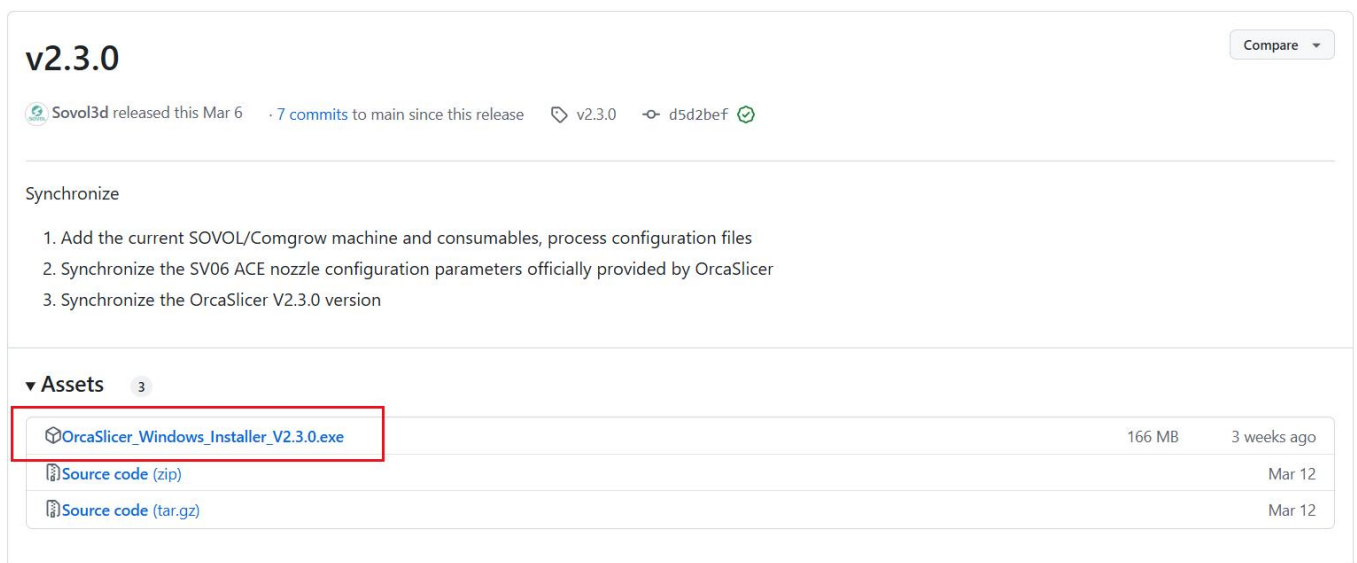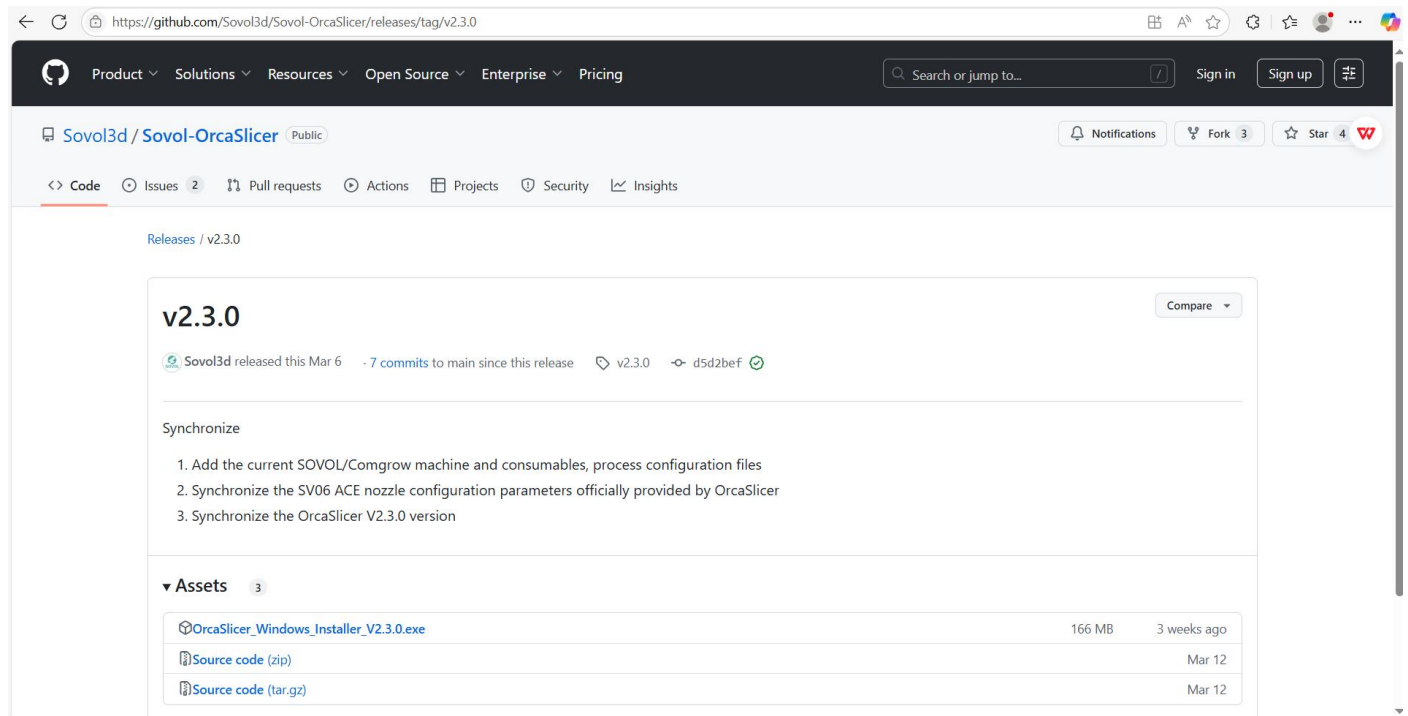# 0Orca slicer import configuration file(Windows)
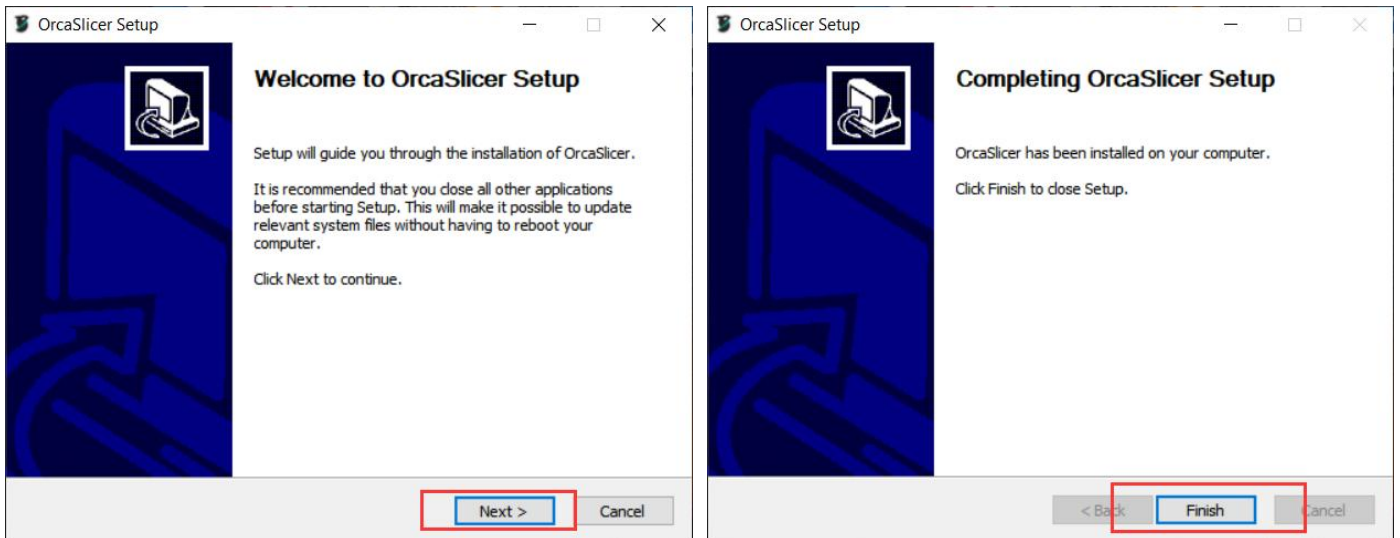
STEP 1   Download and Install

1.Find the download link of Orcaslicer in Github，Download the exe file that suits your current version on your PC.

Link：https://github.com/Sovol3d/Sovol-OrcaSlicer/releases/tag/v2.3.0

2.Install the Orcaslicer, software according to the prompts.

Note: This step is to export the configuration file. It is suitable for users who have used Orcaslicer and saved the new configuration process. If it is the first installation, please skip this step.

How to export the configuration file: Click "File" "Export" "Export Preset Bundle", select "Printer config bundle (.orca _printer)" and the model to be exported to complete the configuration file export.
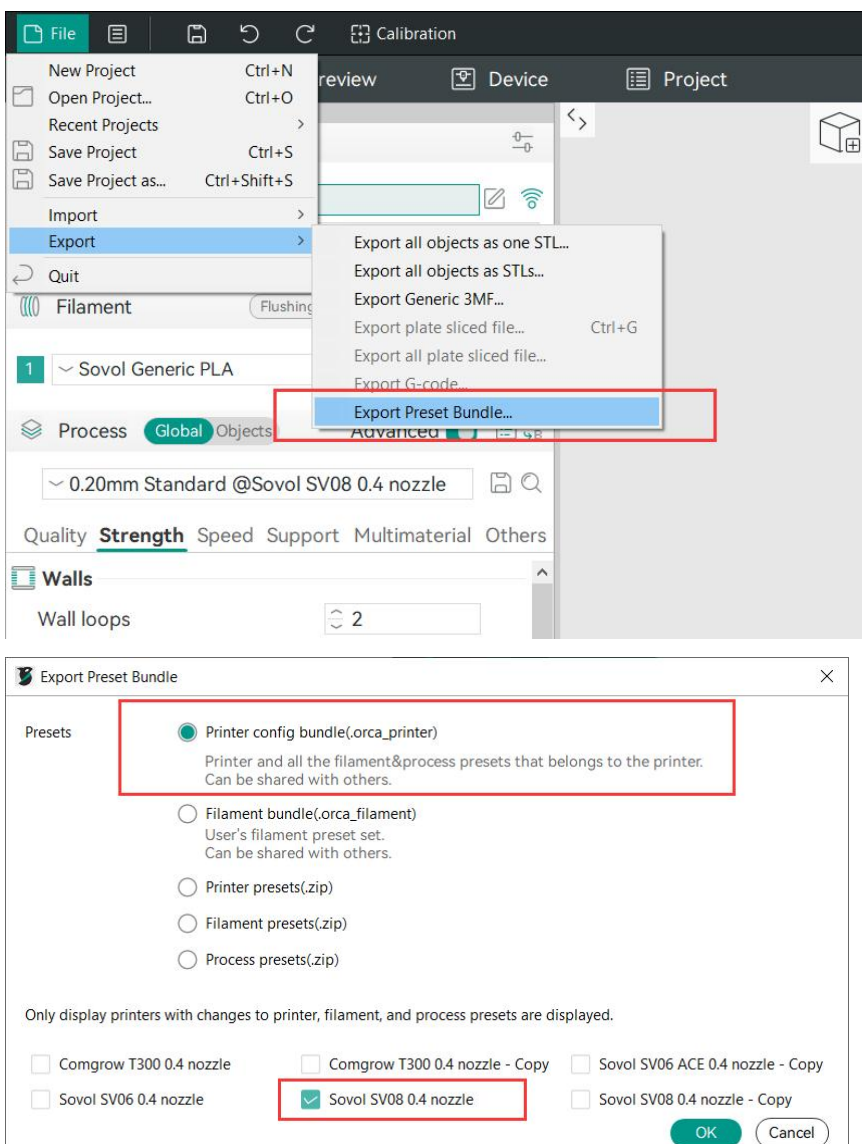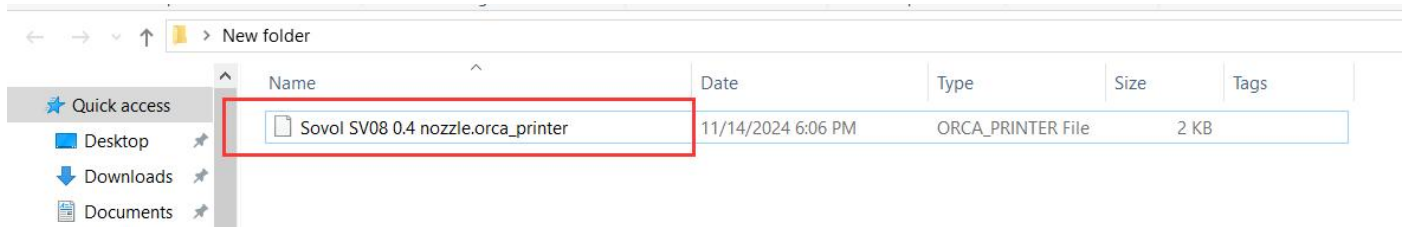
STEP 2　Import the configuration file

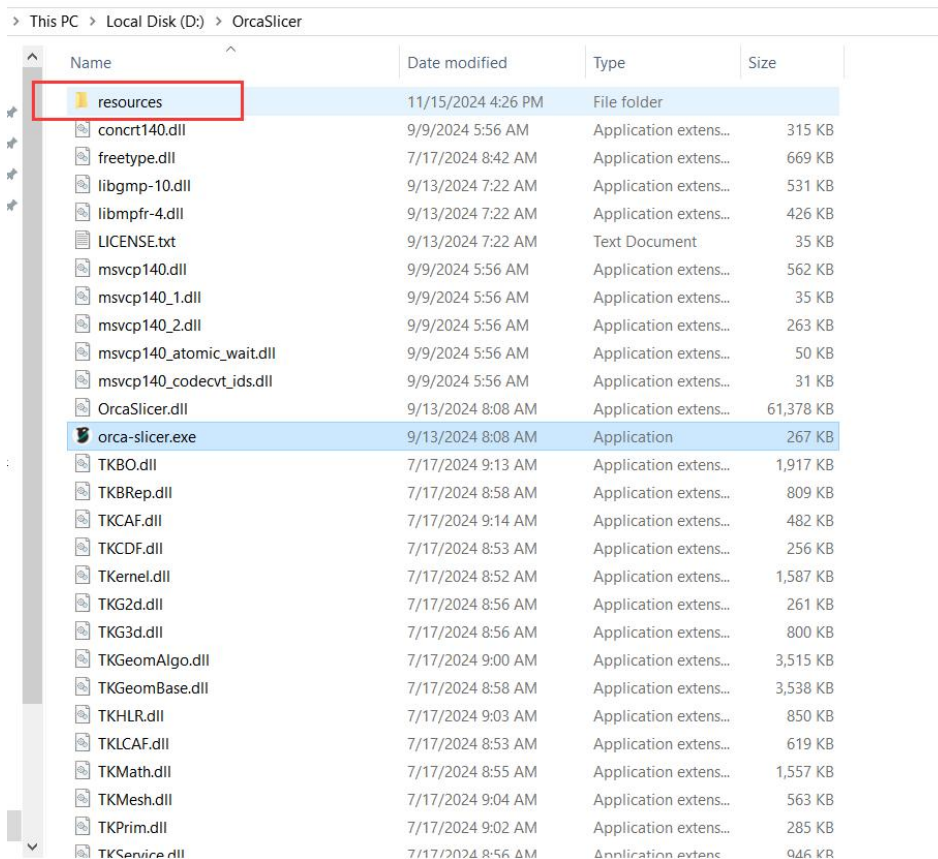(Orcaslicer's profiles do not include the latest model, so this step is required to replace and update)
(Using SV06 ACE as an example, this tutorial is applicable to all Sovol printer models)

3.Open the location ofthe file under the installation path of Orcaslicer

4.Open the "resources" directory under the "OrcaSlicer" directory

5.Open the "profiles" directory under the "resources" directory.
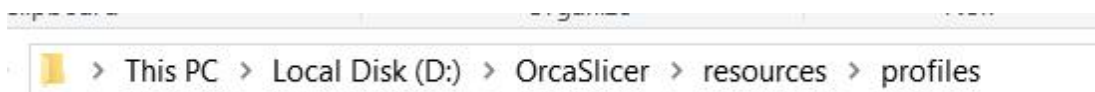
6.Copy the contents under the "Orcaslicer Windows"directory in the materials to the "profiles" directory.
(Tip: You can find the machine's configuration file in a USB stick, or download it on the SOVOL WIKI)
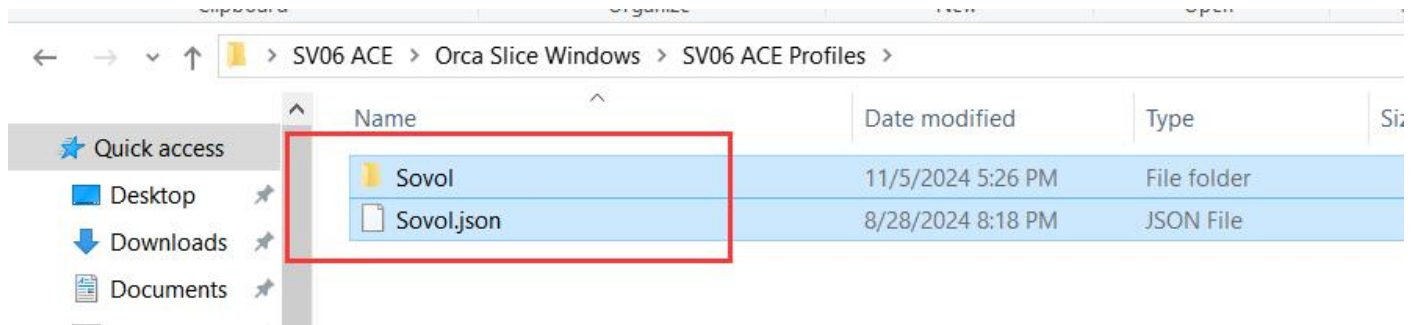
7.Click "Replace the files in the destination"
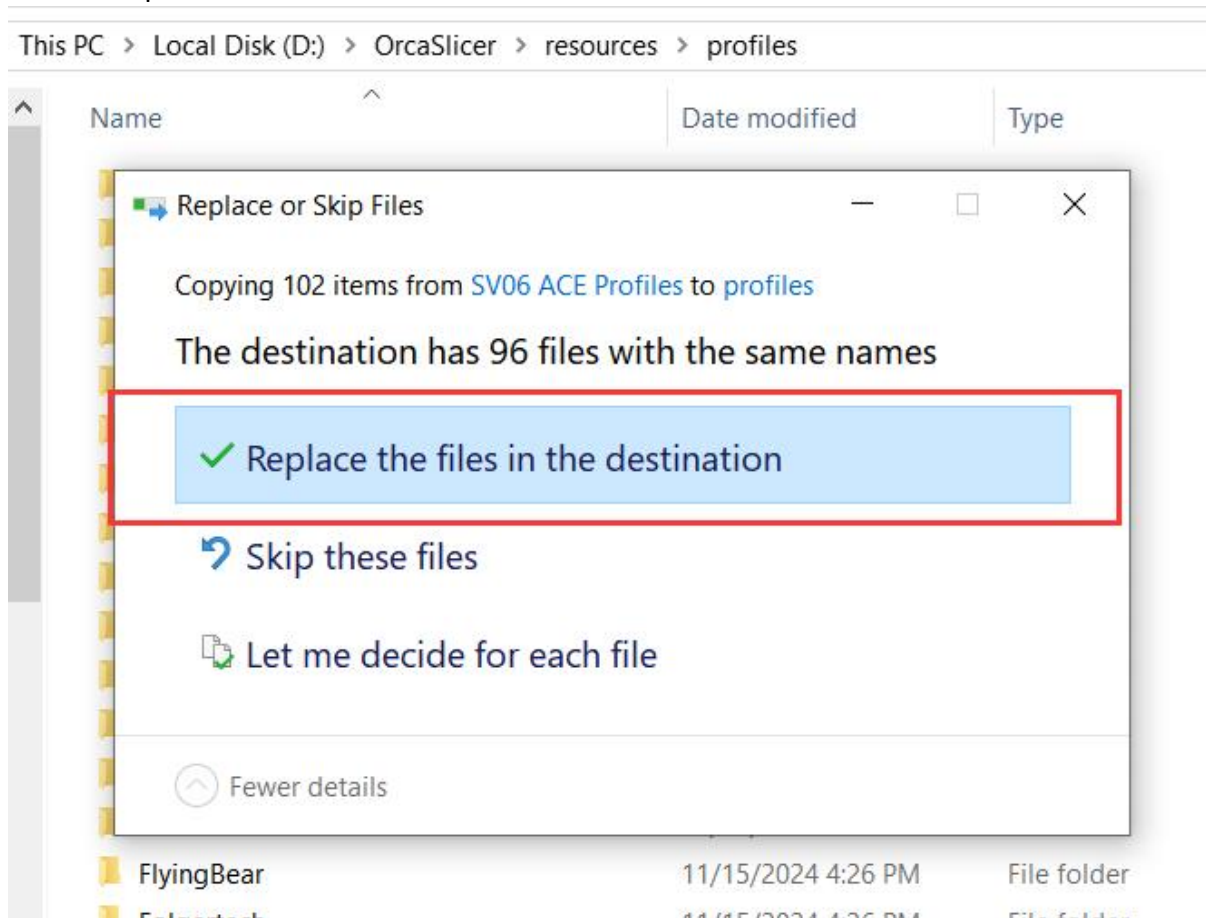
STEP 3 Delete the software's cache files

8.Open Orcaslicer, click "Help" "Show Confiquration Folder" and delete all files in the OrcaSlicer folder under this path. (Note: the previously added machine configuration will also be deleted. Please export the configuration before deleting the folder. Please refer to step 5 to export your configuration file.)
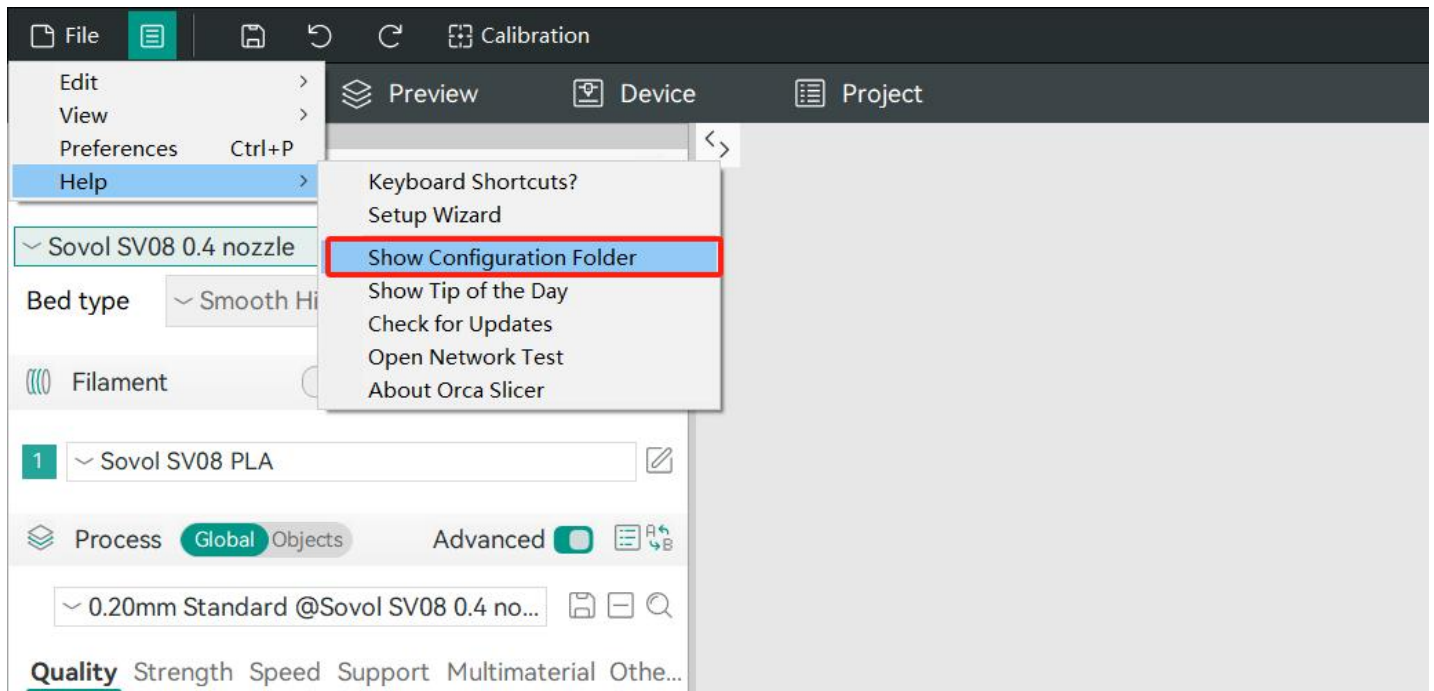
If you cannot open the configuration folder this way, please Open the location of the file under the installation path of Orcaslicer, find   \Roaming\OrcaSlicer , and delete all files in the OrcaSlicer folder under this path.
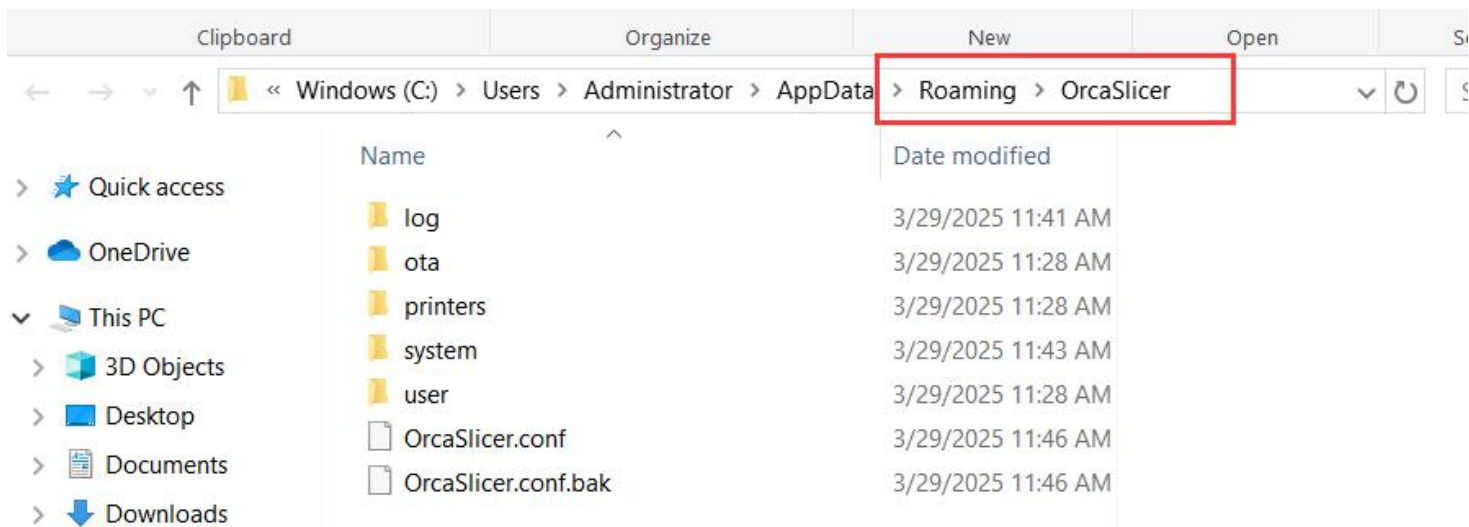
STEP 4 Select the model and filament

(Using SV06 ACE as an example, this tutorial is applicable to all Sovol printer models)

9.Reopen OrcaSlicer, search for "Sovol", and select the machine model want to add

File   Calibration   *Untitled

Prepare   Preview   Device   Project   Slice plate   Export G-code file

Printer

Sovol SV08 0.4 nozzle

Bed type   Smooth PEI Plate / High Temp Plate
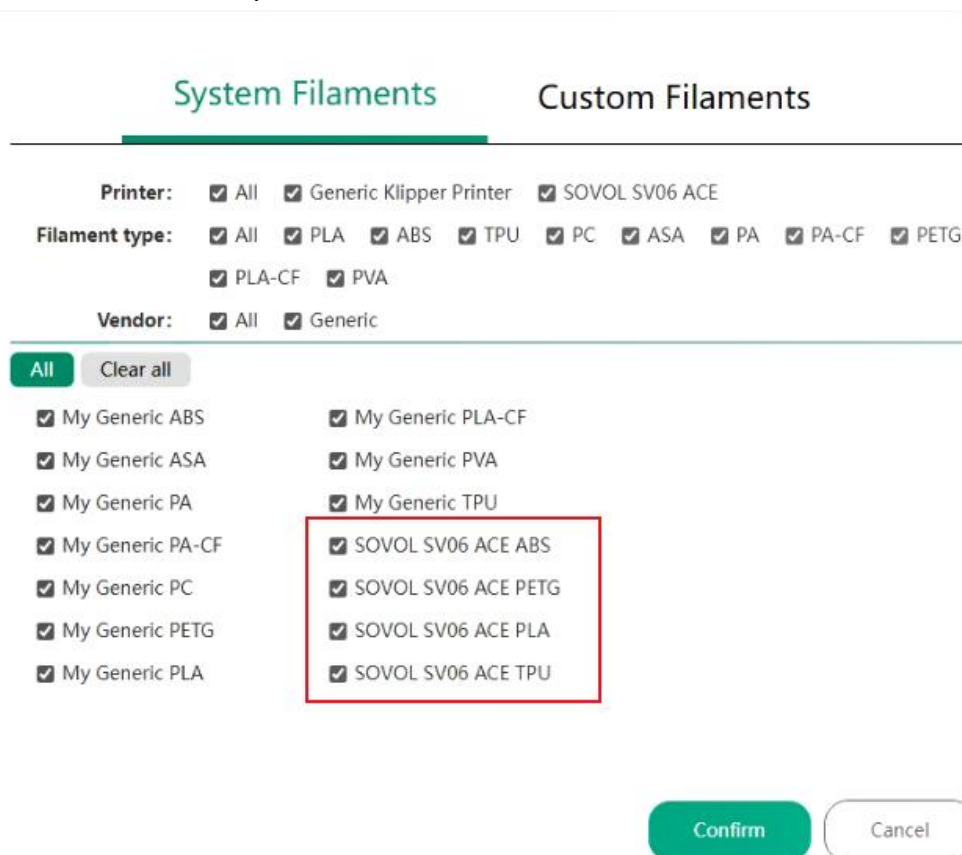
Filament   (Flushing volumes)

1   Sovol SV08 PLA

Process   Global   Objects   Advanced

0.20mm Standard @Sovol SV08 0.4 nozzle

Quality   Strength   Speed   Support   Multimaterial

Layer height
| Layer height | 0.2 | mm |
| First layer height | 0.25 | mm |

Line width
Default	0.4	mm or %
First layer	0.5	mm or %
Outer wall	0.4	mm or %
Inner wall	0.42	mm or %
Top surface	0.42	mm or %
Sparse infill	0.45	mm or %
Internal solid infill	0.45	mm or %
Support	0.45	mm or %

Seam
| Seam position | Aligned |

Printer Selection

SOVOL

Sovol   All   Clear all

Sovol SV01 Pro   Sovol SV02   Sovol SV05
☐ 0.4mm nozzle   ☐ 0.4mm nozzle   ☐ 0.4mm nozzle

Confirm   Cancel

10.Select the filament you need to add.

System Filaments   **Custom Filaments**

Printer: ☑ All ☑ Generic Klipper Printer ☑ SOVOL SV06 ACE

Filament type: ☑ All ☑ PLA ☑ ABS ☑ TPU ☑ PC ☑ ASA ☑ PA ☑ PA-CF ☑ PETG
☑ PLA-CF ☑ PVA

Vendor: ☑ All ☑ Generic

All   Clear all

☑ My Generic ABS   ☑ My Generic PLA-CF

☑ My Generic ASA   ☑ My Generic PVA

☑ My Generic PA   ☑ My Generic TPU

☑ My Generic PA-CF   ☑ SOVOL SV06 ACE ABS

☑ My Generic PC   ☑ SOVOL SV06 ACE PETG

☑ My Generic PETG   ☑ SOVOL SV06 ACE PLA

☑ My Generic PLA   ☑ SOVOL SV06 ACE TPU
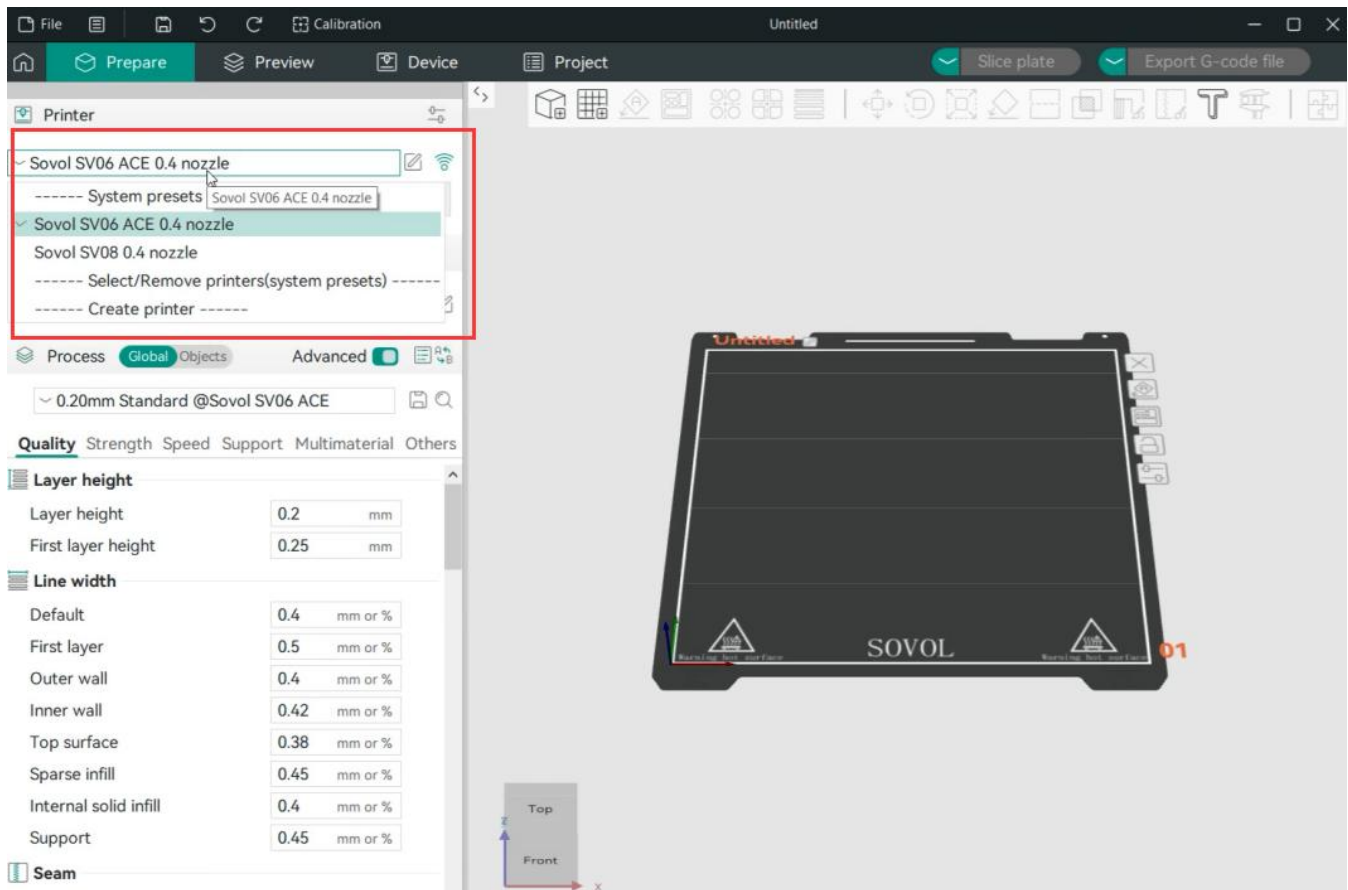
Confirm   Cancel

11.If the filament or model is not found, please return to the previous step and select Generic klipper Printer, then select the corresponding Sovol/Comgrow printer model, and click Next.
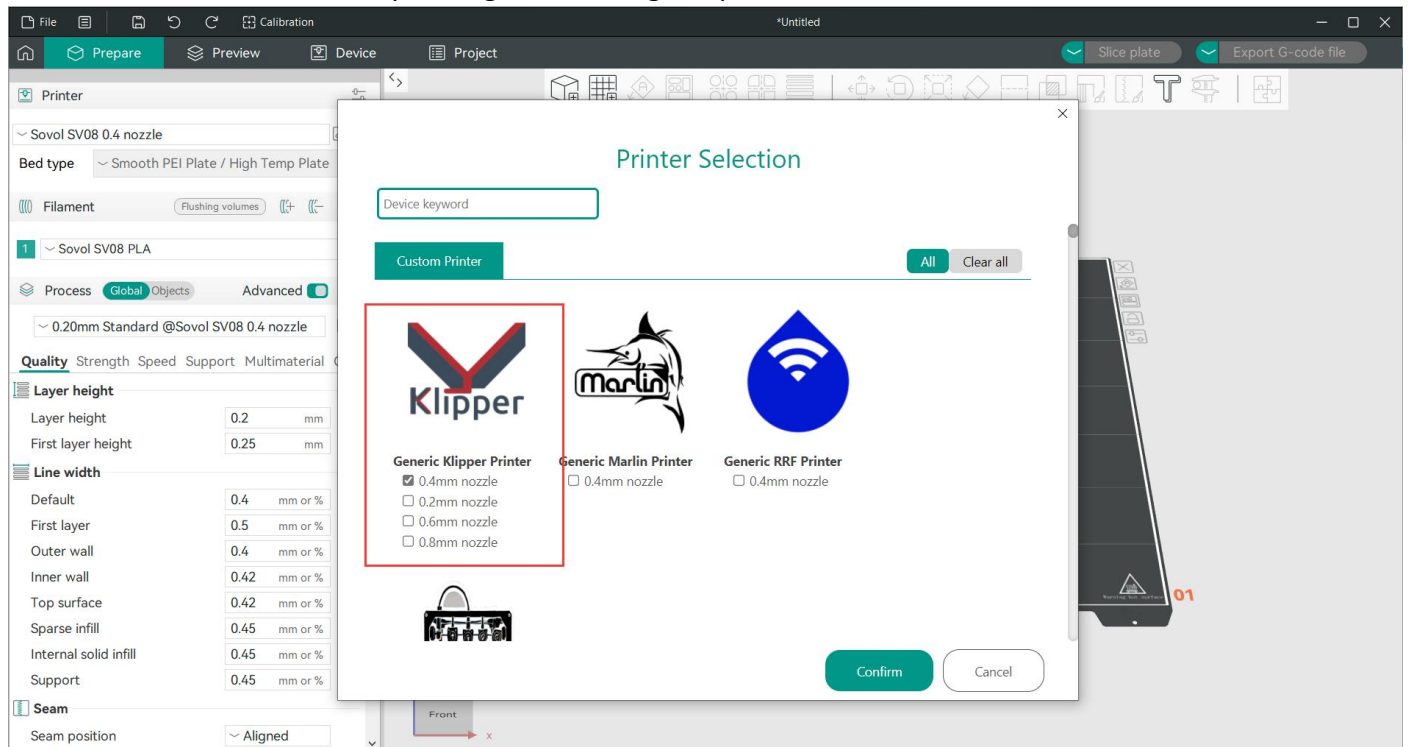
12.Add Generic filament and complete the setup
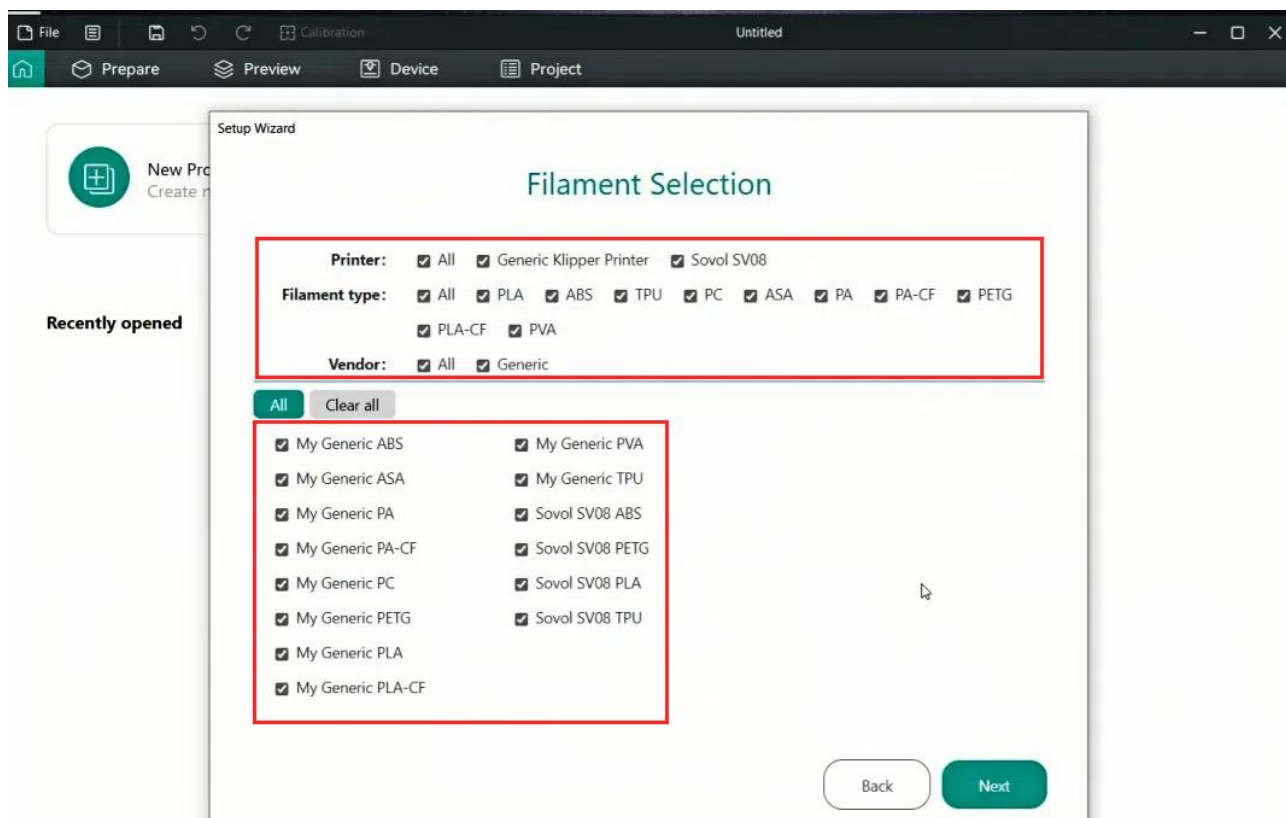
13.Click Prepare to enter the main interface of the slicing software and switch to the printer model you selected

14.Complete the Orcaslicer installation

15.If Orcaslicer displays "The configured post-processing script does not exist: C:\Users\Administrato r\AppData\Roaming\OrcaSlicer-sovol\Plugins\SpixiDecelerationTool.exe" when exporting Gcode after sli cing, you need to use a plug-in to allow the printer to decelerate according to the increase in h eight and weight when printing higher models or heavier models.

Plugin download address: https://github.com/Sovol3d/Sovol-OrcaSlicer/tree/main/Document
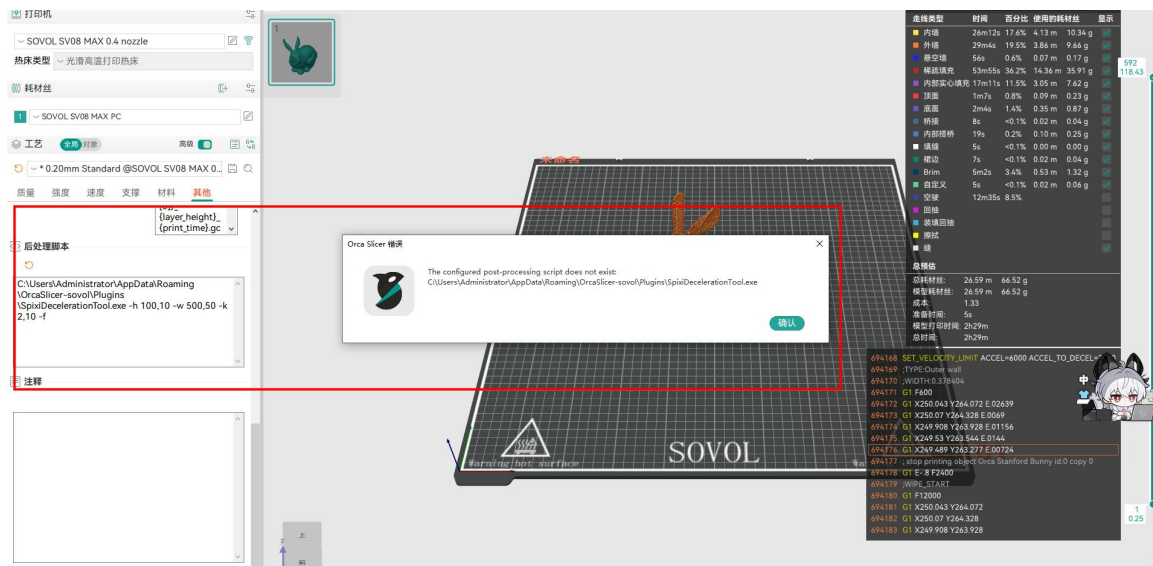
Step 1: Click "Help", "Show Configuration Folder", create a new "Plugins" folder, and copy the plug-in to the folder.
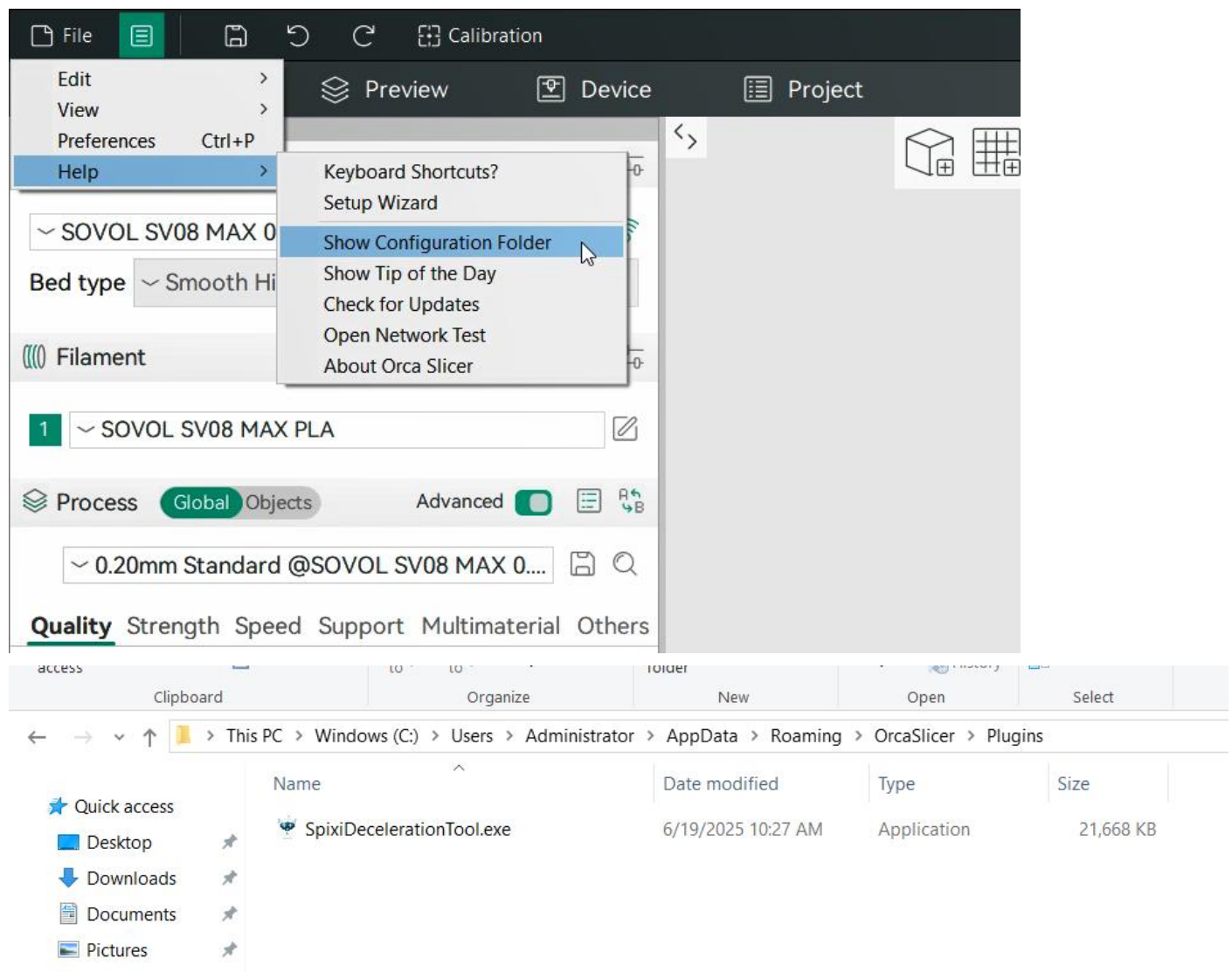
Step 2: Then copy the absolute path of the plug-in into the location of the post-processing script, and add the "-h 100,10 -w 500,50 -k 2,10 -f" parameters at the end to use the plug-in normally.
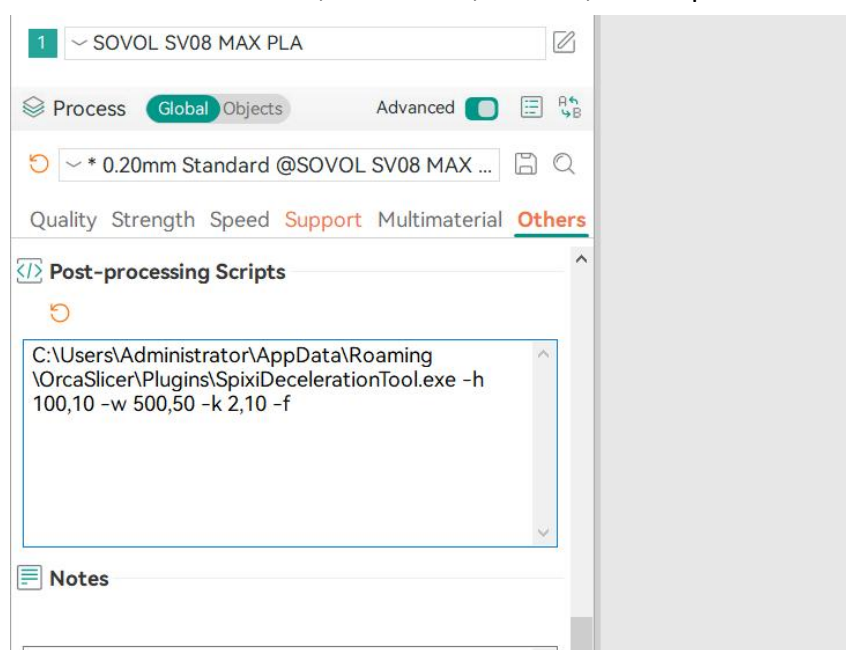
Notes：

-h: height	100: height to start deceleration	10: deceleration interval
-w: weight	500: weight at which the speed reduction starts	50: one gear is reduced every 50g
-k: speed reduction ratio	2: ratio of each speed reduction	10: minimum speed reduction value
-f is a file(no parameters, but must be added)		

Tip：Orcaslicer is an open source slicing software for 3D printing. It is deeply optimized based on PrusaSlicer, SuperSlicer and Bambulab studio. It is very suitable for new users and makers. You can download Orcaslicer on Github or install it using the installer in the USB memory stick. If you encounter any problems during use, please contact SOVOL

16.If you export the orca_printer machine configuration and then import the Printer config bundle (.orca_printer) again:
Must add the same machine to import the orca_printer config bundle ; otherwise, the import will fail.