printer

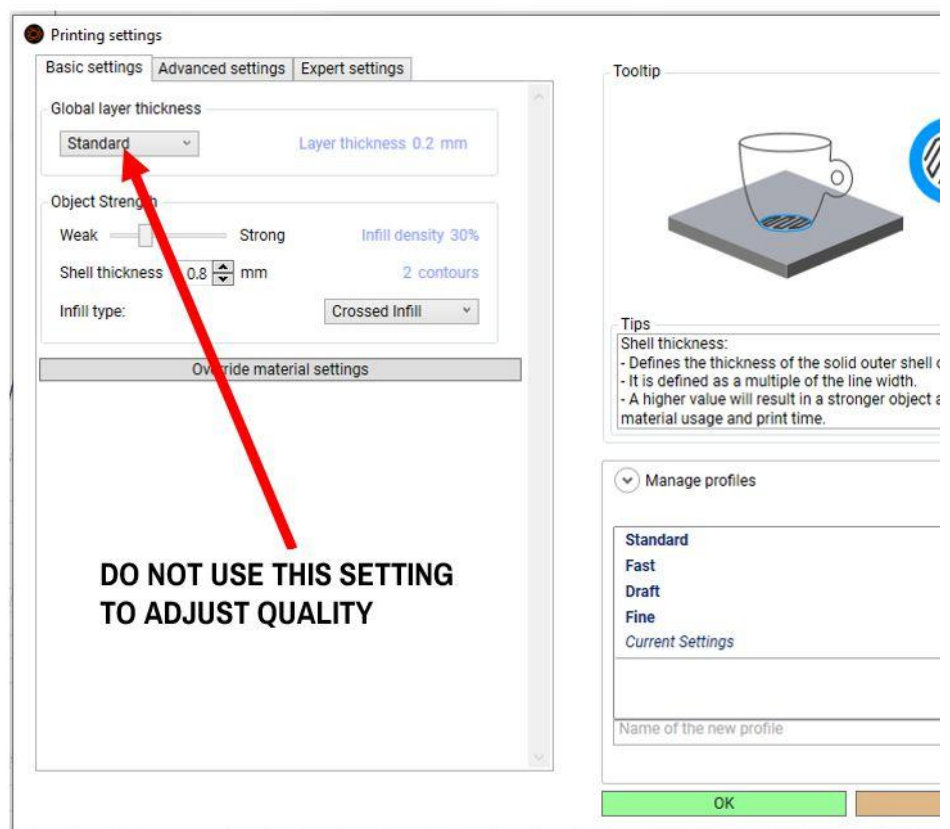
Three new Fusion3 3D printer options can be found in the top dropdown in the upper-left. You will see "EDGE 0.4MM", "EDGE 0.6MM" and "EDGE 0.8MM".

2. Quality Presets

Quality presets work differently with EDGE in REACTOR. Select your print quality preset on the left hand side of the main screen under quality settings.



NOTE: If you had previously used REACTOR, you would select your print quality by selecting from the pull down menu in the *Print Settings Menu > Basic settings > Global Layer Thickness*. Do not use this setting at all!



DO NOT USE THIS SETTING TO ADJUST QUALITY

Related Articles / Resources:

Tags: edge, reactor, slicing