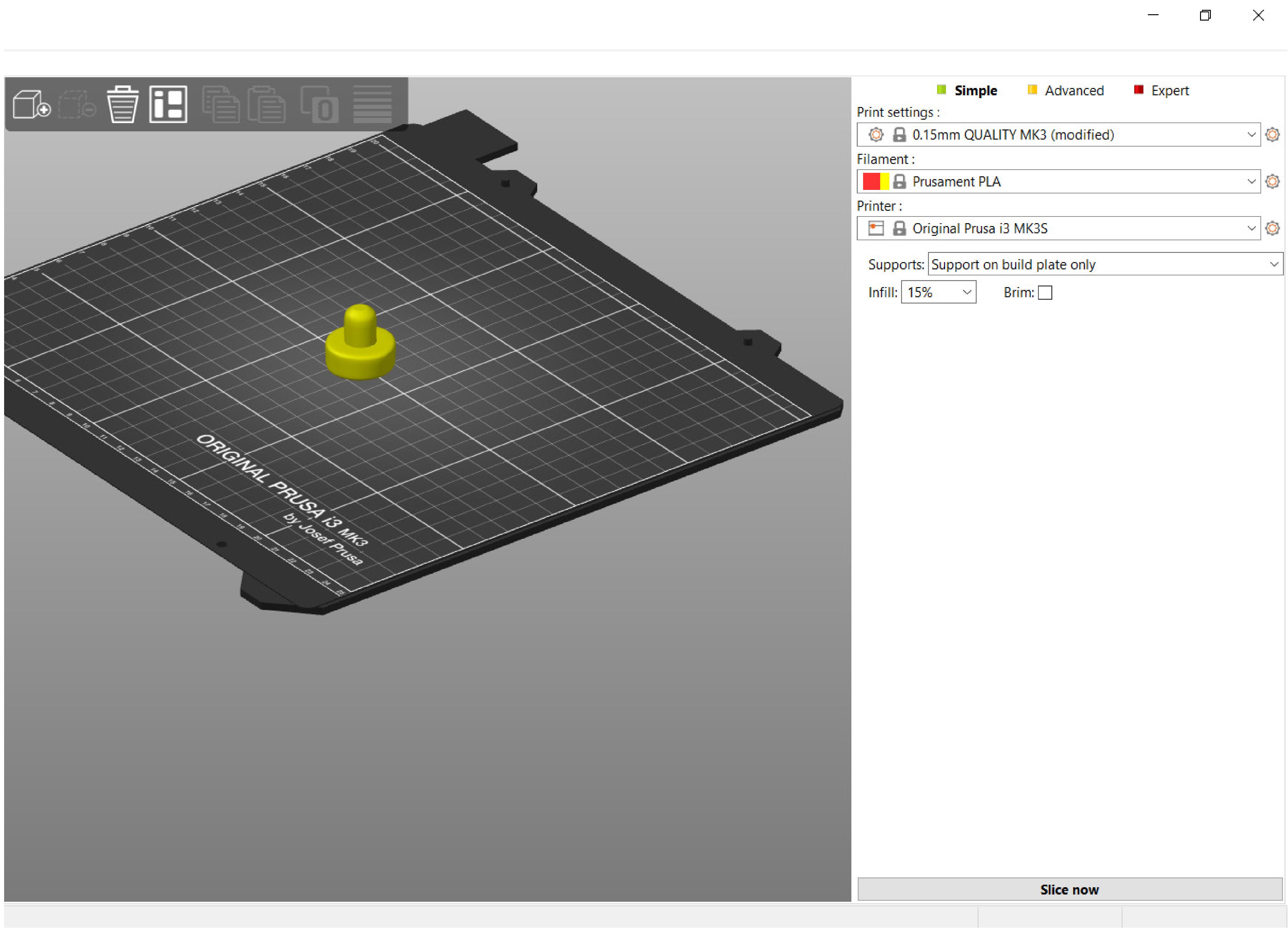
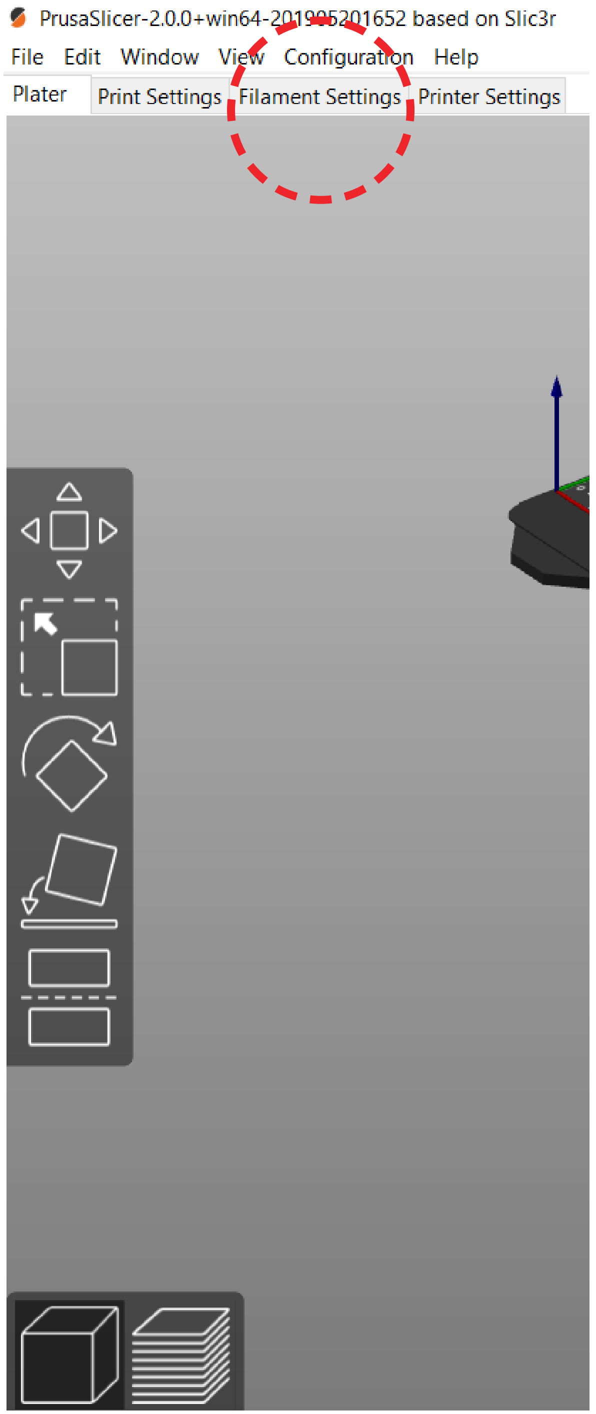


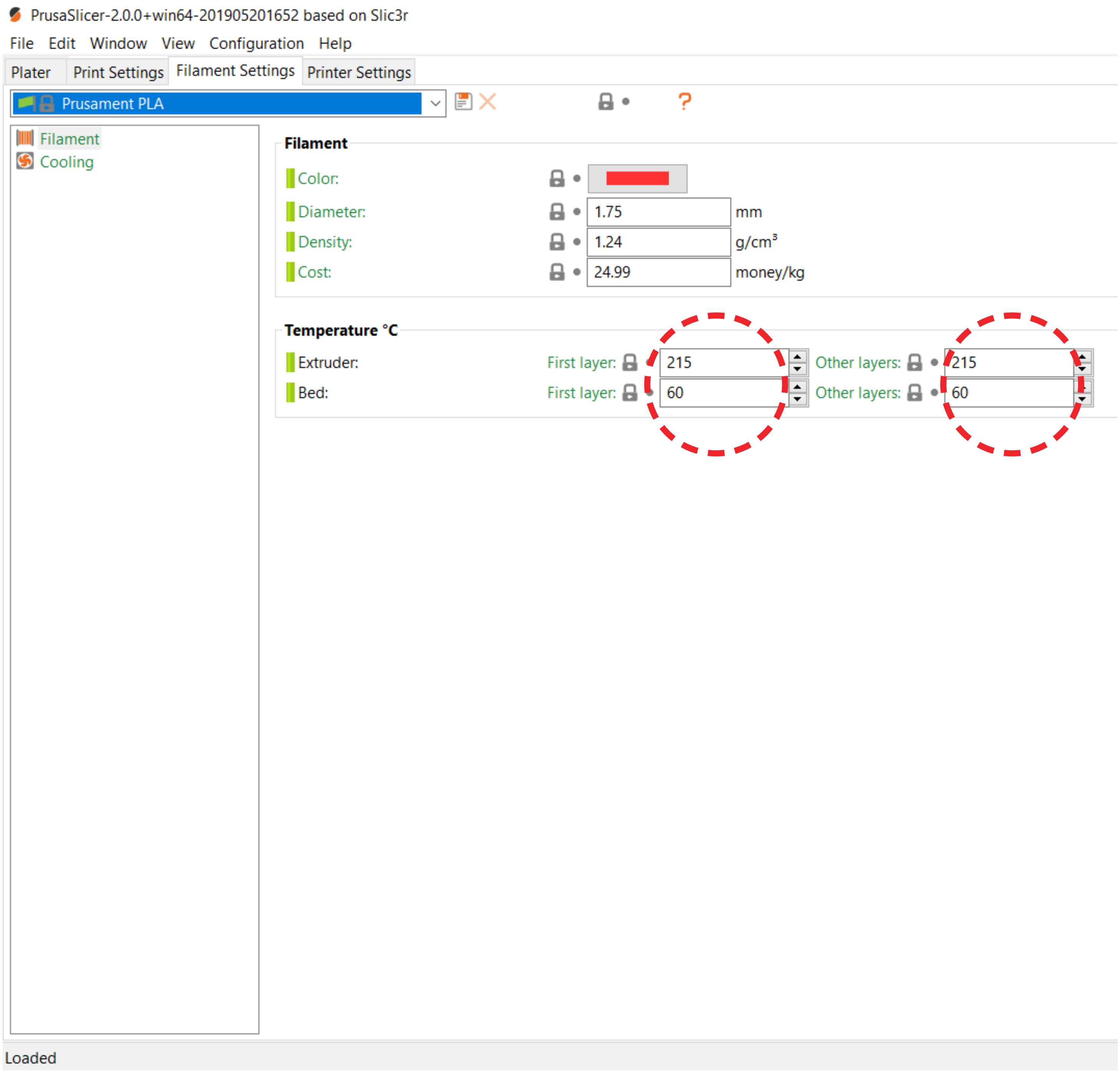
Depending on the quality you desire for your print, select the layer height - the usual one is - 0.15 mm QUALITY MK3 (modified), which should be enough for a normal print, if you are not doing texturing.

You also need to decide how many supports you need - for a simple model, supports on build plate only is enough. For a model with many cantilevers, you should choose either the “For support enforcers only” or “Everywhere” options.

You also need to decide what percentage of infill your model needs. Infill represents a space filling curve, which will determine how heavy and solid your model is. For a simple prototype, 15 % is enough.

If you are printing with PLA, the default material, the only thing you need to do now is to select the SLICE NOW option and EXPORT GCODE. If you are printing with another material, please check the next page.





1. Select the Filament Settings option.

2. Once in this menu, you will have to set the temperature settings for the material you are printing with. Make sure to use for both layers and other layers. The settings are as follows:

Wood filament: - Extruder: 180
- Bed: 60

Cork filament: - Extruder: 240
- Bed: 60

Filaflex filament: - Extruder: 230
- Bed: 50

Beer filament: - Extruder: 190
- Bed: 45

Trash filament: - Extruder: 190
- Bed: 45

Glass filament: - Extruder: 200
- Bed: 40

Conductive filament: - PLA Standard settings