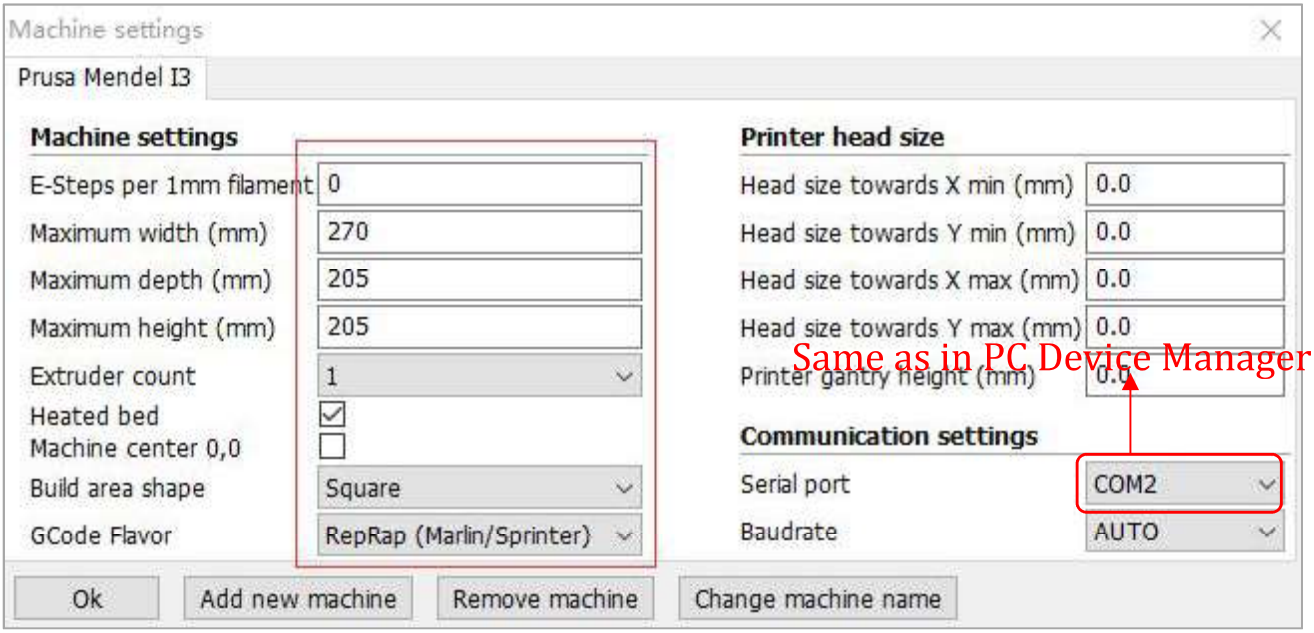
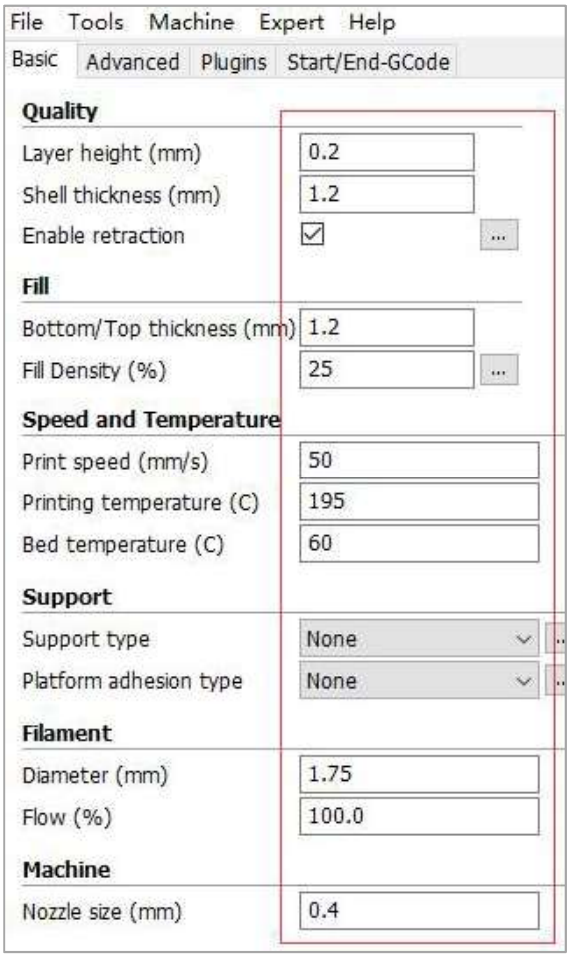
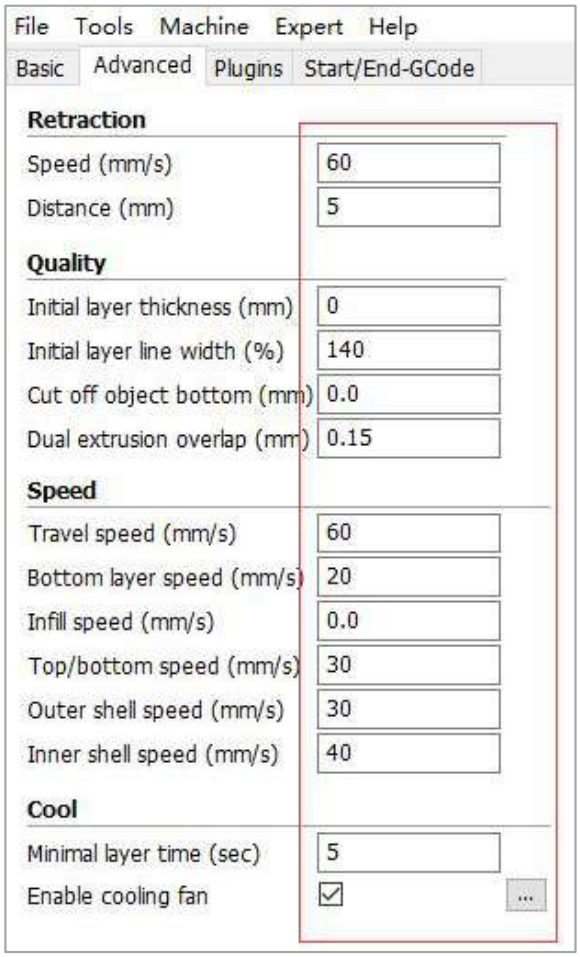
3. Cura settings

(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 COM*x* other than the example COM2 below).



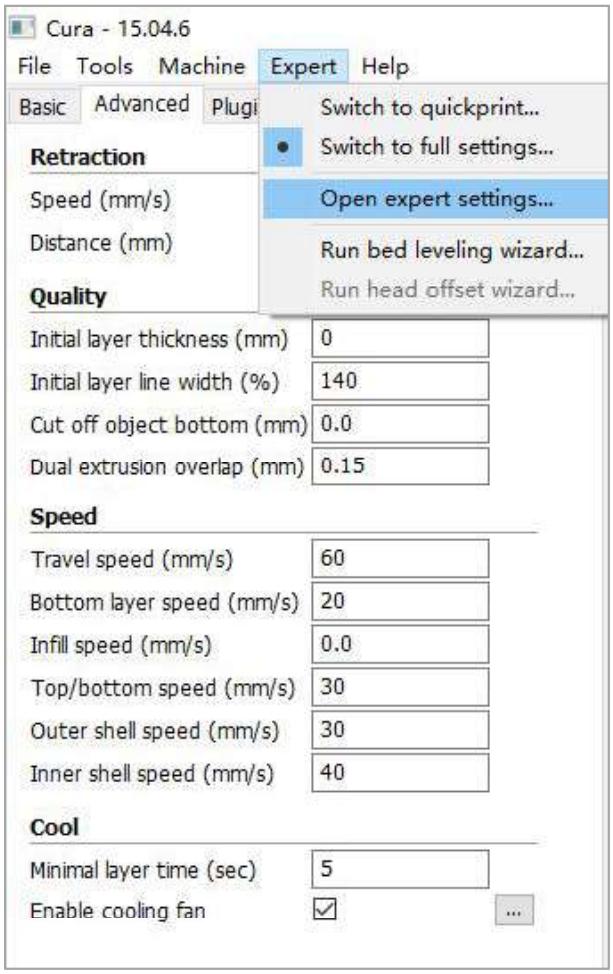
(2) Click “OK” for the settings to return to the main interface, and then set the “Basic” and “Advanced” parameters separately (For PLA filament), as shown below:

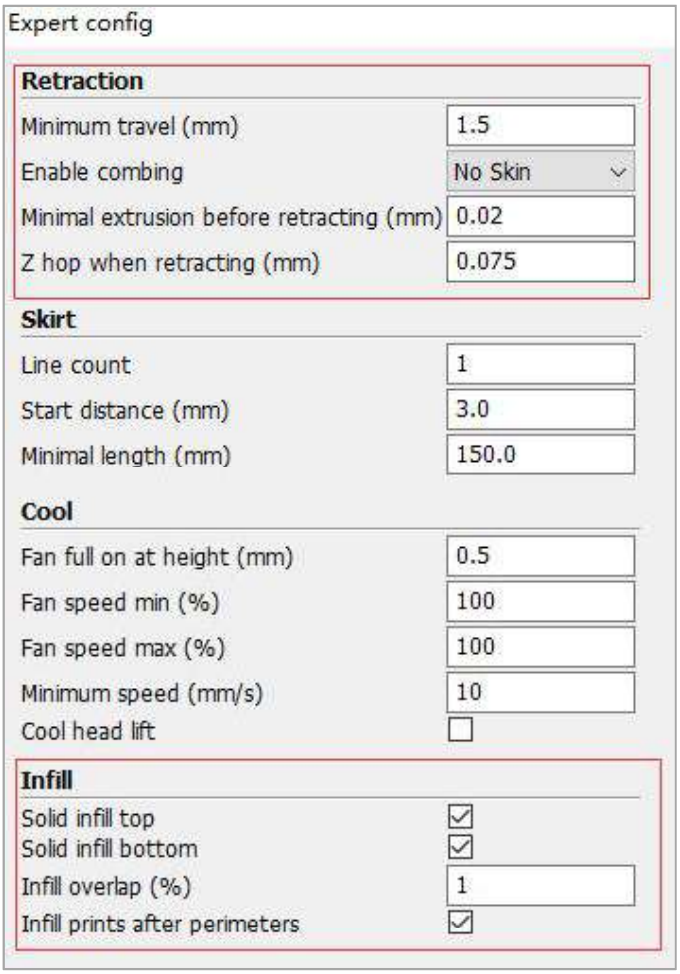




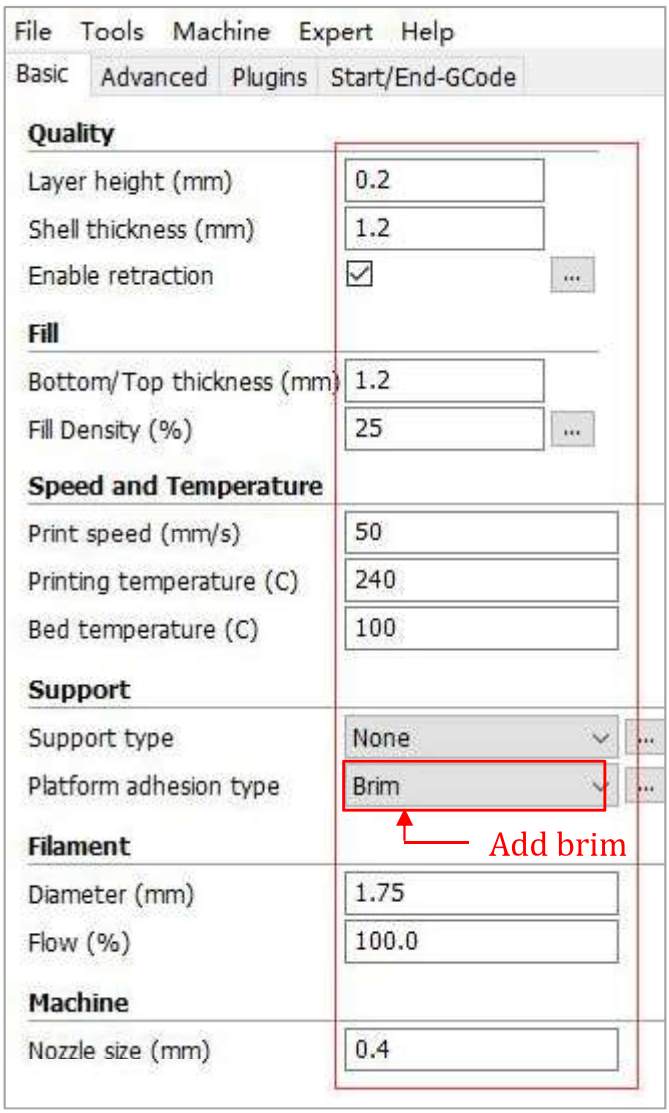
(3) In the menu bar, select “Expert” → “Open expert settings ”， and then set the parameters separately, as shown below:

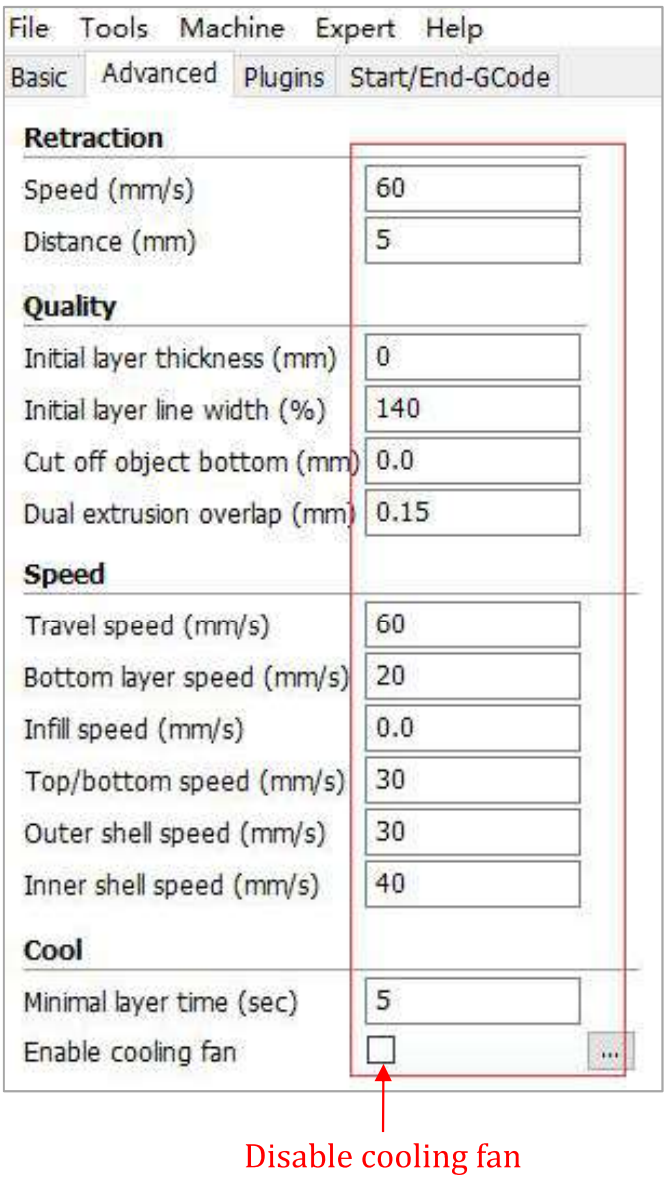
Those parameters are just for example and reference, users may have to fine tune those parameters to gain the best printing results.



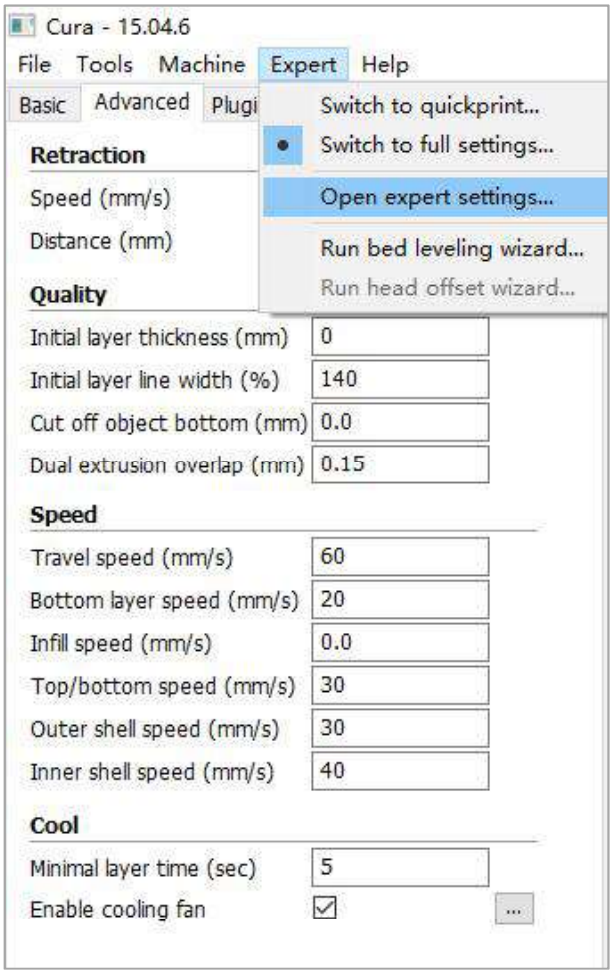


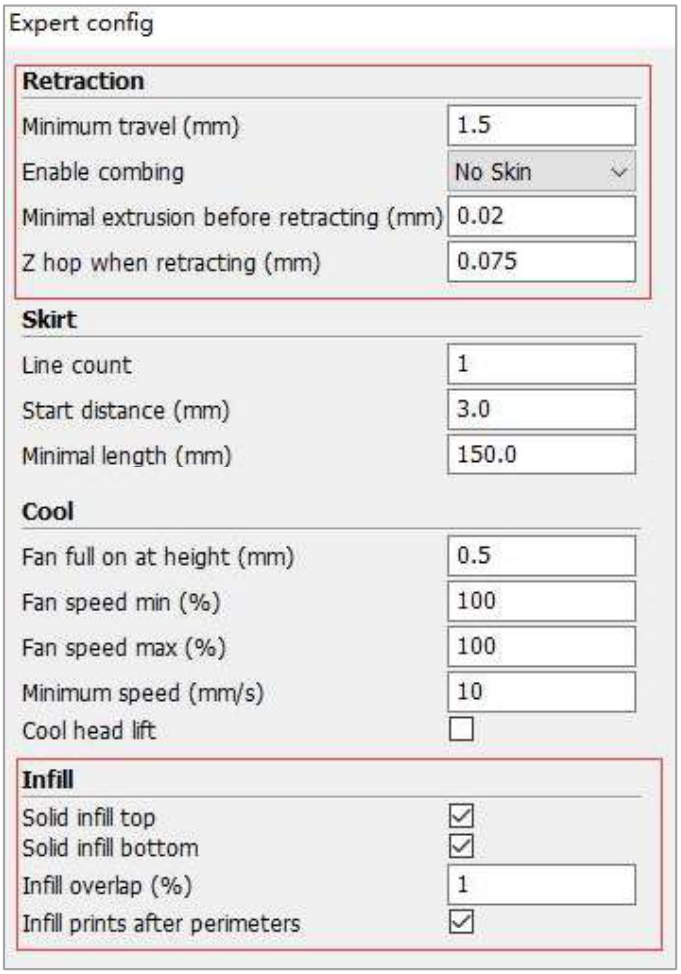
4Max Pro is compatible with ABS filament, and we provide the settings as shown below (It is recommended to print in places with good air circulation)





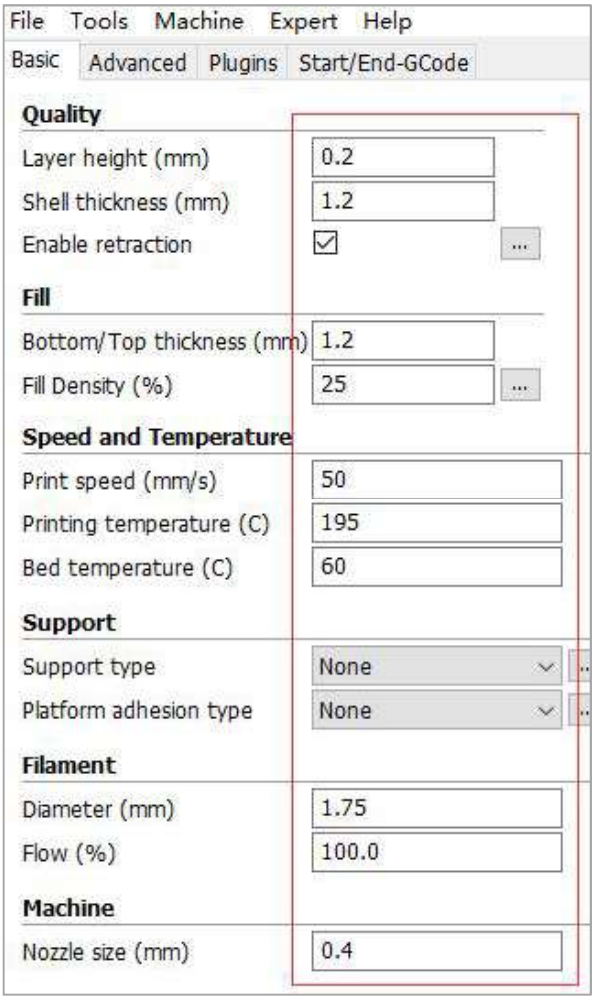
In the menu bar, select “Expert” → “Open expert settings ”， and then set the parameters separately, as shown below:

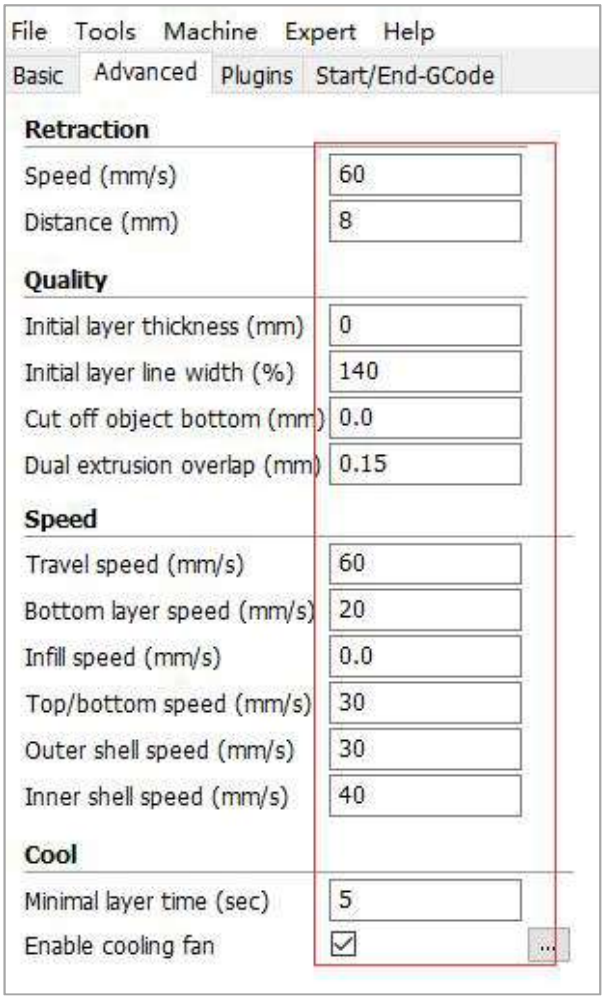




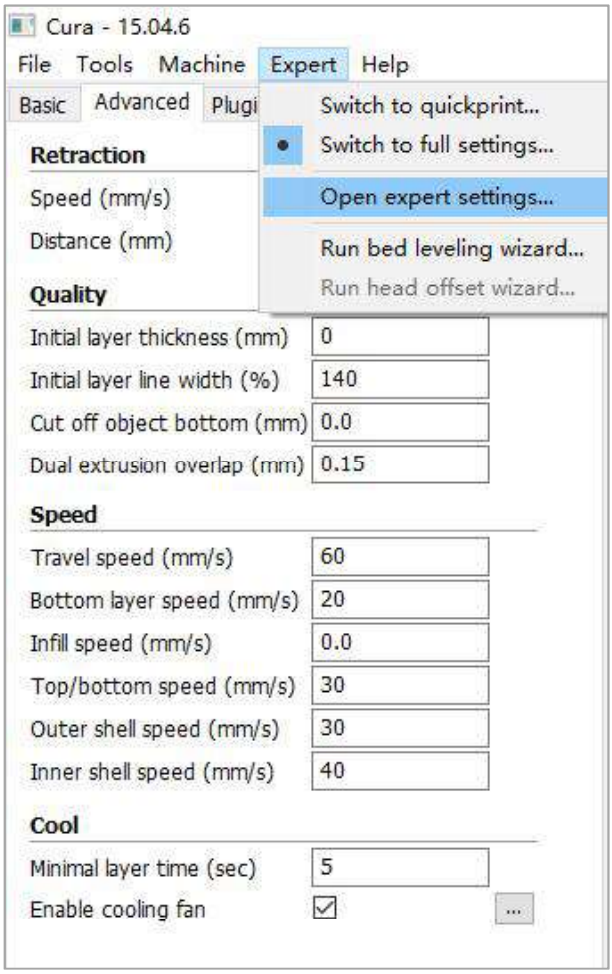
When setting the parameters of ABS filament, adding birm and disabling cooling fan helps the filament adhere the platform.

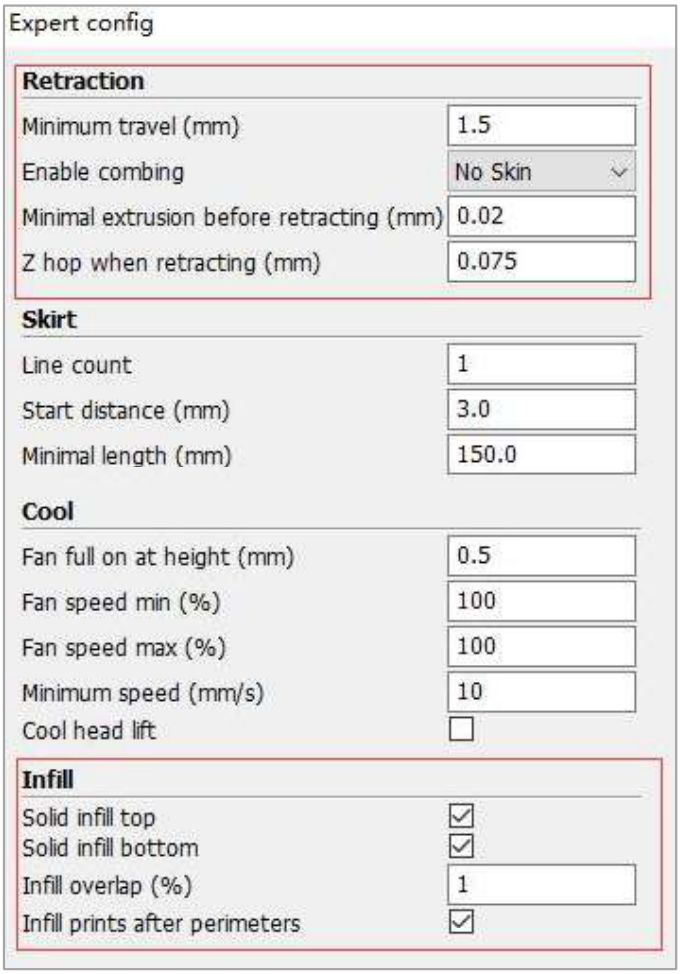
4Max Pro is compatible with flexible filament, and we provide the settings as shown below if using ANYCUBIC flexible filaments (**users may have to fine-tune the settings based on the actual printing conditions, and type of filaments, etc**.).





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





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℃, ABS should be 230-240℃, and TPU should be 190-220 ℃ .

Filament diameter: 1.75mm.

Nozzle size: 0.4mm.

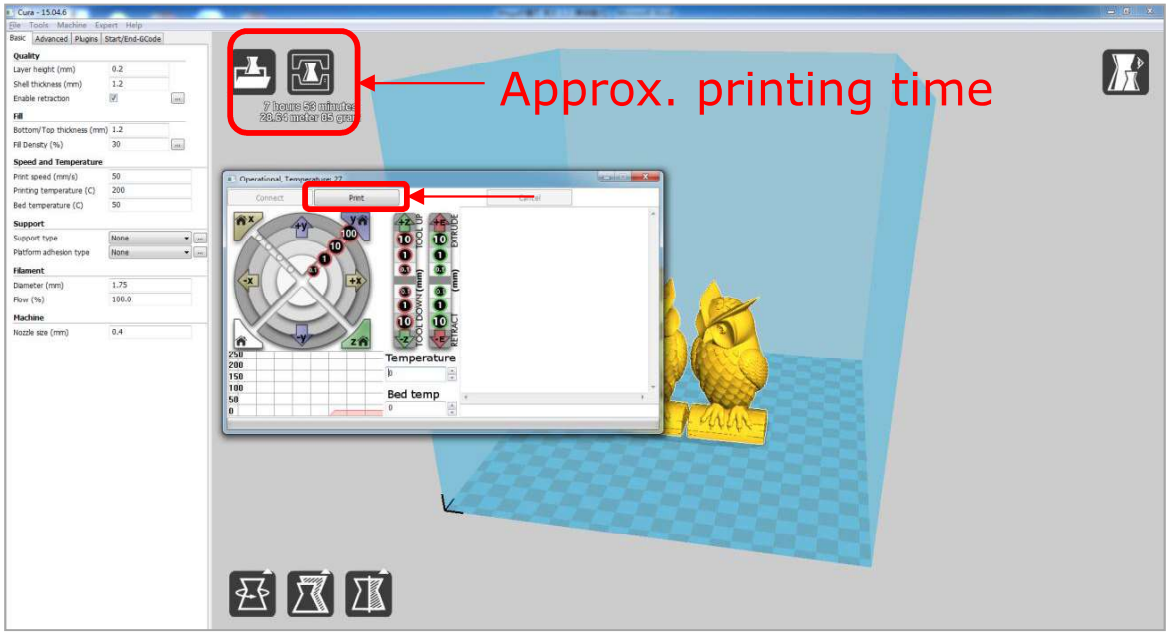
Retraction speed: Increasing the retraction speed and distance can reduce the stringing problem, but set it too high may cause clogging.

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.

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.