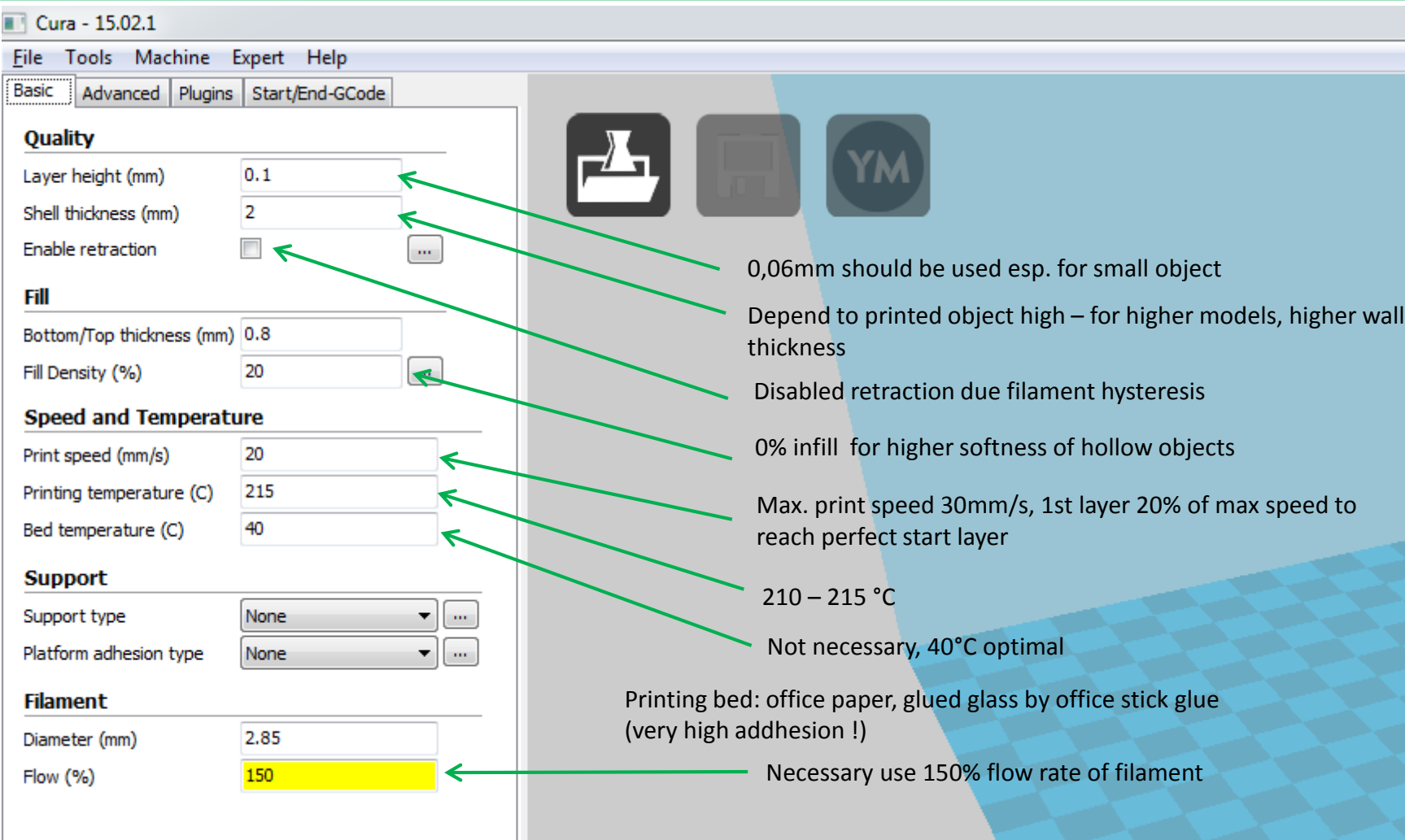


# User guide

For printing with **Eco<sup>3D</sup> Flex filament**



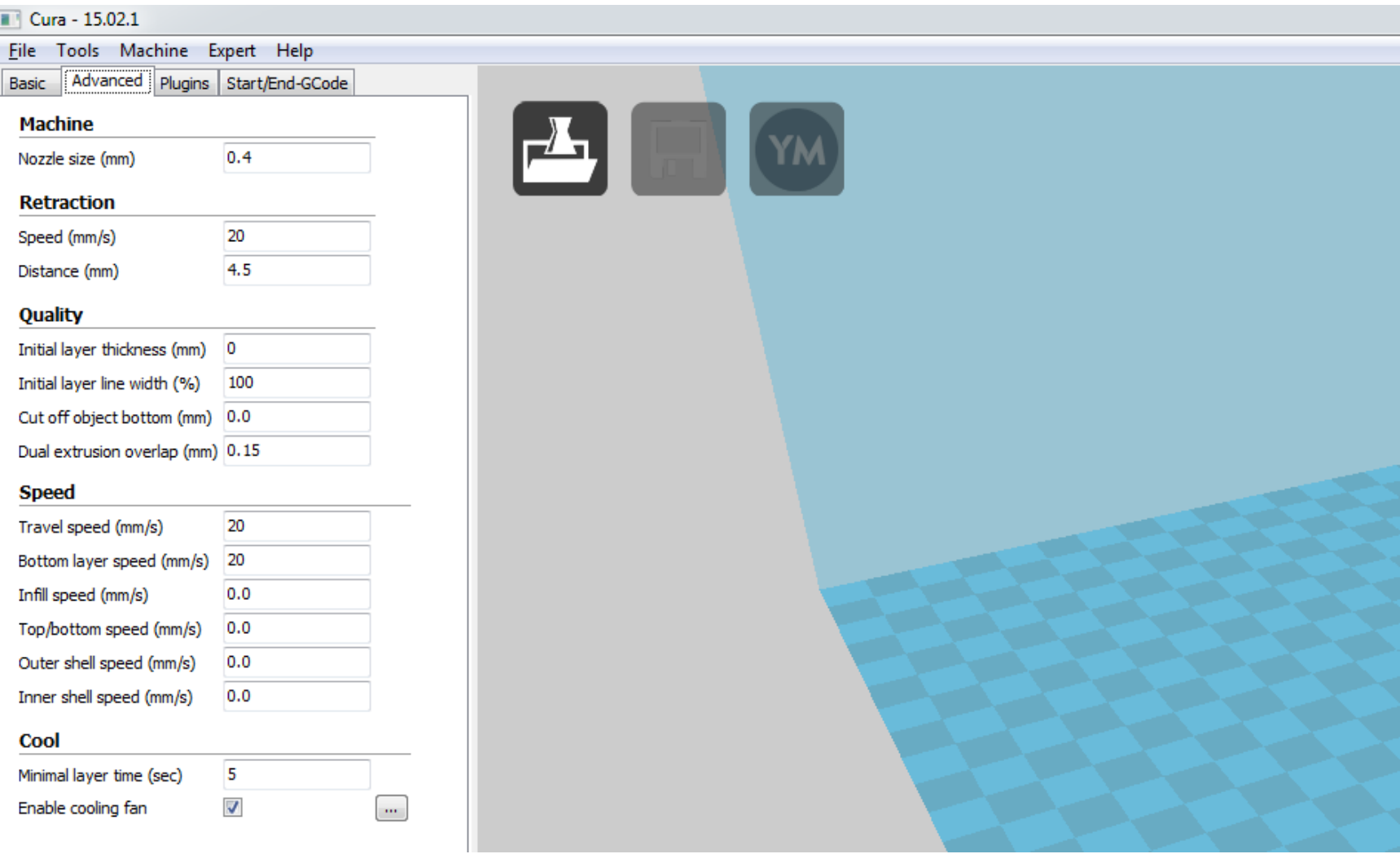


The screenshot shows the Cura 15.02.1 interface with the 'BASIC' tab selected. The settings are as follows:

- Quality:** Layer height (mm) is 0.1; Shell thickness (mm) is 2; Enable retraction is unchecked.
- Fill:** Bottom/Top thickness (mm) is 0.8; Fill Density (%) is 20.
- Speed and Temperature:** Print speed (mm/s) is 20; Printing temperature (C) is 215; Bed temperature (C) is 40.
- Support:** Support type is None; Platform adhesion type is None.
- Filament:** Diameter (mm) is 2.85; Flow (%) is 150.

Annotations with arrows point to specific settings:

- 0,06mm should be used esp. for small object (points to Layer height)
- Depend to printed object high – for higher models, higher wall thickness (points to Shell thickness)
- Disabled retraction due filament hysteresis (points to Enable retraction)
- 0% infill for higher softness of hollow objects (points to Fill Density)
- Max. print speed 30mm/s, 1st layer 20% of max speed to reach perfect start layer (points to Print speed)
- 210 – 215 °C (points to Printing temperature)
- Not necessary, 40°C optimal (points to Bed temperature)
- Printing bed: office paper, glued glass by office stick glue (very high adhesion !)
- Necessary use 150% flow rate of filament (points to Flow %)



**In case of any questions with printing, do not hesitate to contact us:**

[info@vivasadesign.com](mailto:info@vivasadesign.com)