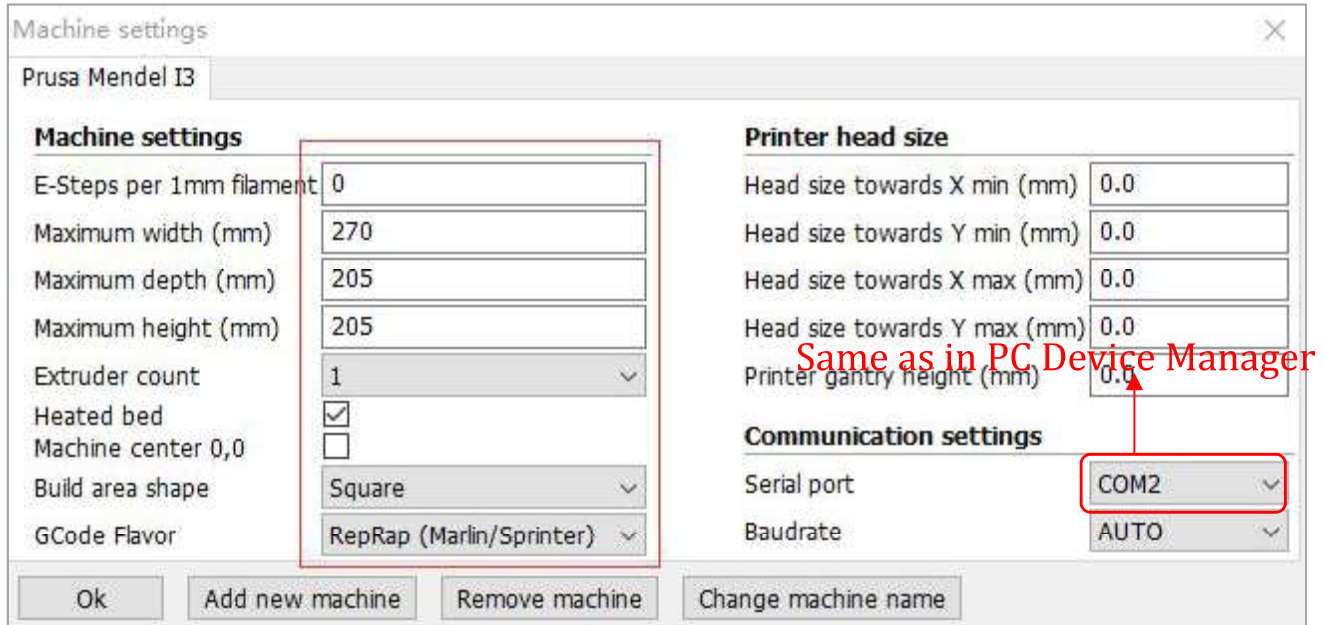


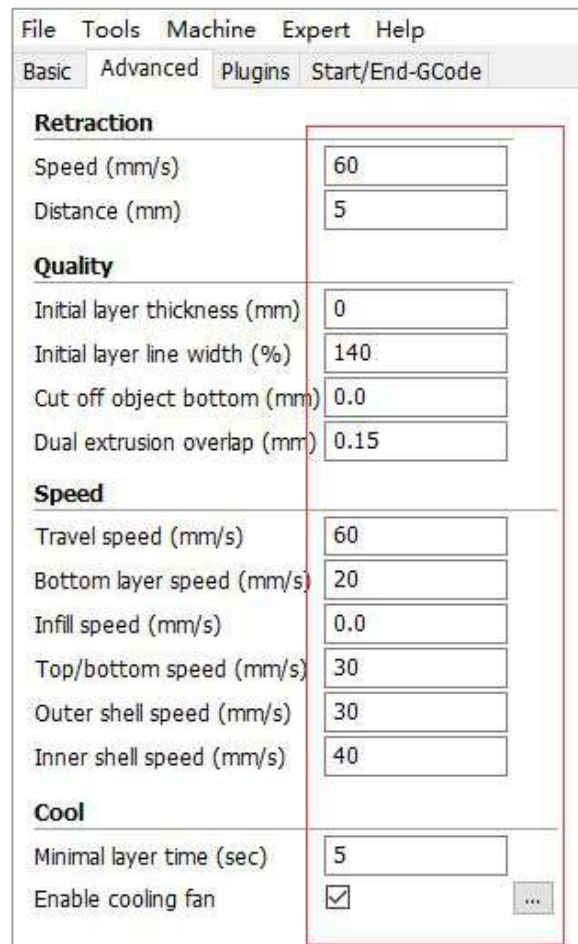
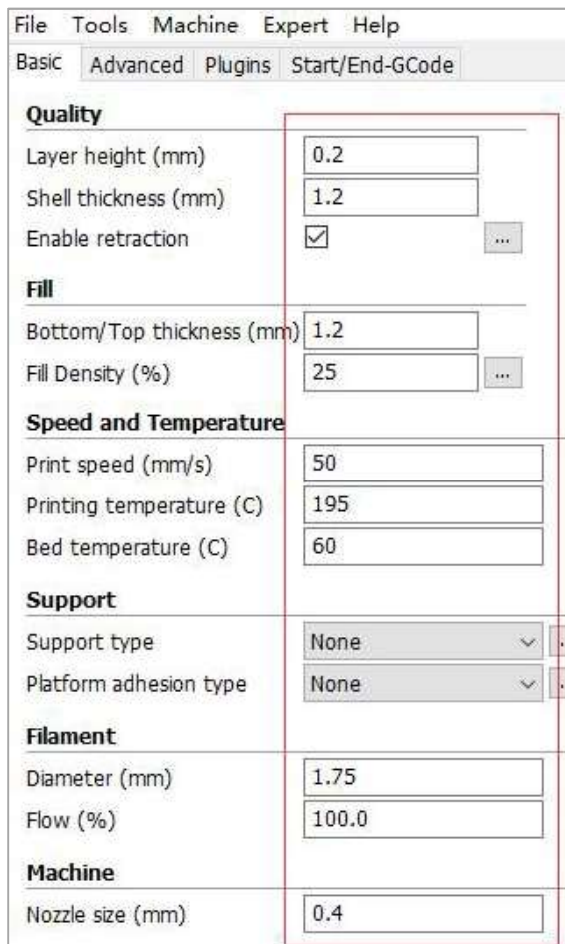
Introduction to slicing software

3. Cura settings

(1) In the menu bar, select “Machine” → “Machine settings”. Please choose the same Serial (**COM**) Port as shown in your PC → Device Manager → Port (customers may have a different COMx other than the example COM2 below).



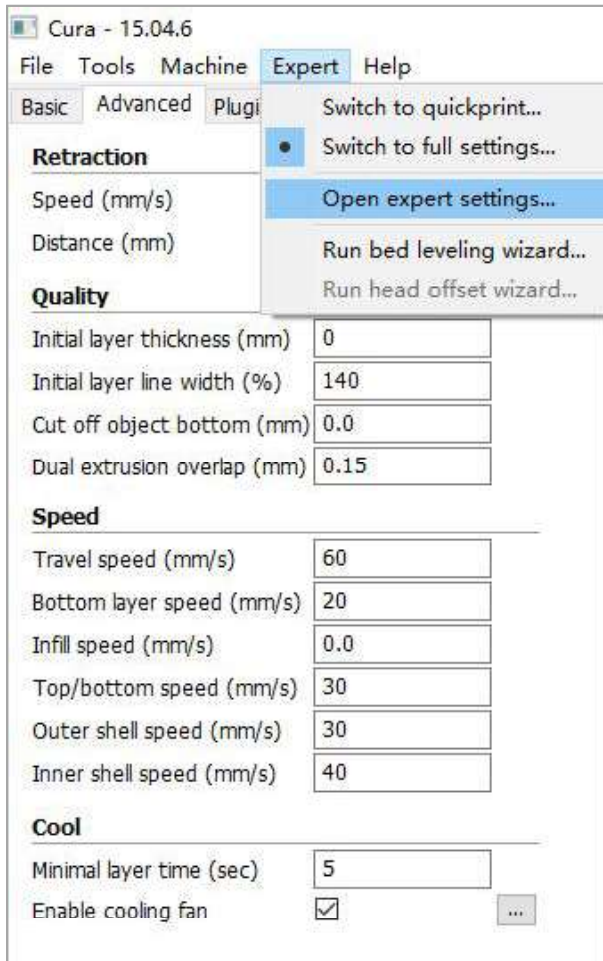
(2) Click “OK” for the settings to return to the main interface, and then set the “Basic” and “Advanced” parameters separately (For PLA filament), as shown below:



Introduction to slicing software

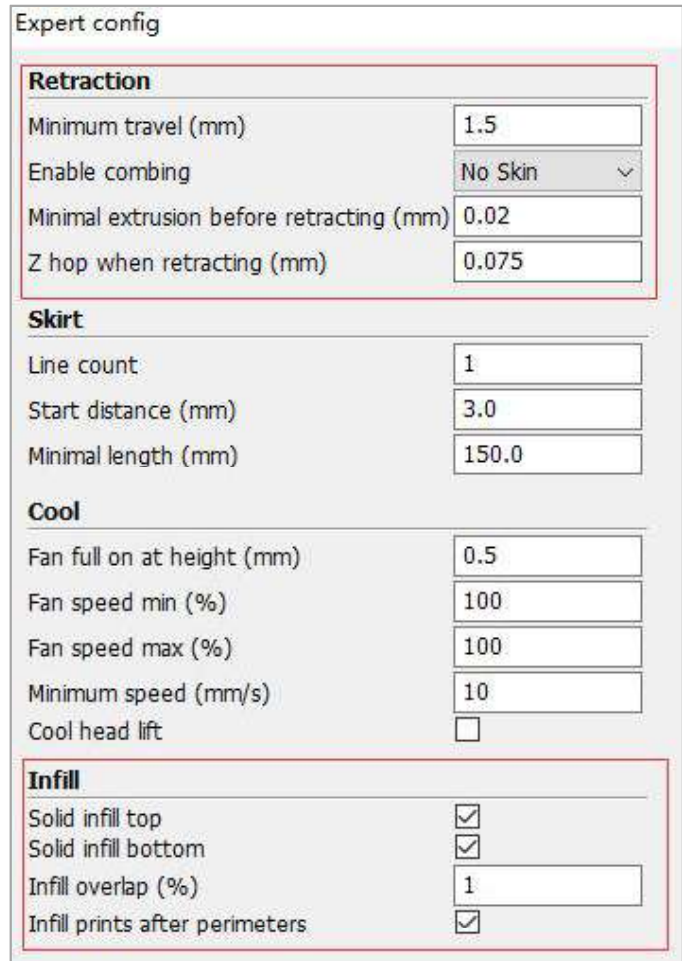
(3) In the menu bar, select “Expert” → “Open expert settings ”, and then set the parameters separately, as shown below:

Those parameters are just for example and reference, users may have to fine tune those parameters to gain the best printing results.



The screenshot shows the Cura 15.04.6 interface. The menu bar includes File, Tools, Machine, Expert, and Help. The Expert menu is open, showing options: Switch to quickprint..., Switch to full settings..., Open expert settings... (highlighted), Run bed leveling wizard..., and Run head offset wizard... The settings panel on the right is divided into sections: Retraction (Speed (mm/s), Distance (mm)), Quality (Initial layer thickness (mm), Initial layer line width (%), Cut off object bottom (mm), Dual extrusion overlap (mm)), Speed (Travel speed (mm/s), Bottom layer speed (mm/s), Infill speed (mm/s), Top/bottom speed (mm/s), Outer shell speed (mm/s), Inner shell speed (mm/s)), and Cool (Minimal layer time (sec), Enable cooling fan).

| Section | Parameter | Value |
|------------|------------------------------|-------------------------------------|
| Retraction | Speed (mm/s) | |
| | Distance (mm) | |
| Quality | Initial layer thickness (mm) | 0 |
| | Initial layer line width (%) | 140 |
| | Cut off object bottom (mm) | 0.0 |
| | Dual extrusion overlap (mm) | 0.15 |
| Speed | Travel speed (mm/s) | 60 |
| | Bottom layer speed (mm/s) | 20 |
| | Infill speed (mm/s) | 0.0 |
| | Top/bottom speed (mm/s) | 30 |
| | Outer shell speed (mm/s) | 30 |
| | Inner shell speed (mm/s) | 40 |
| Cool | Minimal layer time (sec) | 5 |
| | Enable cooling fan | <input checked="" type="checkbox"/> |



The screenshot shows the Expert config settings panel. It is divided into sections: Retraction (Minimum travel (mm), Enable combing, Minimal extrusion before retracting (mm), Z hop when retracting (mm)), Skirt (Line count, Start distance (mm), Minimal length (mm)), Cool (Fan full on at height (mm), Fan speed min (%), Fan speed max (%), Minimum speed (mm/s), Cool head lift), and Infill (Solid infill top, Solid infill bottom, Infill overlap (%), Infill prints after perimeters).

| Section | Parameter | Value |
|------------|--|-------------------------------------|
| Retraction | Minimum travel (mm) | 1.5 |
| | Enable combing | No Skin |
| | Minimal extrusion before retracting (mm) | 0.02 |
| | Z hop when retracting (mm) | 0.075 |
| Skirt | Line count | 1 |
| | Start distance (mm) | 3.0 |
| | Minimal length (mm) | 150.0 |
| Cool | Fan full on at height (mm) | 0.5 |
| | Fan speed min (%) | 100 |
| | Fan speed max (%) | 100 |
| | Minimum speed (mm/s) | 10 |
| | Cool head lift | <input type="checkbox"/> |
| Infill | Solid infill top | <input checked="" type="checkbox"/> |
| | Solid infill bottom | <input checked="" type="checkbox"/> |
| | Infill overlap (%) | 1 |
| | Infill prints after perimeters | <input checked="" type="checkbox"/> |

Introduction to slicing software

4Max Pro is compatible with ABS filament, and we provide the settings as shown below (It is recommended to print in places with good air circulation)

The image displays two screenshots of the 4Max Pro slicing software interface, showing the 'Expert' settings for ABS filament. The left screenshot shows the 'Basic' tab, and the right screenshot shows the 'Advanced' tab. Red boxes and arrows highlight specific settings.

Basic Tab Settings:

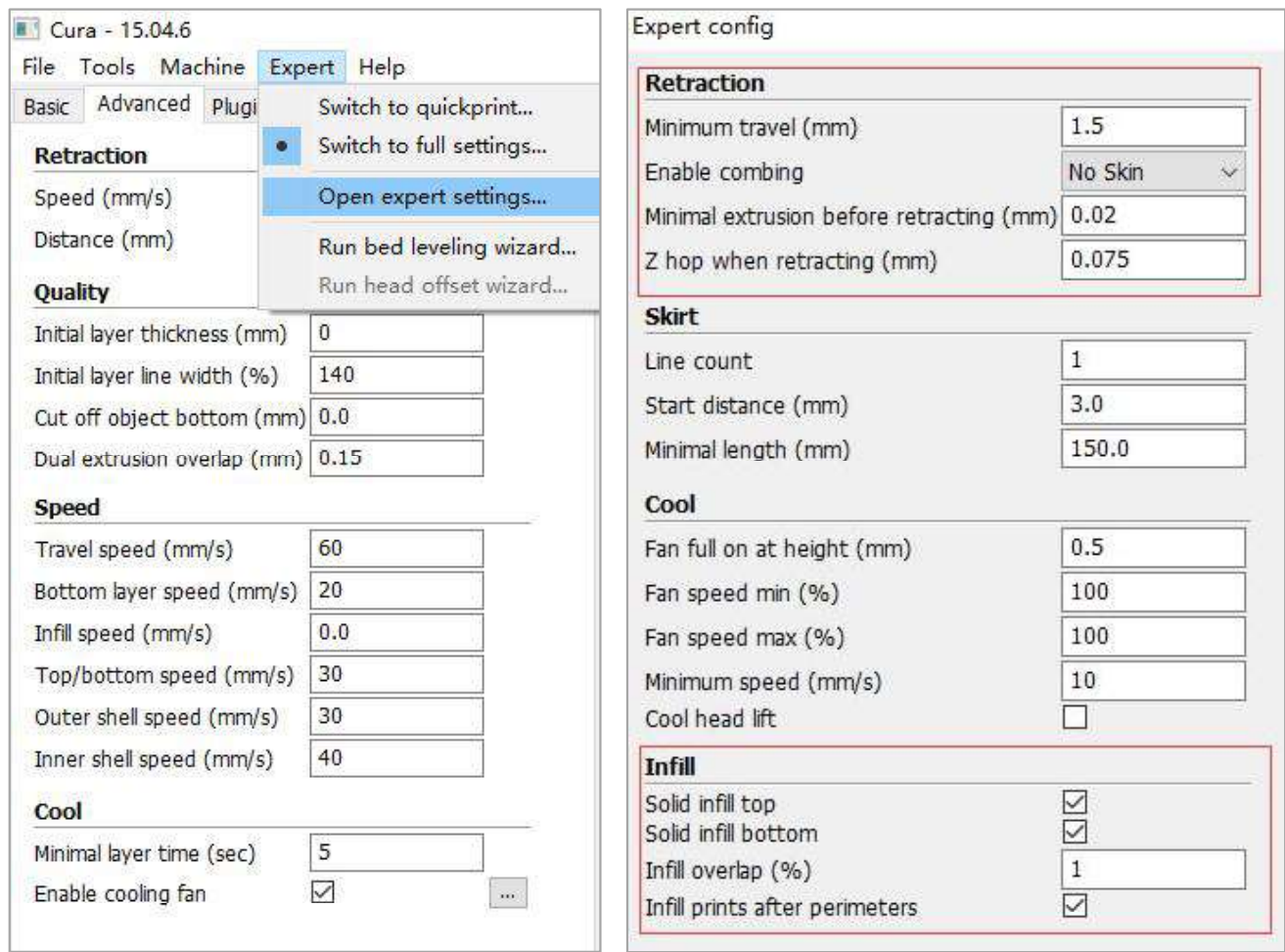
- Quality:** Layer height (mm) 0.2, Shell thickness (mm) 1.2, Enable retraction ☒
- Fill:** Bottom/Top thickness (mm) 1.2, Fill Density (%) 25
- Speed and Temperature:** Print speed (mm/s) 50, Printing temperature (C) 240, Bed temperature (C) 100
- Support:** Support type None, Platform adhesion type **Brim** (highlighted with a red box and arrow labeled "Add brim")
- Filament:** Diameter (mm) 1.75, Flow (%) 100.0
- Machine:** Nozzle size (mm) 0.4

Advanced Tab Settings:

- Retraction:** Speed (mm/s) 60, Distance (mm) 5
- Quality:** Initial layer thickness (mm) 0, Initial layer line width (%) 140, Cut off object bottom (mm) 0.0, Dual extrusion overlap (mm) 0.15
- Speed:** Travel speed (mm/s) 60, Bottom layer speed (mm/s) 20, Infill speed (mm/s) 0.0, Top/bottom speed (mm/s) 30, Outer shell speed (mm/s) 30, Inner shell speed (mm/s) 40
- Cool:** Minimal layer time (sec) 5, Enable cooling fan ☐ (highlighted with a red box and arrow labeled "Disable cooling fan")

In the menu bar, select "Expert" → "Open expert settings ", and then set the parameters separately, as shown below:

Introduction to slicing software



When setting the parameters of ABS filament, adding birm and disabling cooling fan helps the filament adhere the platform.

Introduction to slicing software

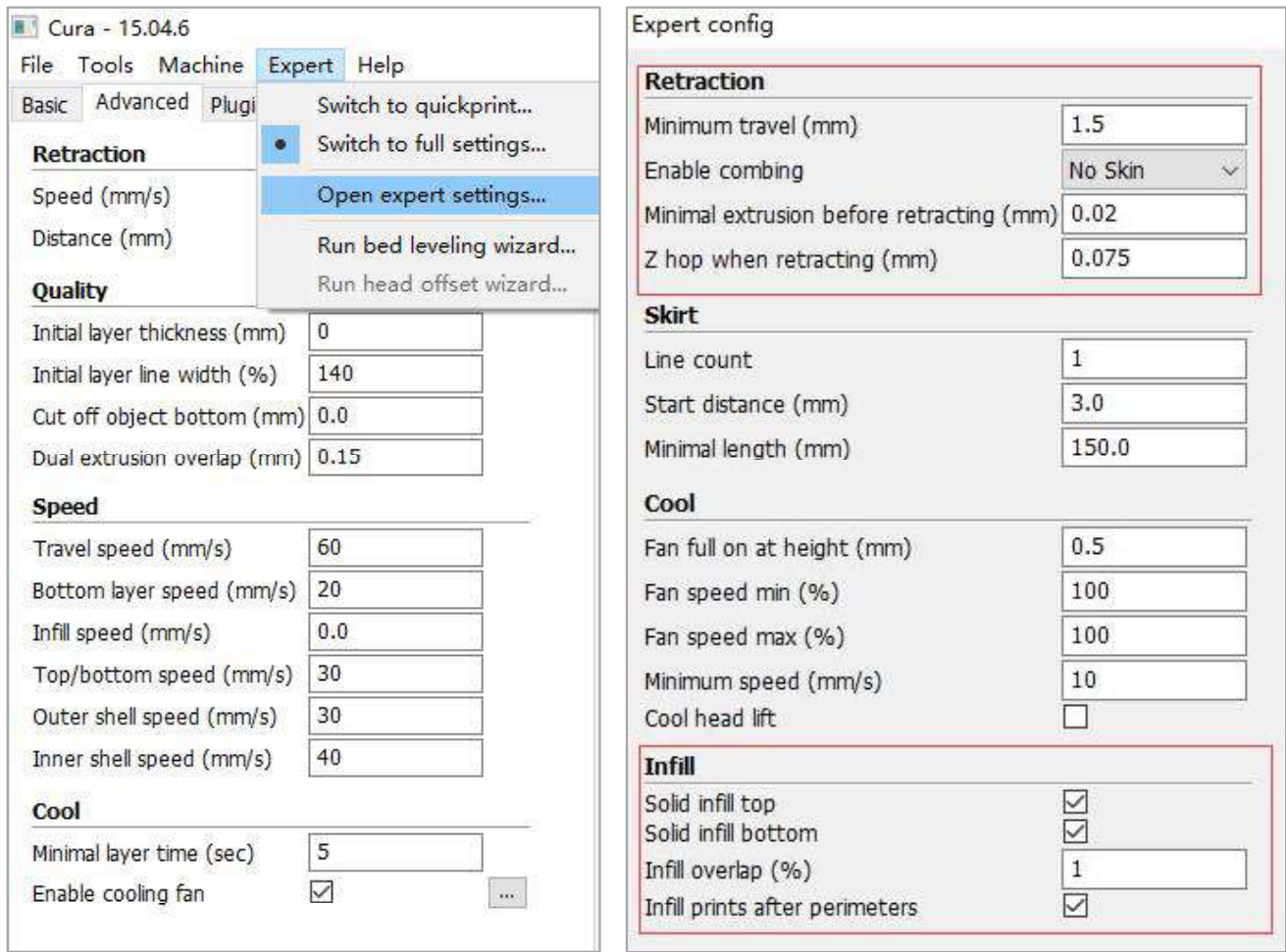
4Max Pro is compatible with flexible filament, and we provide the settings as shown below if using **ANYCUBIC** flexible filaments (**users may have to fine-tune the settings based on the actual printing conditions, and type of filaments, etc.**).

| File Tools Machine Expert Help | |
|--|-------------------------------------|
| Basic Advanced Plugins Start/End-GCode | |
| Quality | |
| Layer height (mm) | 0.2 |
| Shell thickness (mm) | 1.2 |
| Enable retraction | <input checked="" type="checkbox"/> |
| Fill | |
| Bottom/Top thickness (mm) | 1.2 |
| Fill Density (%) | 25 |
| Speed and Temperature | |
| Print speed (mm/s) | 50 |
| Printing temperature (C) | 195 |
| Bed temperature (C) | 60 |
| Support | |
| Support type | None |
| Platform adhesion type | None |
| Filament | |
| Diameter (mm) | 1.75 |
| Flow (%) | 100.0 |
| Machine | |
| Nozzle size (mm) | 0.4 |

| File Tools Machine Expert Help | |
|--|-------------------------------------|
| Basic Advanced Plugins Start/End-GCode | |
| Retraction | |
| Speed (mm/s) | 60 |
| Distance (mm) | 8 |
| Quality | |
| Initial layer thickness (mm) | 0 |
| Initial layer line width (%) | 140 |
| Cut off object bottom (mm) | 0.0 |
| Dual extrusion overlap (mm) | 0.15 |
| Speed | |
| Travel speed (mm/s) | 60 |
| Bottom layer speed (mm/s) | 20 |
| Infill speed (mm/s) | 0.0 |
| Top/bottom speed (mm/s) | 30 |
| Outer shell speed (mm/s) | 30 |
| Inner shell speed (mm/s) | 40 |
| Cool | |
| Minimal layer time (sec) | 5 |
| Enable cooling fan | <input checked="" type="checkbox"/> |

In the menu bar, select “Expert” → “Open expert settings ”, and then set the parameters separately, as shown below:

Introduction to slicing software



Explanation:

Layer height: determine the important parameters for print quality, usually set to 0.1-0.3.

Shell thickness: usually set to a multiple of the nozzle diameter.

Fill density: The larger the parameter, the more solid the model is.

Print speed: printing too fast may make printer shaky, 30-60 is suggested.

Printing temperature: PLA should be 190-210°C, ABS should be 230-240°C, and TPU should be 190-220 °C .

Filament diameter: 1.75mm.

Nozzle size: 0.4mm.

Retraction speed: Increasing the retraction speed and distance can reduce the stringing problem, but set it too high may cause clogging.

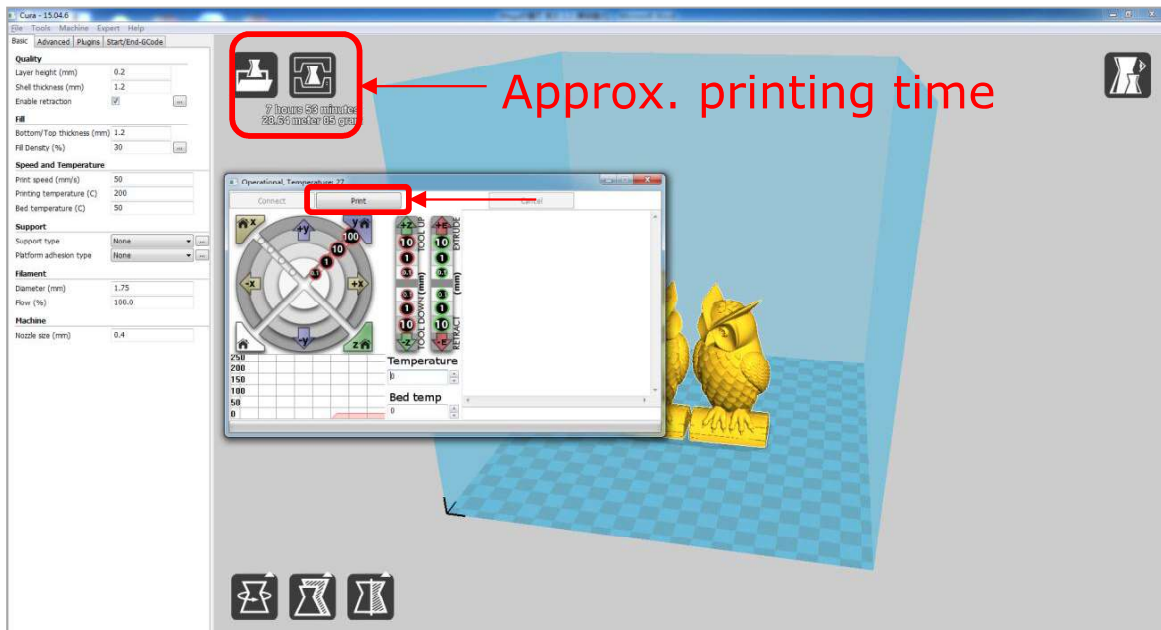
Travel speed: 60mm/s is suggested, the printing accuracy would be affected if it is too fast.

Outer shell speed: Reducing the outer shell print speed would create a more smooth model surface.

Introduction to slicing software

4. Print online

After the parameters have been set up, you can print online via Cura. As shown below, click on the upper left corner “File” → “Print”, Cura will automatically connect to the printer. The user can click “Print” icon when it is available. Then the temperature would rise and it will start to print when reaching to the target temperature. Use tweezers to carefully get rid of the pre-extruded filament at the nozzle tip.



5. Print offline

After completing all the parameter settings, click on the Cura software main interface “File” → “Save GCode”. Save the model GCode file to the **memory card**, and then insert the memory card to the printer and control via the touch screen for offline printing.

Note: the file name should only contain English letters, underscore and space. File name contains special characters could not be recognized by the printer.