ElegooSlicer Instructions
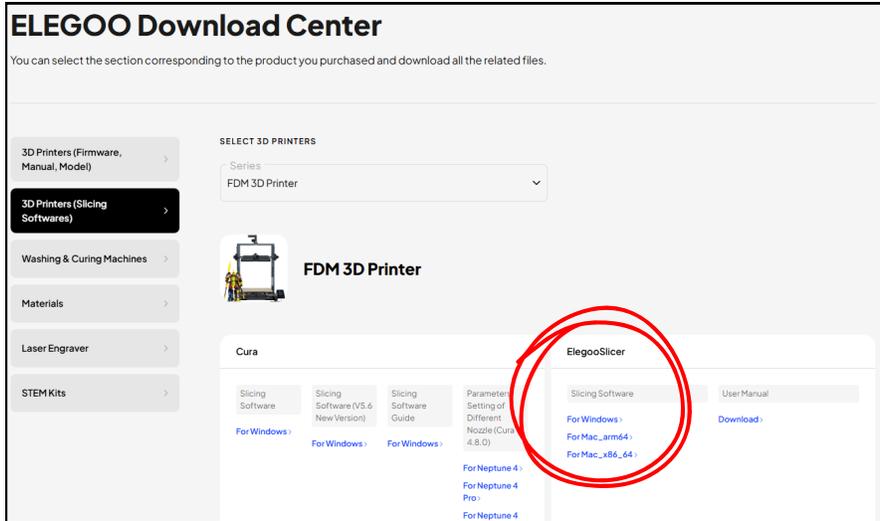
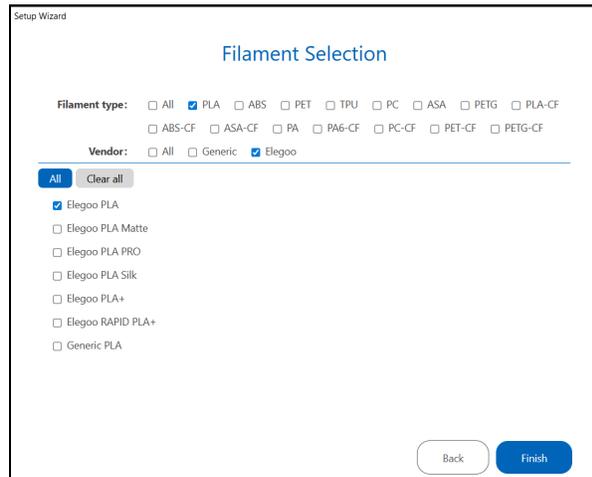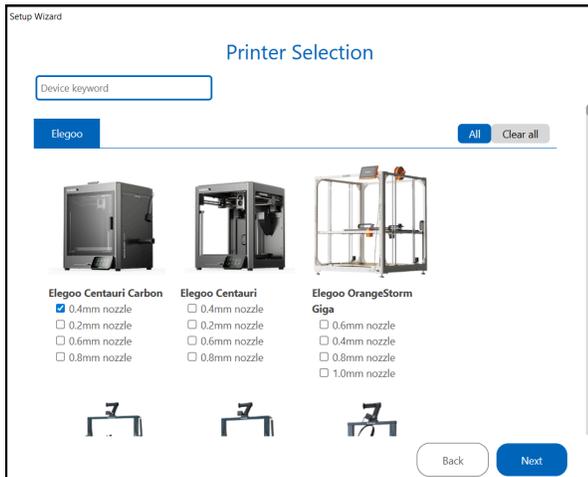*For more detailed instructions, see the official manual <u>here</u>.

Download the ElegooSlicer from <u>www.elegoo.com</u> and install it onto your computer.

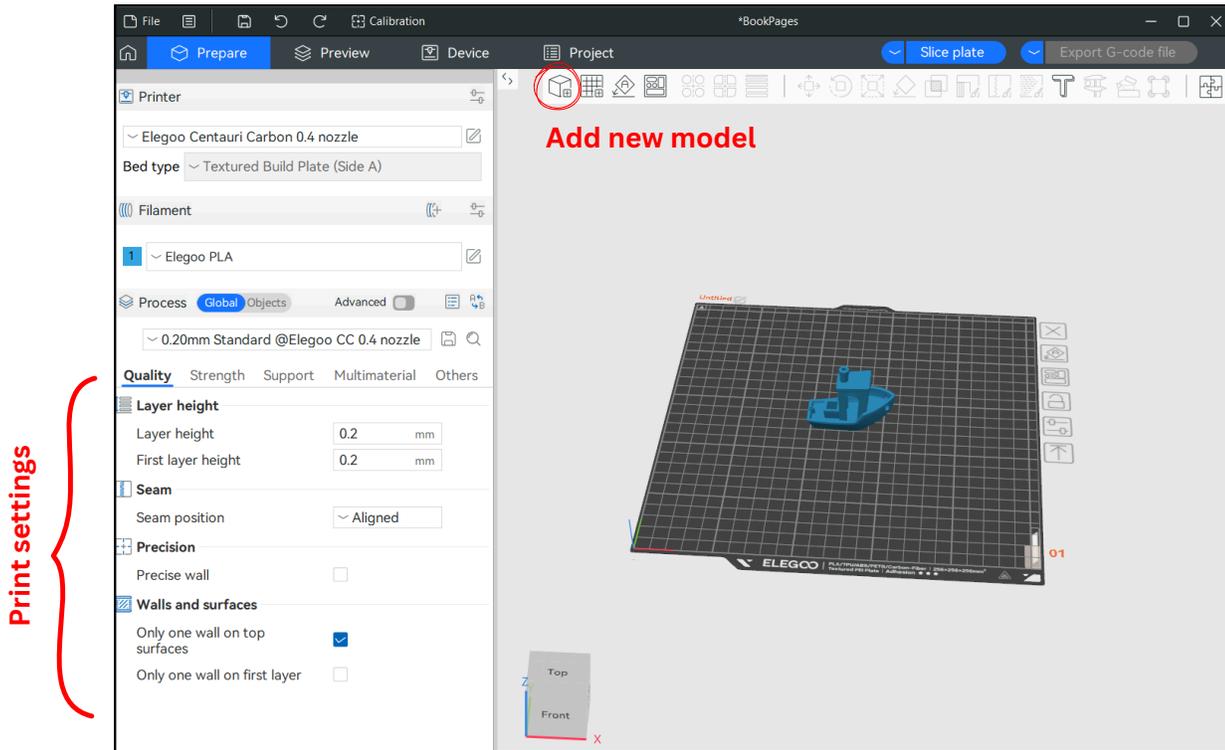Once the software is installed, follow the prompts to get startd.

Printer Selection: choose Elegoo Centauri Carbon, 0.4mm nozzle
Filament Selection: PLA, Elegoo PLA

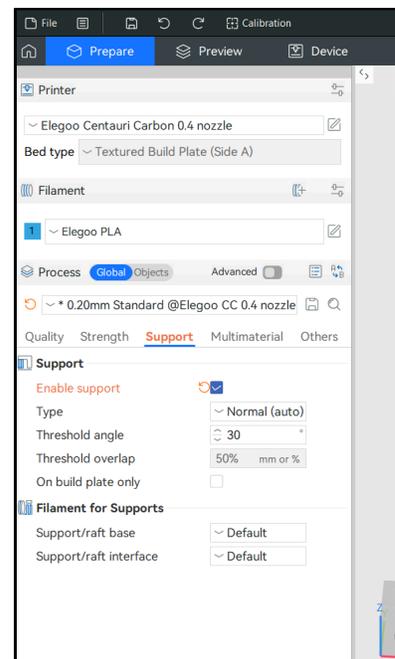Select **Prepare** on the top menu bar to navigate to the preparation interface.

Select the cube icon from the tool bar to add a file to the plate.

Modify print settings as desired. We recommend keeping the default settings.
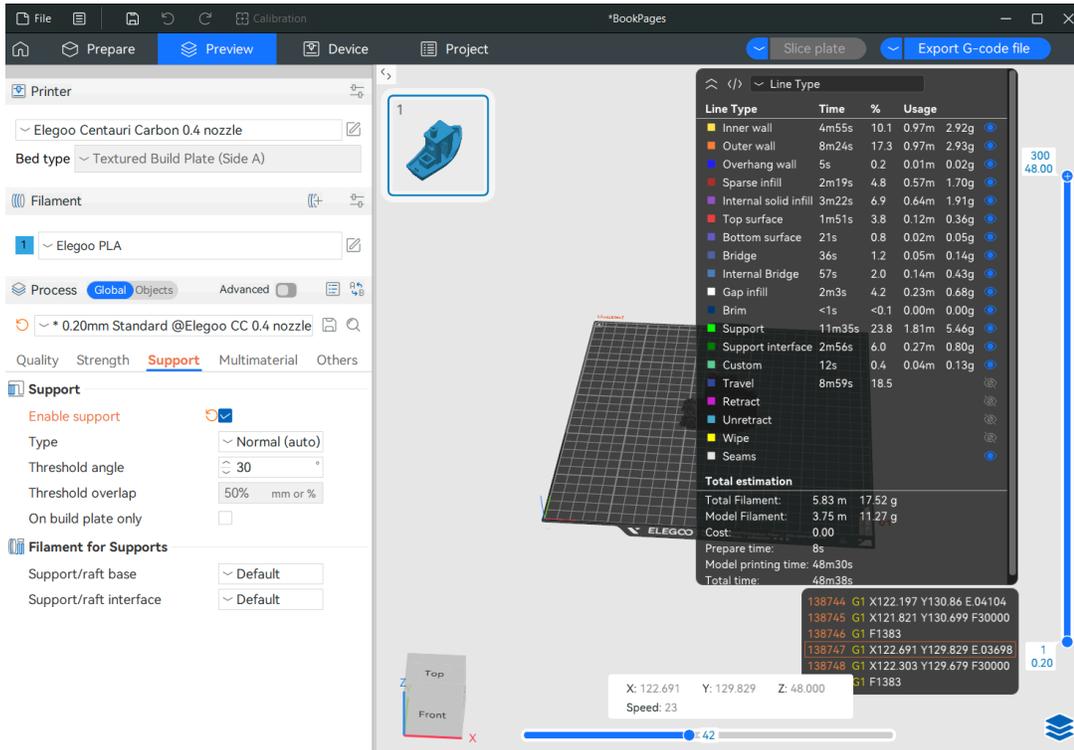
⚠️ If your object has any overhanging parts (think of the arms of a T), make sure **Enable Support** is selected. The software will generate supports to print with your object, and they can be removed after printing.

To see how long the print will take and how much filament it will use, select **Slice plate** on the top right.

If you need to go back and adjust print settings (e.g. decrease infill density to reduce print time), click **Prepare** on the top menu bar.

When you are ready to export your file, select **Export G-code file** and choose where you would like to save the downloaded file.

For more detailed instructions, including details about print settings, see the official ElegooSlicer manual <u>here</u>.