ideaMaker 5.0.5 Beta Release Notes

Introduction

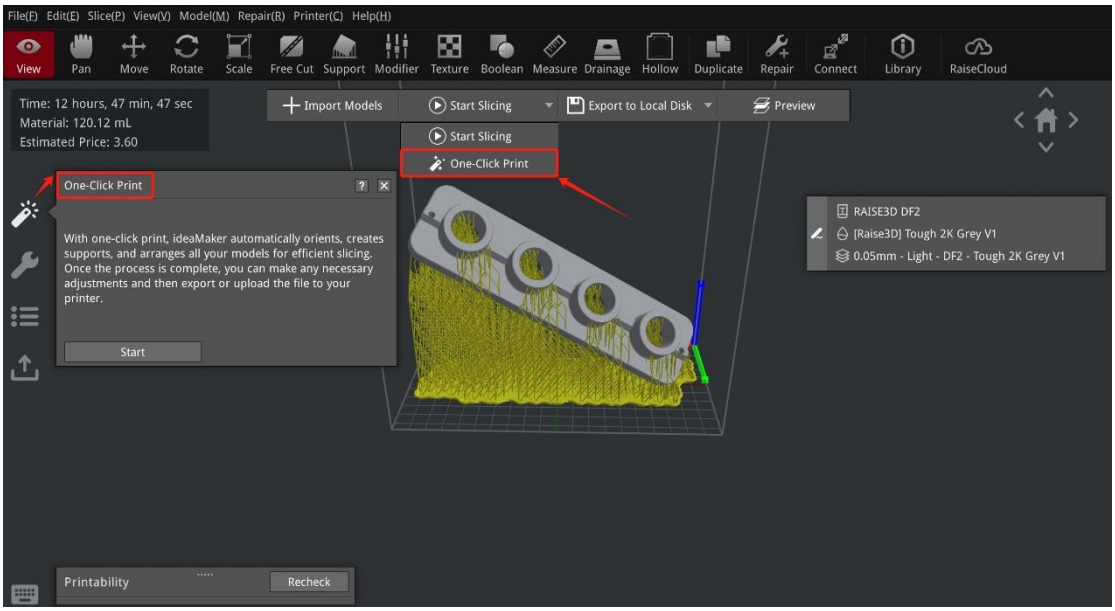
This new version primarily adds support for resin 3D printers, including several key features for resin printing and some new features for filament printing.

- This version of ideaMaker, the 5.0 version, is released in Beta this time, covering the features of ideaMaker 4.3 and ideaMaker 4.4.
- Supports upgrading from ideaMaker 4.3/4.4 versions to ideaMaker 5.0. During the upgrade, users can choose which of their user data they wish to migrate to the new version.
- Users are allowed to have ideaMaker 4.3/4.4 and ideaMaker 5.0 installed simultaneously.
- During the installation process, the configuration of ideaMaker 4.3/4.4 will be fully copied into the configuration directory of version 5.0, without affecting the use of ideaMaker 4.3/4.4, thus enhancing user experience.

DLP New Features

1. One-Click Print

With One-Click Print, ideaMaker automatically creates supports and orients and arranges the models for efficient slicing. Once the process is complete, you can make any necessary adjustments and then export or upload the file to your printer.

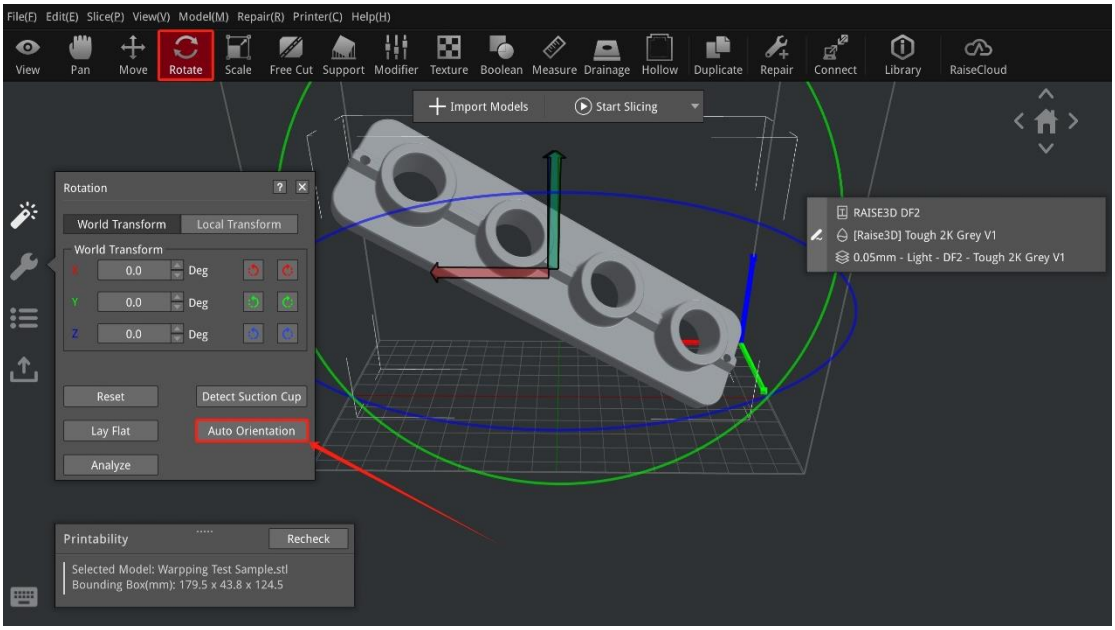


2. New Features for Resin Printing

2.1 Model Preparation

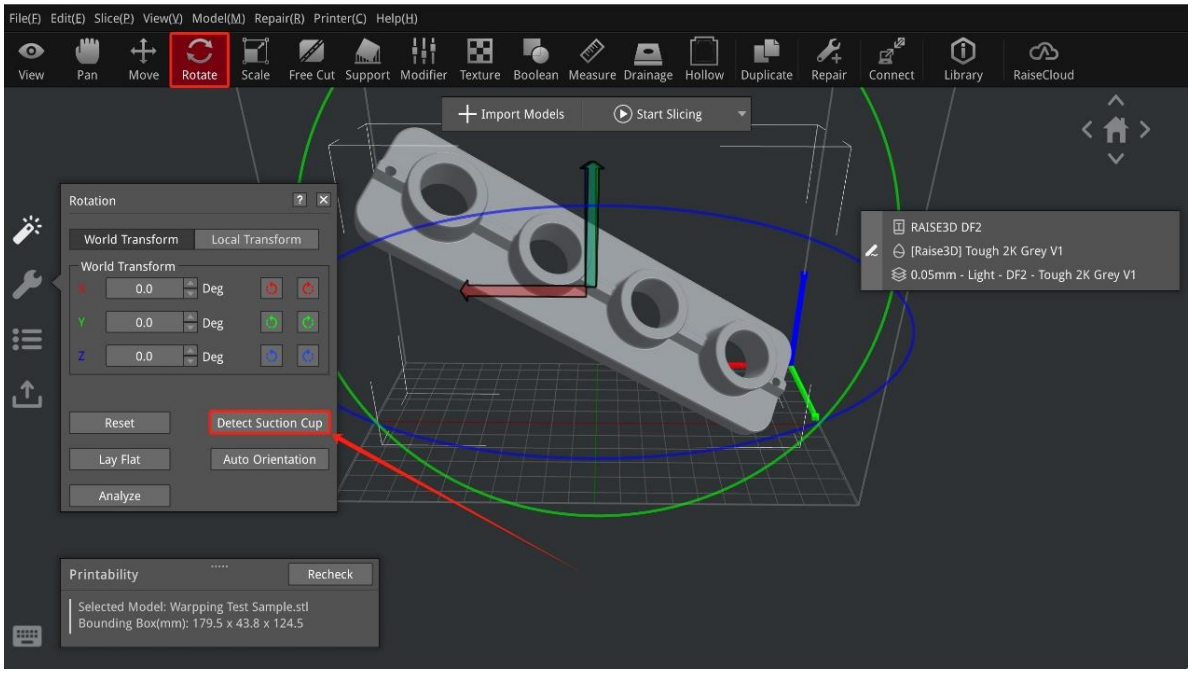
2.1.1 Auto Orientation

Click "Auto Orientation" to automatically select an angle for the selected model to improve the printing stability, and try to avoid some details on the model facing the printing platform that may affect the surface smoothness of the model.



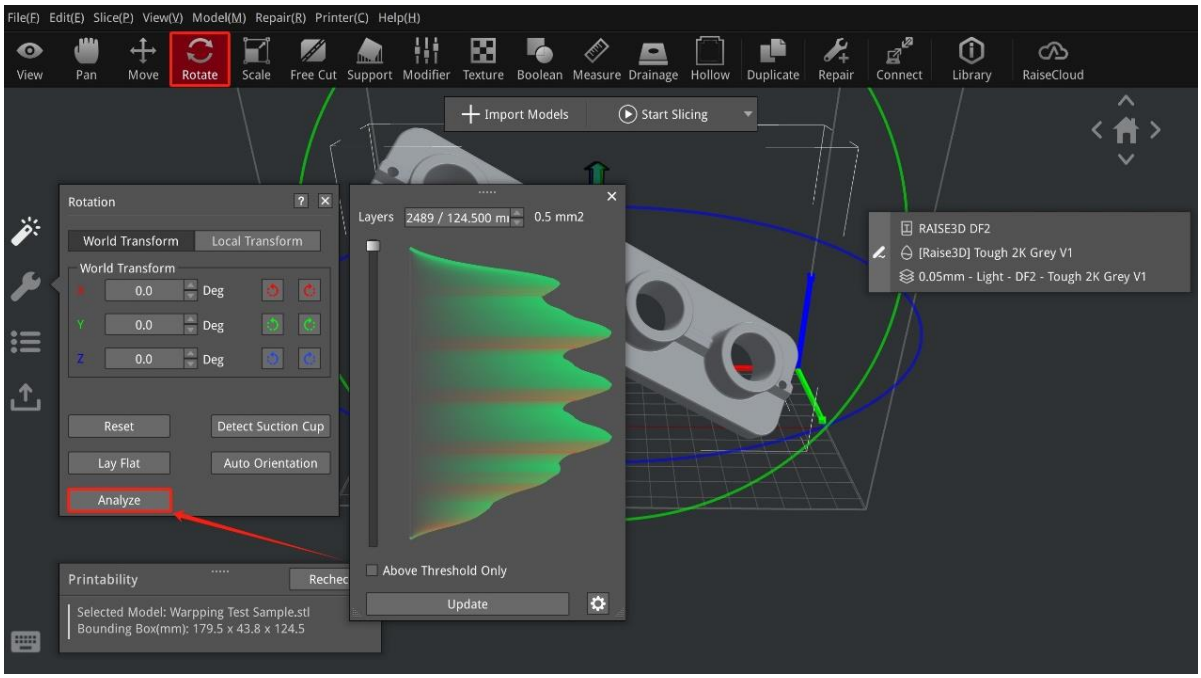
2.1.2 Detect Suction Cup

Click "Detect Suction Cup" to check suction cups for the selected model.



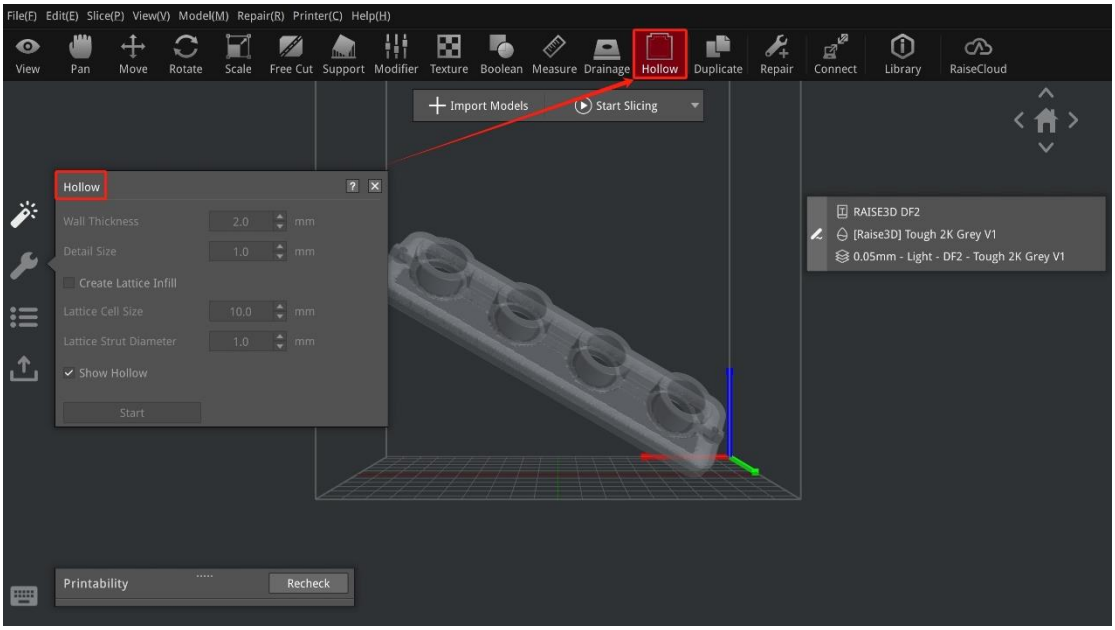
2.1.3 Cross-sectional Area Analyze

Click "Analyze" to check for any sharp changes of model cross-sectional area (the red area represents the rate of change of cross-sectional area on the model that exceeds the threshold value).



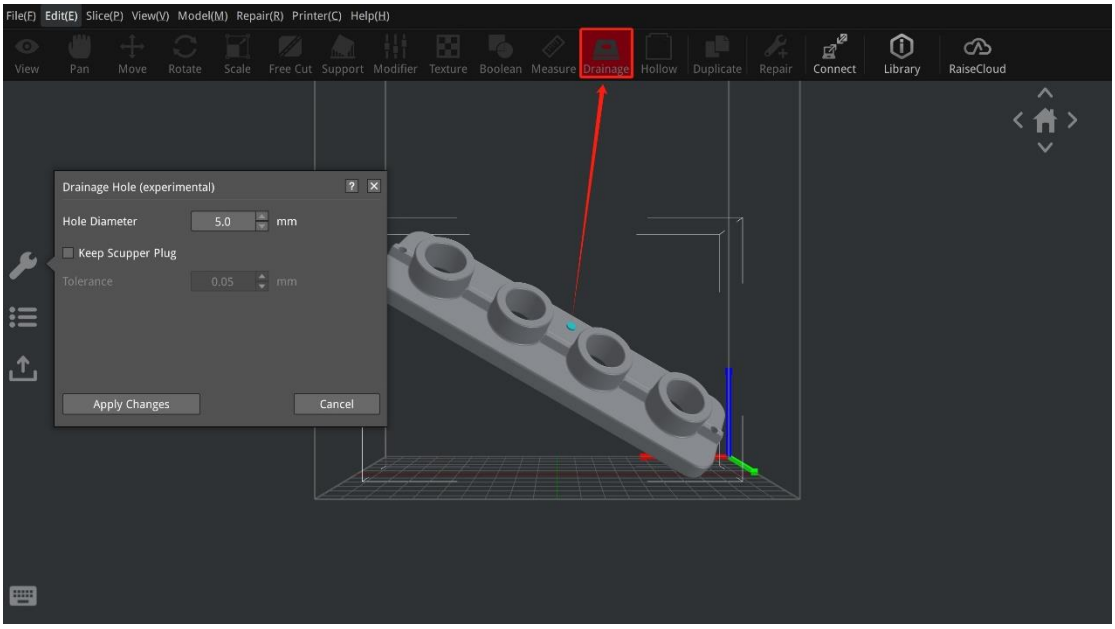
2.1.4 Hollow

Hollow out the selected model.



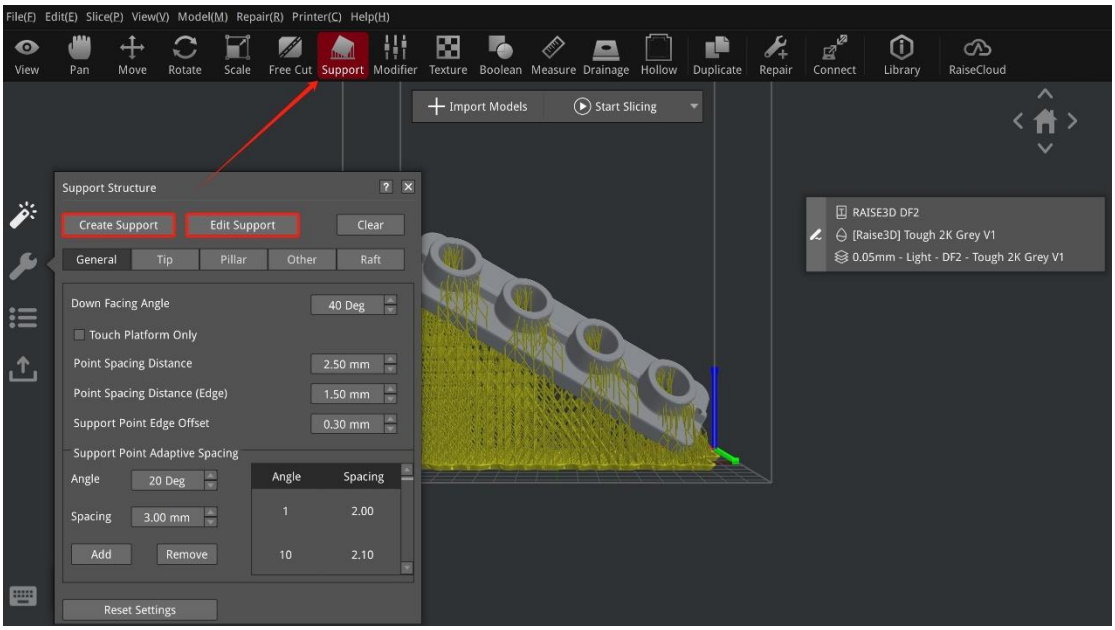
2.1.5 Drainage

Add drainage holes to the models.



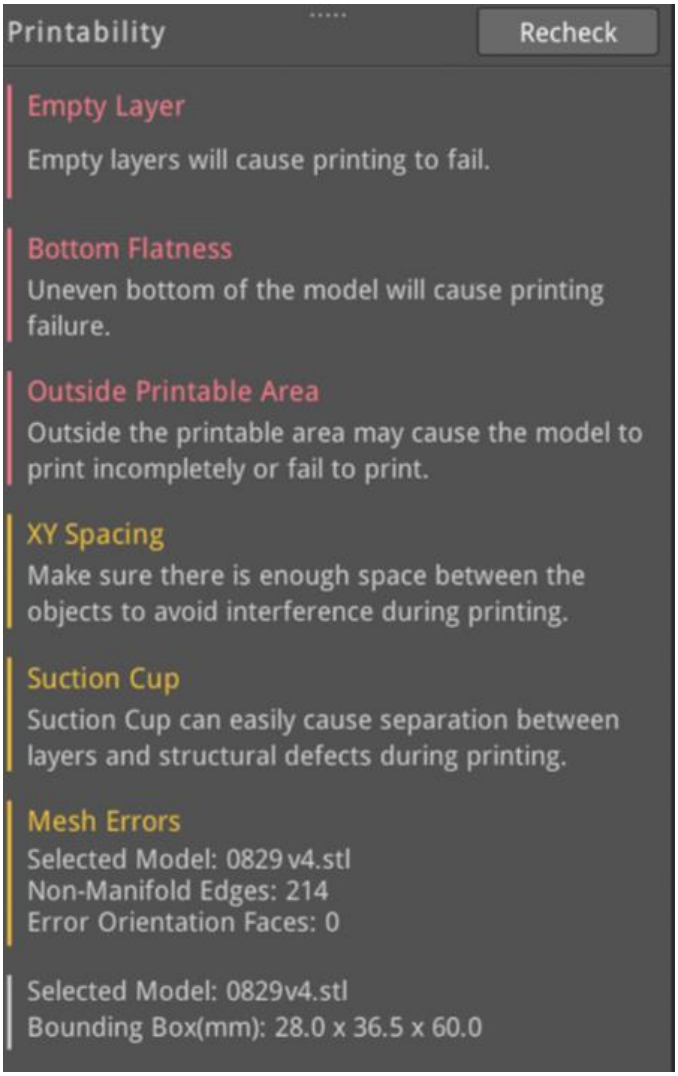
2.2 Add Supports

Supports can be auto-generated or added manually.



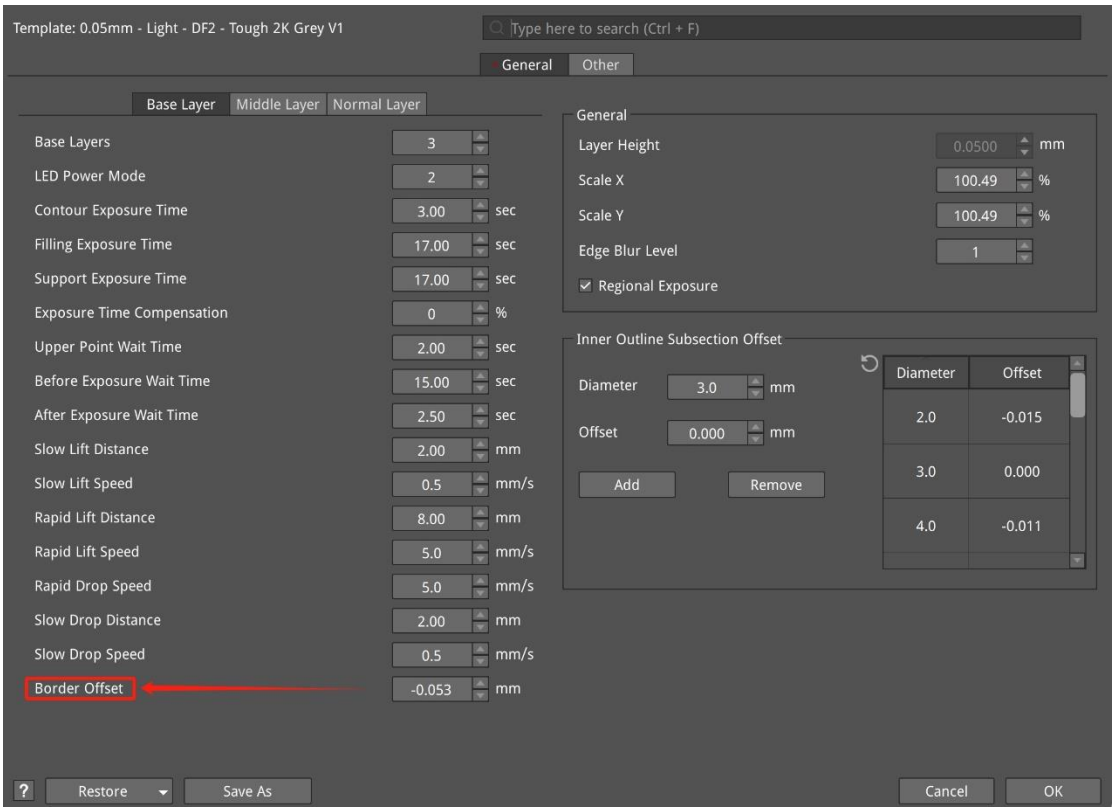
2.3 Printability Check

- Outside Printable Area: Check to see if the model is outside the printing boundaries
- Mesh Errors: Non-Manifold Edges, Face Orientation Error
- Light Curing Errors (these need to be detected before appearing): Suction Cup, XY Spacing, Empty Layer, Bottom Flatness



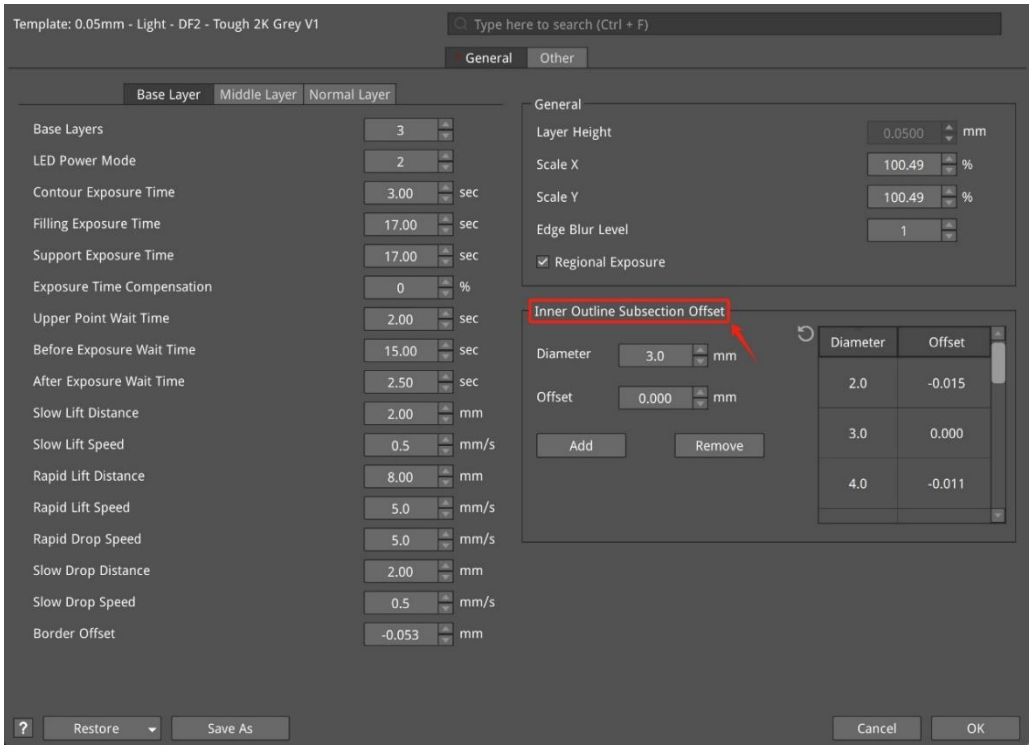
2.4 Border Offset

Optimized resin shrinkage control for high design-to-part consistency.



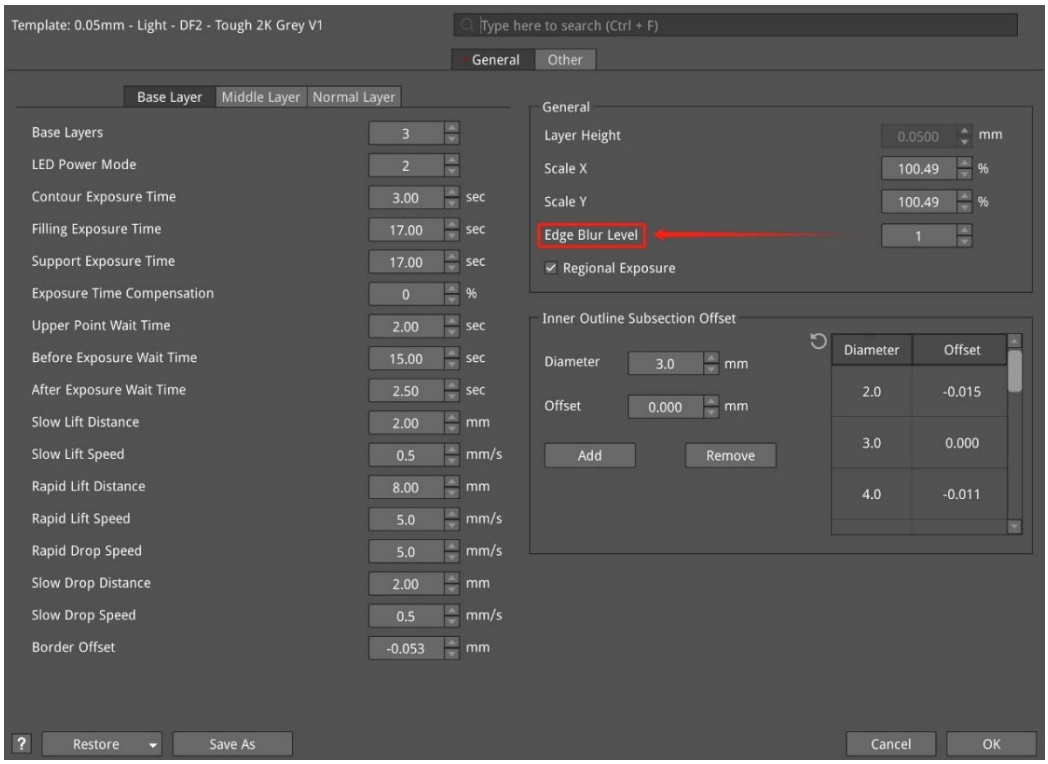
2.5 Inner Outline Subsection Offset

It is used to modify the distance between the inner outline and the outer outline of a model, in order to better control the structure and surface quality of the model during printing.



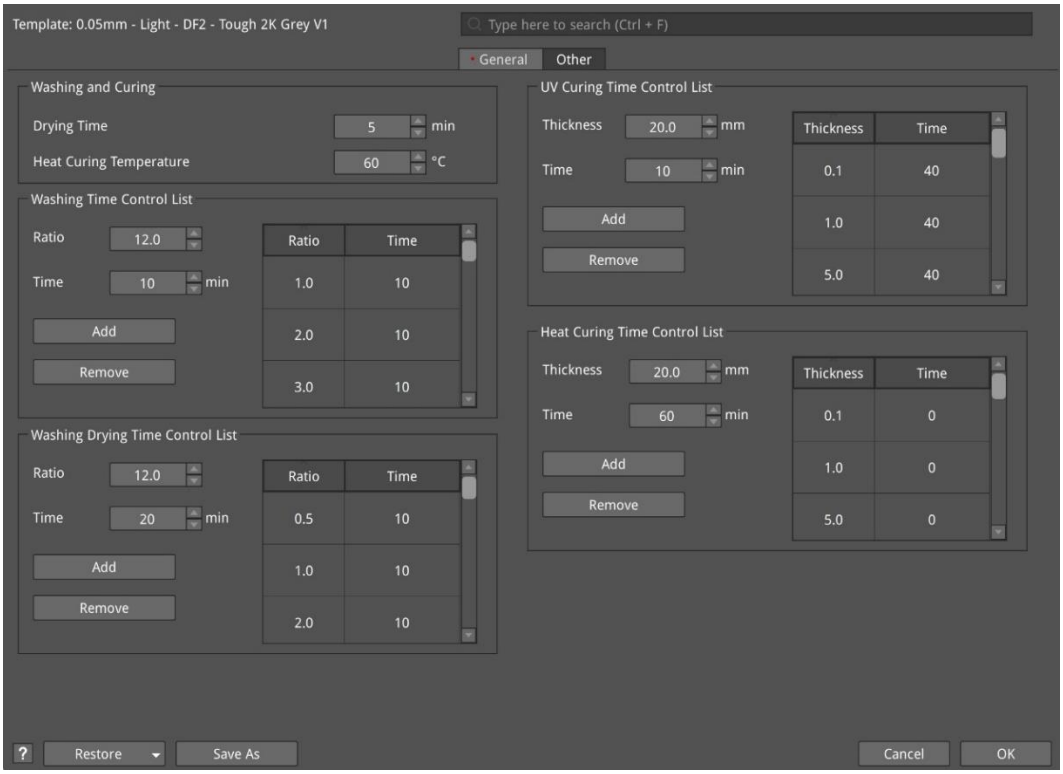
2.6 Edge Blur Level

It is used to adjust the blur level and smoothness of the printed model edges to balance print quality.



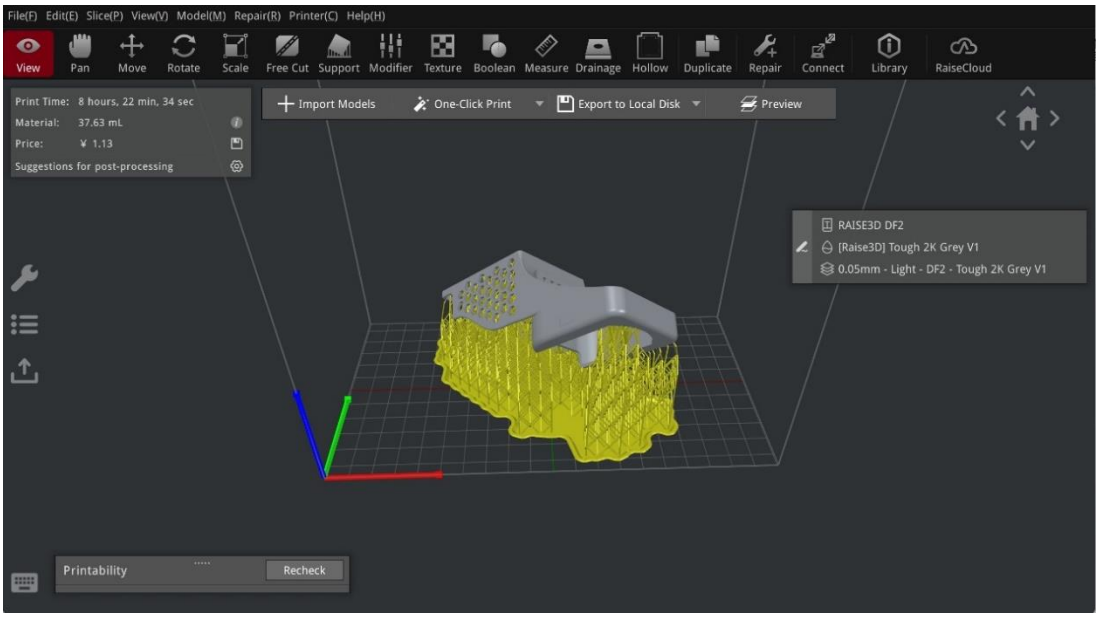
2.7 Post-processing Parameter Settings

Automatically calculates parameters during slicing and inputs the results into the slicing file, for use with Raise3D washing and curing machines.

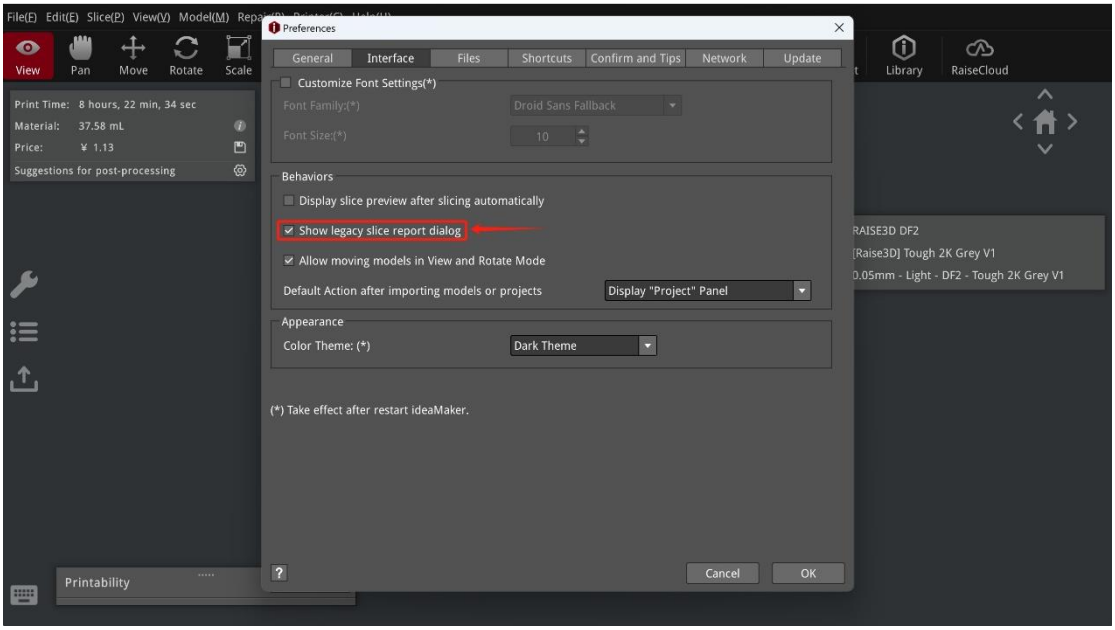


3. Wash&Cure Workflow 2.0

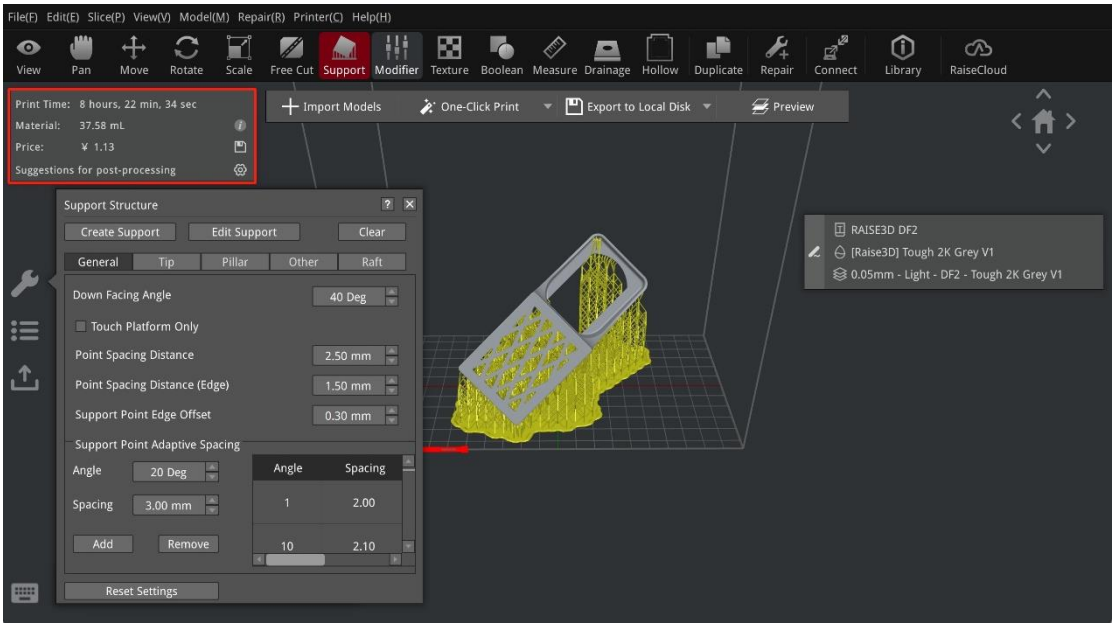
- Removed the one-click print interface icon on the left side.
- After slicing is completed, the main interface layout is adjusted as follows:
  - Modified the slice result preview in the upper-left corner to the new panel style: Slice Result Panel.



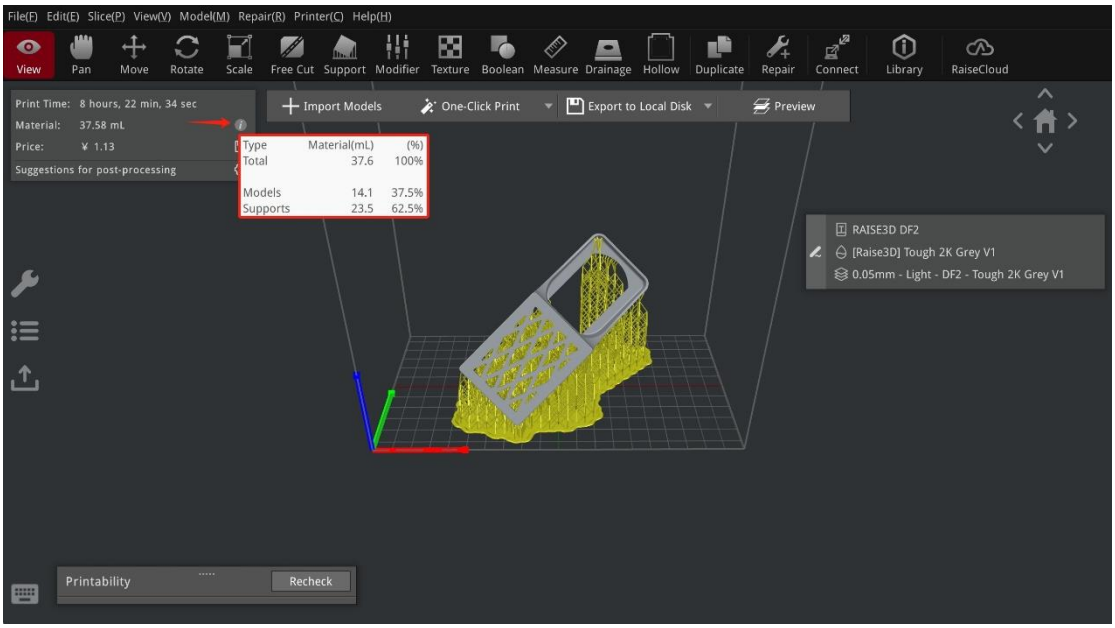
- The original slice report dialog is not displayed by default. You can control whether the dialog is displayed in Preferences -> Interface -> Show Legacy Slice Report Dialog, with the default setting being not displayed.



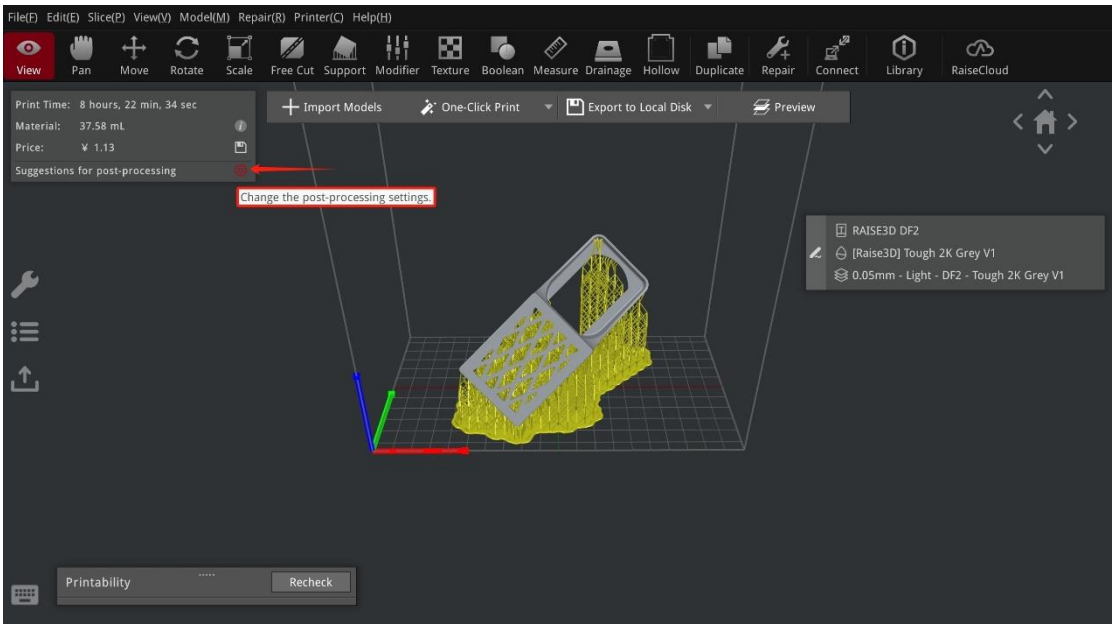
- Slice Result Panel
  - The Slice Result Panel only displays three lines of information: Print Time, Material, and Price, as well as suggestions for post-processing.



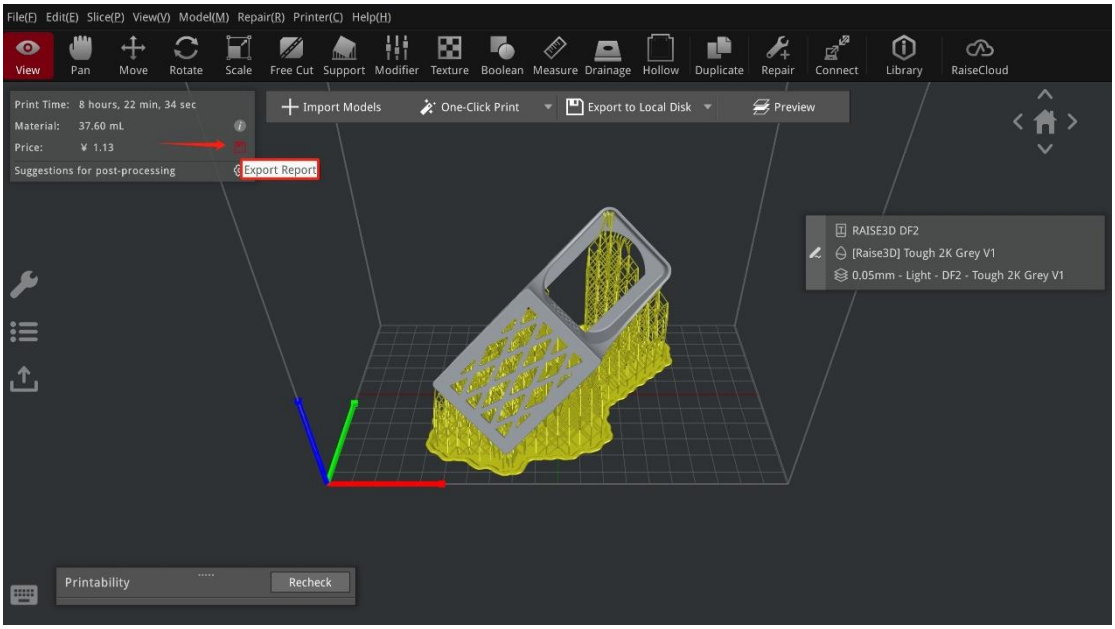
- DLP display model and support for respective material usage (preview of external dlpcode not displayed in interface)



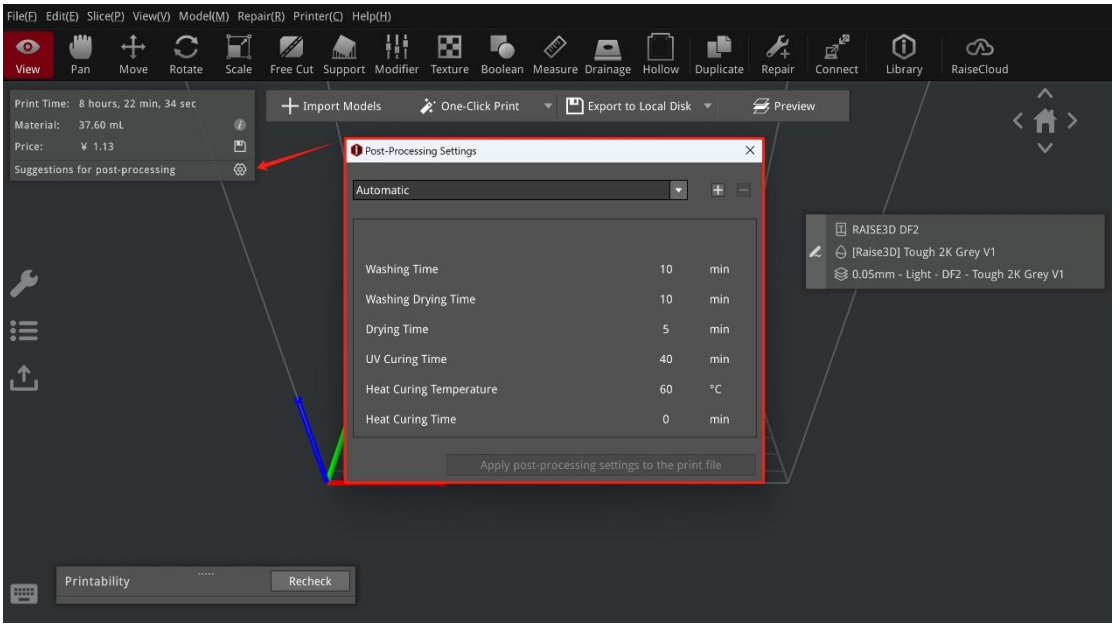
- DLP display washing post-curing settings



- Export slicing results report button provided on the right side of the price information (preview of external gcode, dlpcode not displayed in interface)



- Slice Result Panel -> Washing and Curing Settings Dialog
  - By default, after slicing, the automatically generated washing and curing parameter values are displayed (Automatic). This is for display purposes only and cannot be edited or modified by the user.

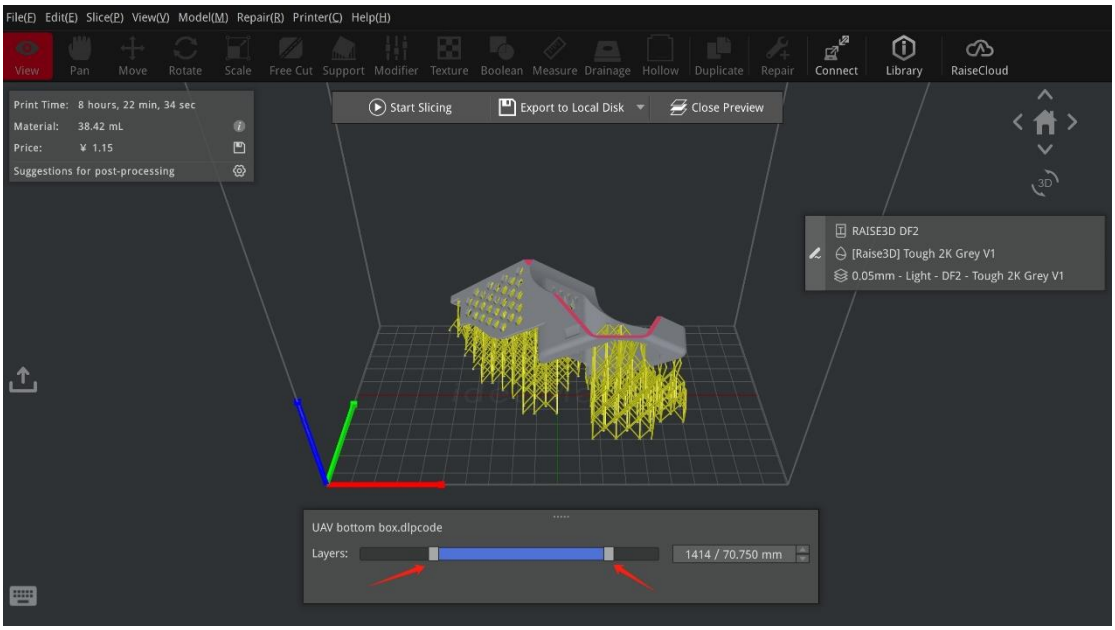


- Users can add or delete custom washing and curing profiles.

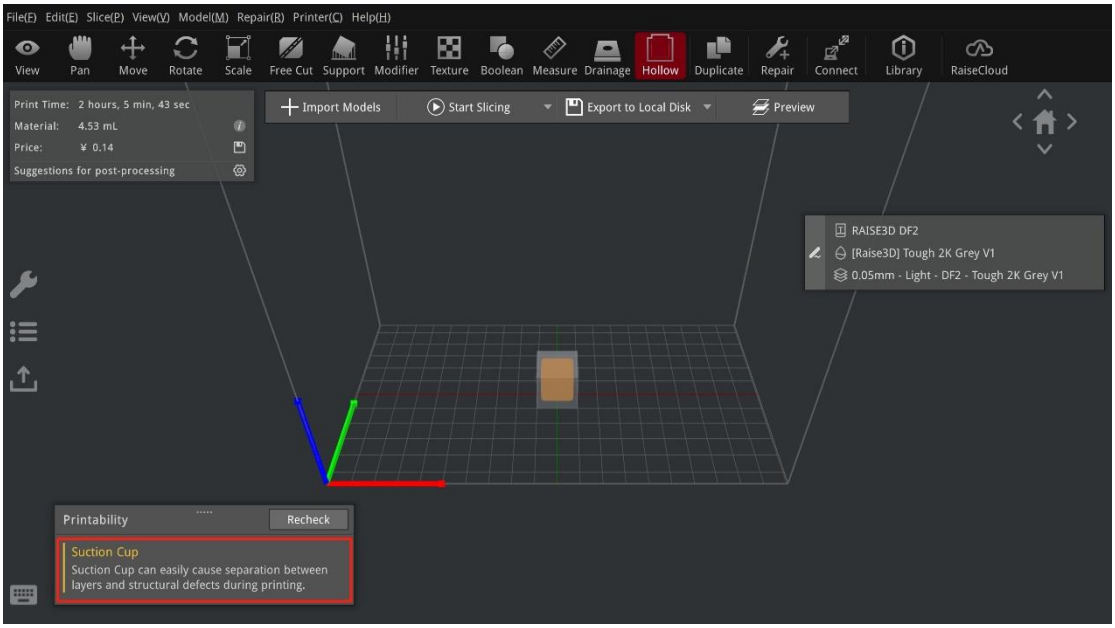
- Slice Result Panel -> Export Slicing Result Report

4. Support editing interface and optimize GCode preview scrollbar.

- DLP editing support, allowing simultaneous bottom and top layer preview control.



5. Manual slicing now includes Suction Cup, displayed in the printability check panel.



6. Automatic scaling of DLP models adjusted to the maximum dimension rules:

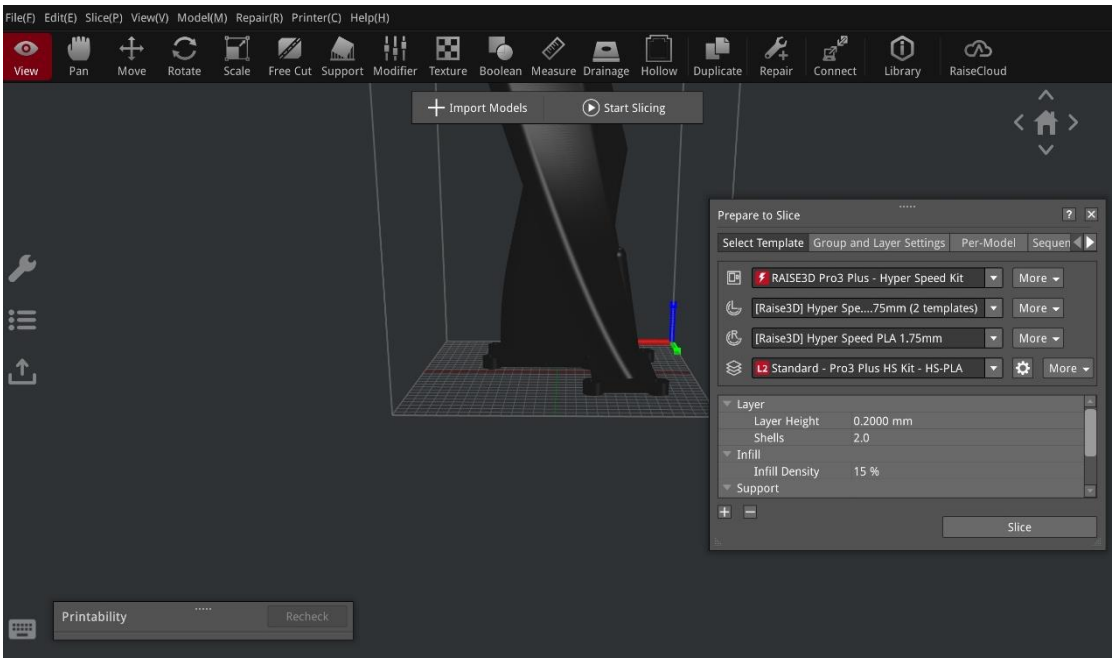
- For DF2 printers, the “Auto Fit to Build Volume” function automatically adjusts to the following dimensions as the maximum:
  - 200.018 mm
  - 112.098 mm

7. User version opens up DLP slicing template parameter adjustments:

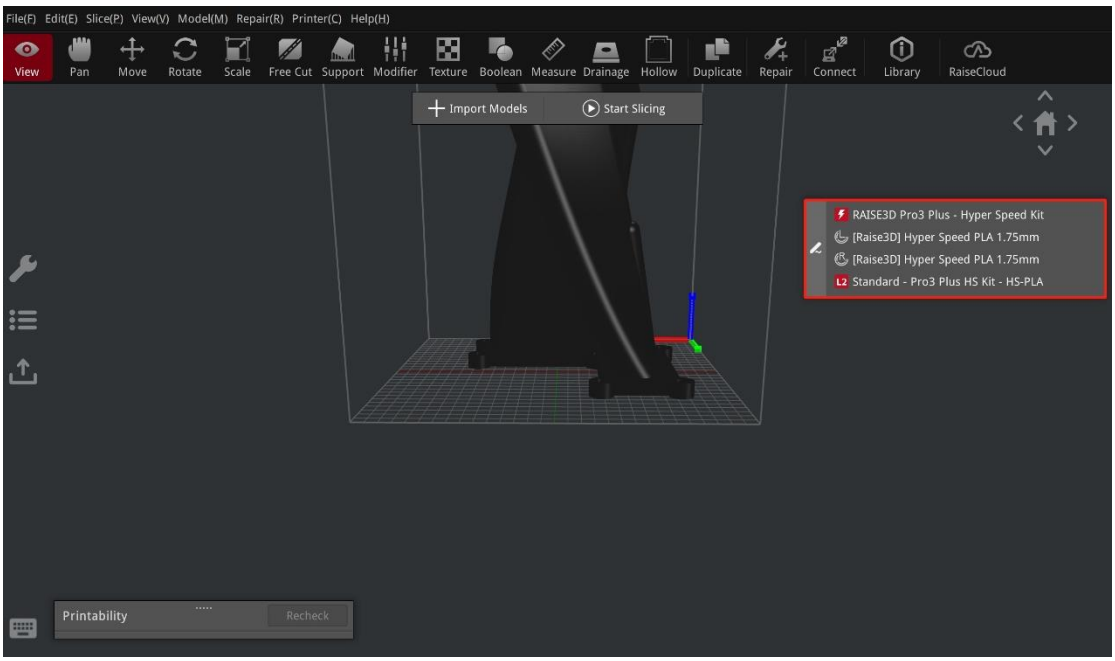
- In the ideaMaker User version, in comparison to version 5.0.4, the following adjustments have been made to the DLP slicing template settings dialog:
  - Temperature Tab opened
  - GCode Tab opened
  - Layer Tab:
    - Exposure Time Compensation are not opened
    - Regional Exposure are not opened

New Features and Improvements for FFF

1. Optimization of the interaction between the main interface and the "Prepare to Slice" panel.

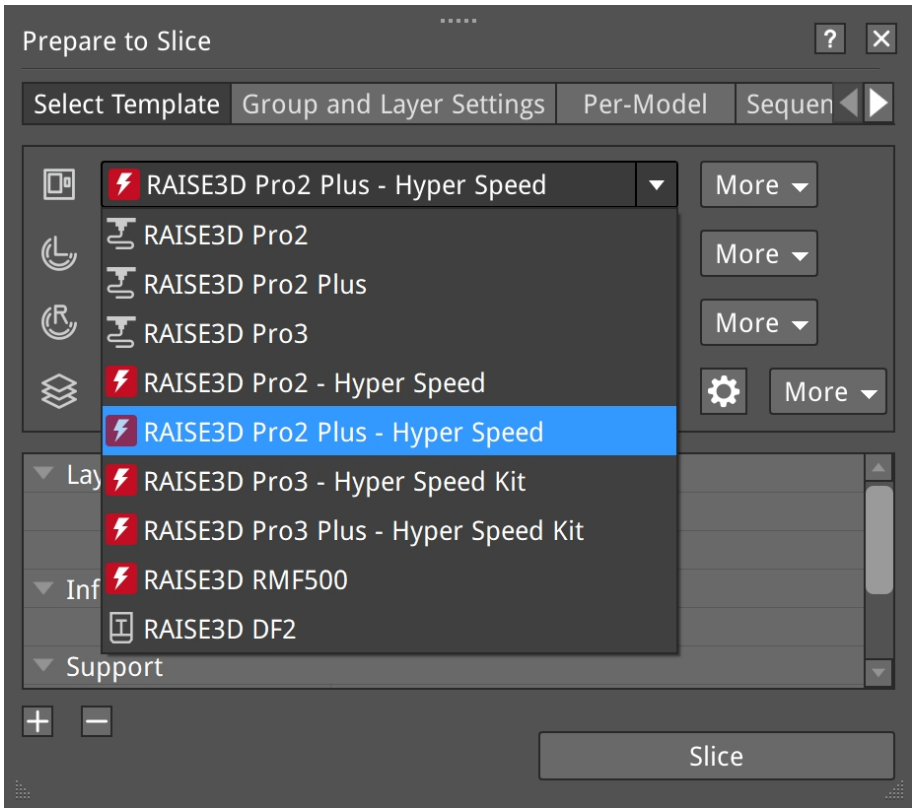


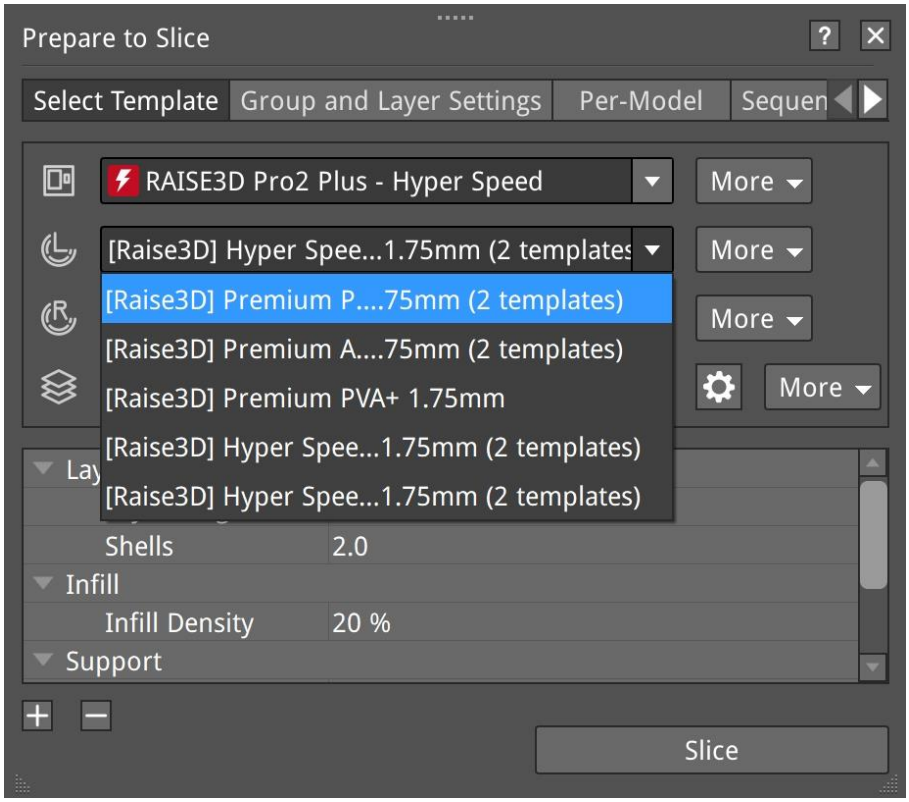
- The "Prepare to Slice" panel is located on the right side of the main interface. After clicking the close button in the top right corner of the panel, simplified panel information will be displayed.



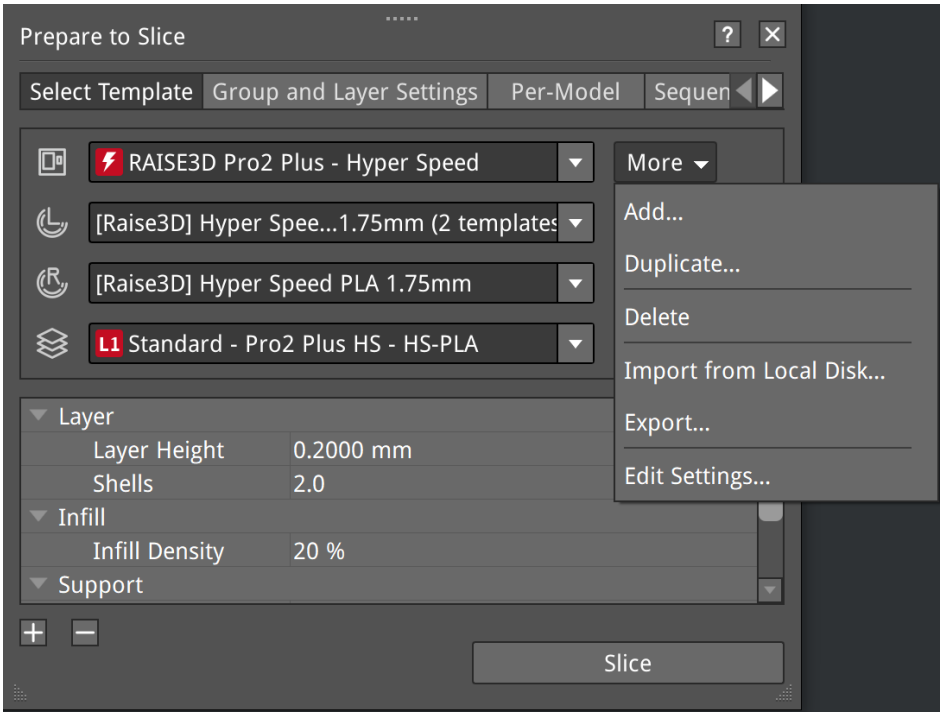
- Prepare to Slice
  - Printer list and material list only show the printers and materials that the user needs to show.
  - Users can freely add or remove official printers and custom printers.

- Users can add or delete official materials and custom materials, facilitating the management of commonly used printers and materials.

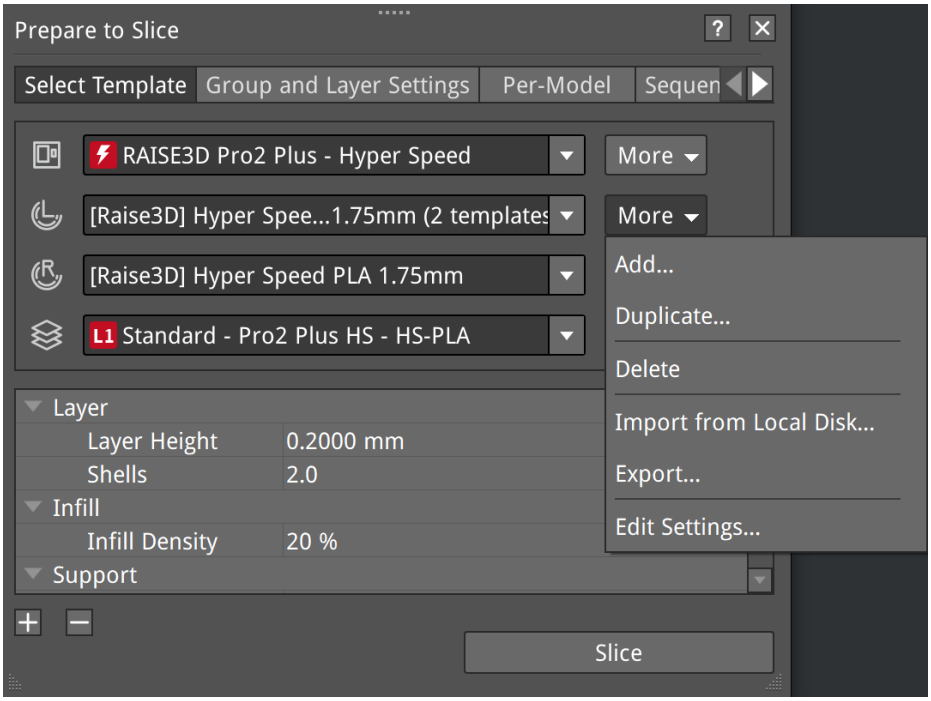




- New Printer Management Feature: Added a "More" menu page, which includes printer-related functions such as Add, Duplicate, Delete, Import, Export, and Edit Settings.

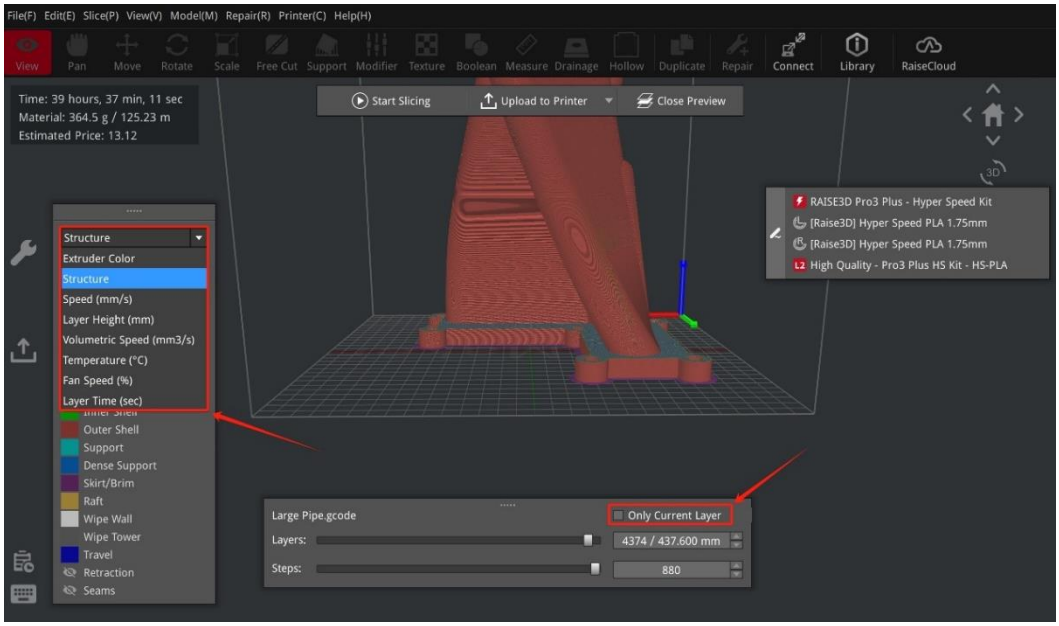


- New Material Management Feature: Added a "More" menu page, which includes material-related functions such as Add, Duplicate, Delete, Import, Export, and Edit Settings.



2. GCode Preview Optimization

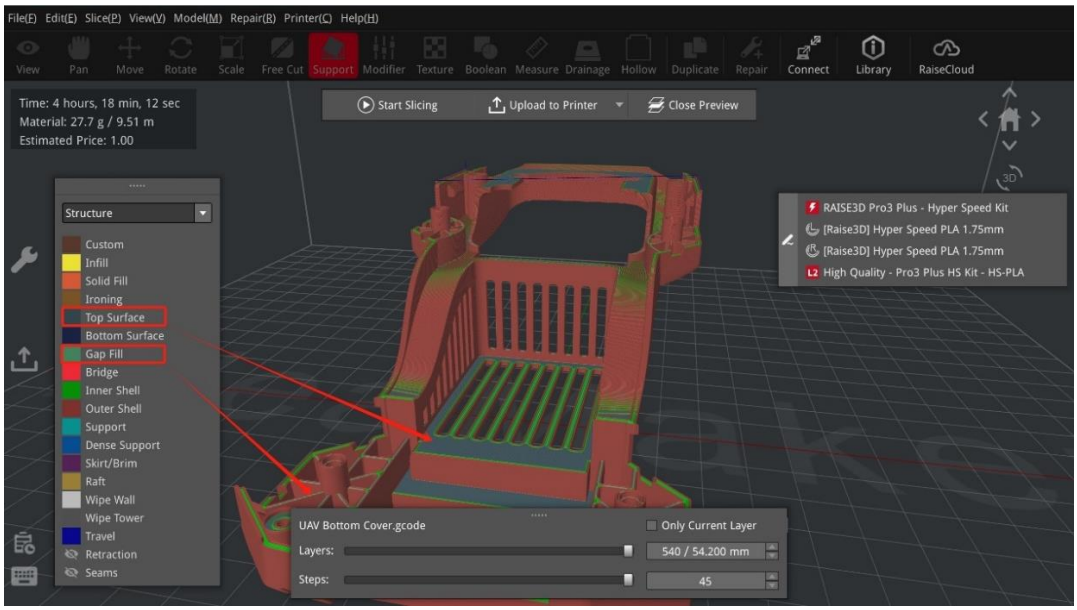
- Slicing preview displays Temperature, Fan Speed, Layer Time
- Slicing preview displays current layer GCode

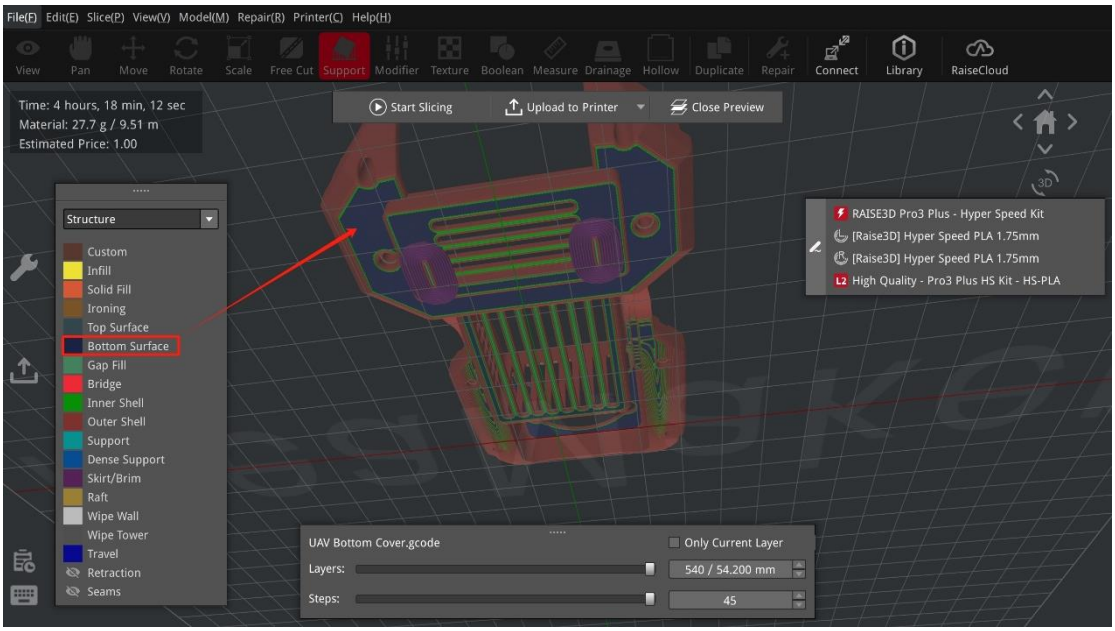


- Slicing preview adds more distinctive colors.
  - Displays top surface, bottom surface, gap filling, and solid fill with different colors in GCode

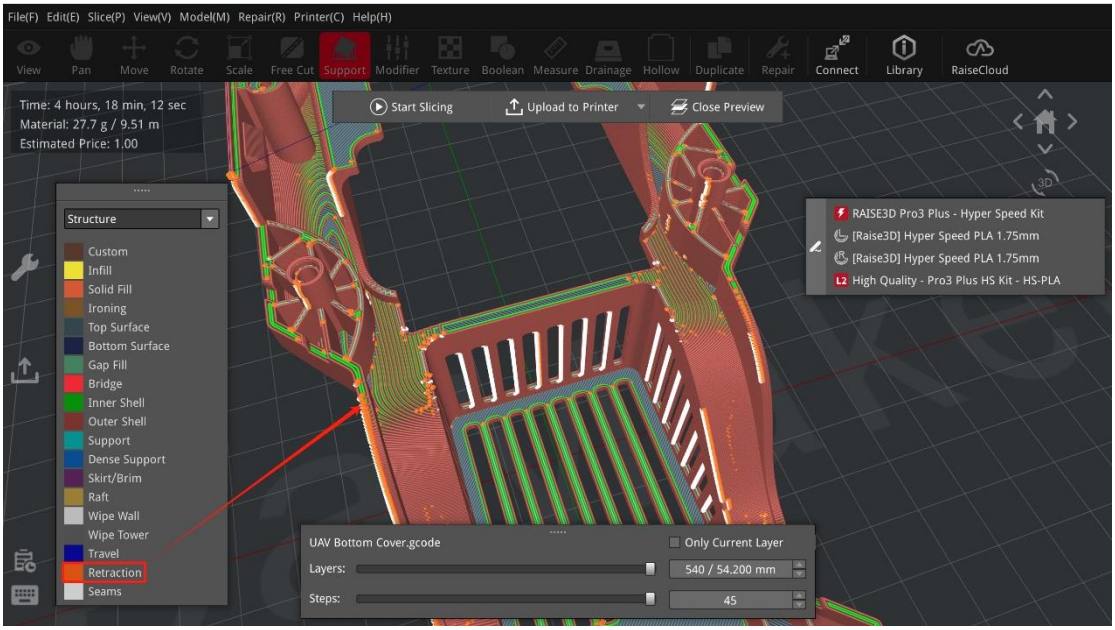
preview.

- Three different feature colors for top surface, bottom surface, and gap filling added.



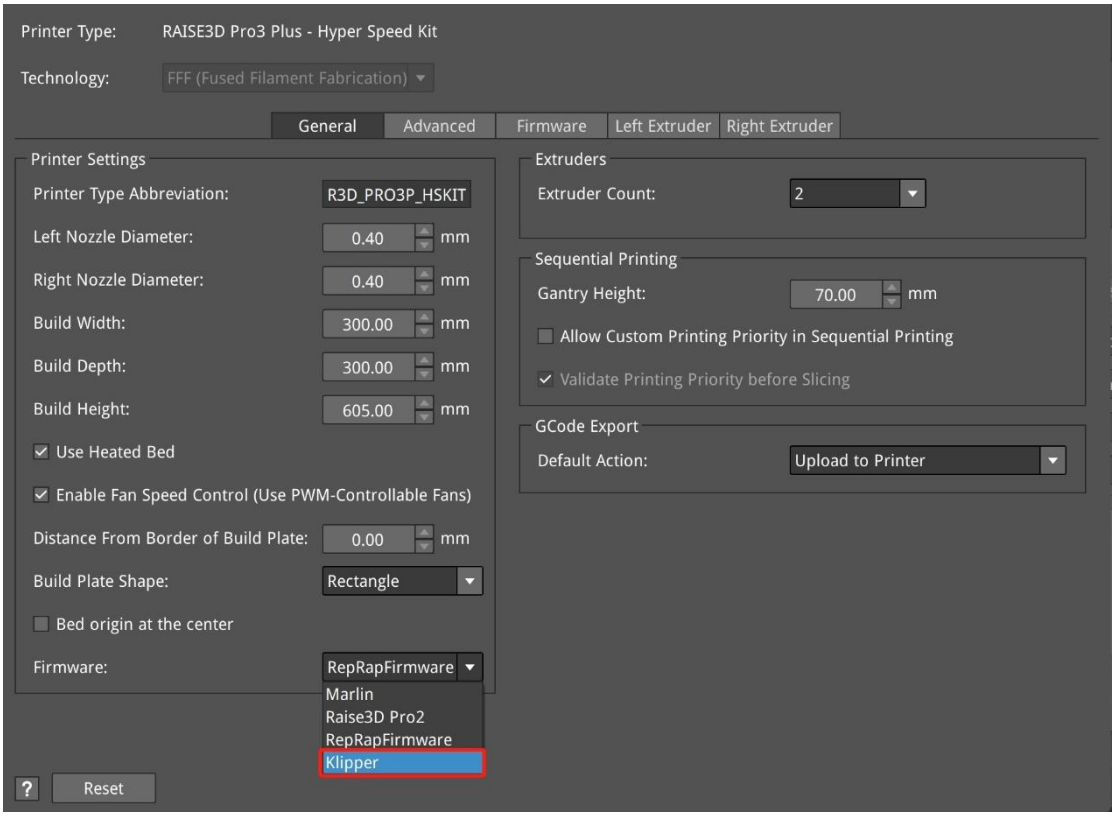


- Optimization of the display of “Retraction” in GCode preview



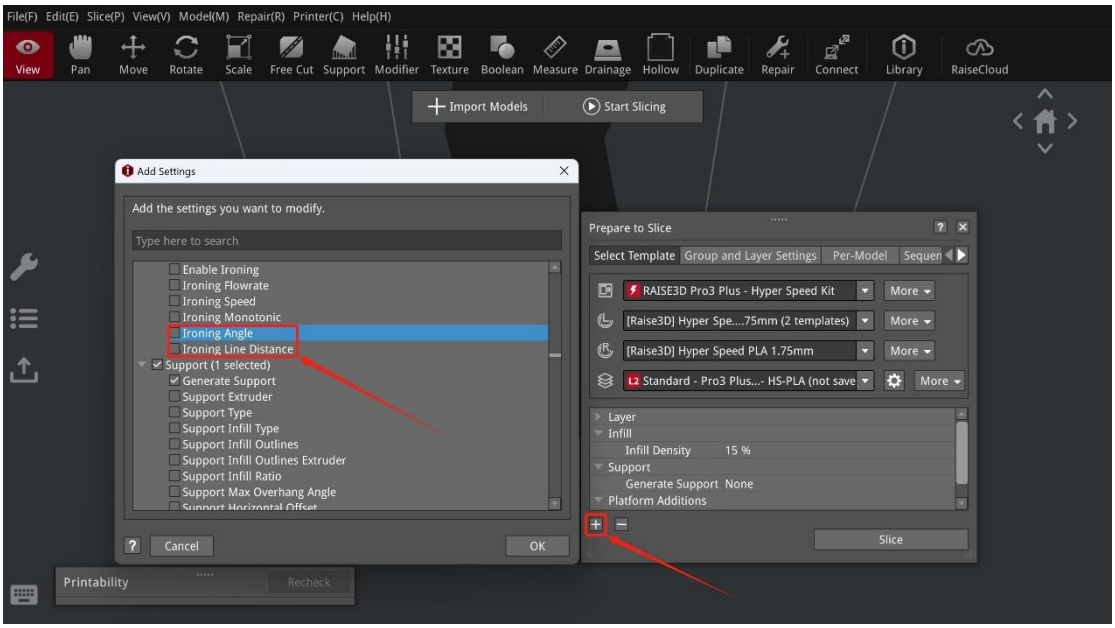
3. ideaMaker Natively Supports Klipper Firmware

- Printer Settings -> Added Klipper option in Firmware Type.

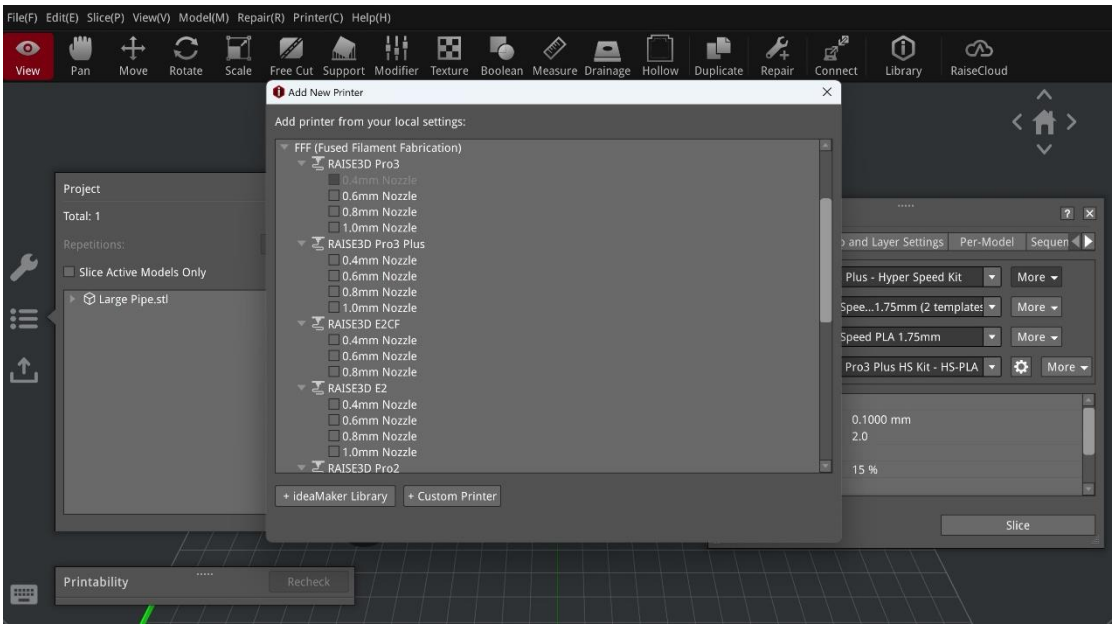


4. Added Slicing Settings for Ironing Line Distance and Ironing Angle.

- Ironing Angle: The angle between the Ironing fill line and the solid fill line.
- Ironing Line Distance: The spacing between adjacent Ironing infill lines, with a default value of 0mm, is equivalent to automatic calculation.

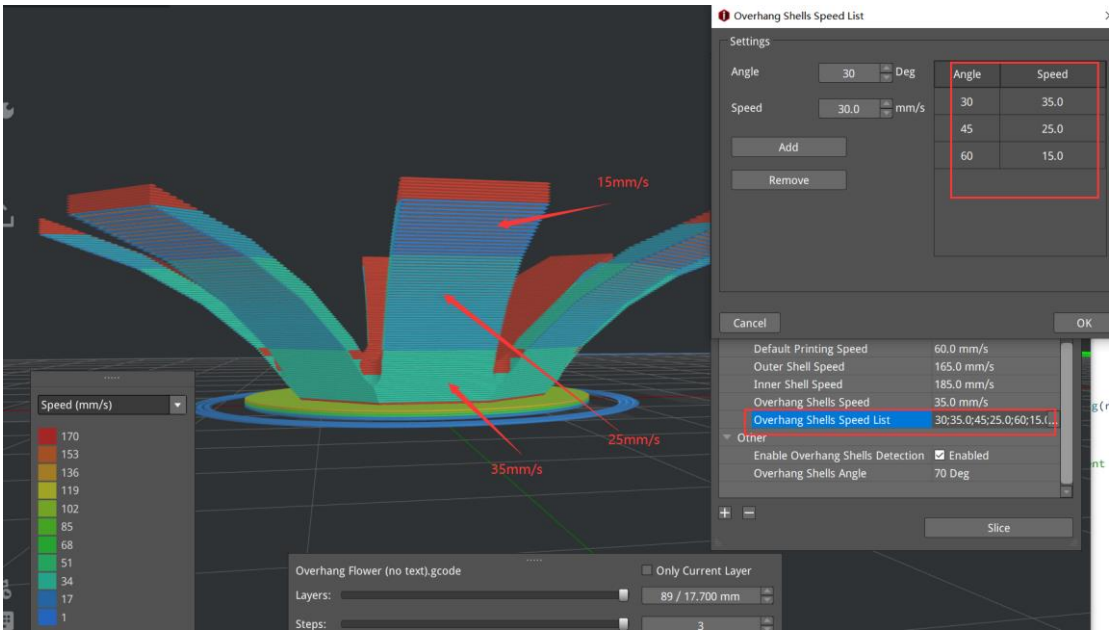


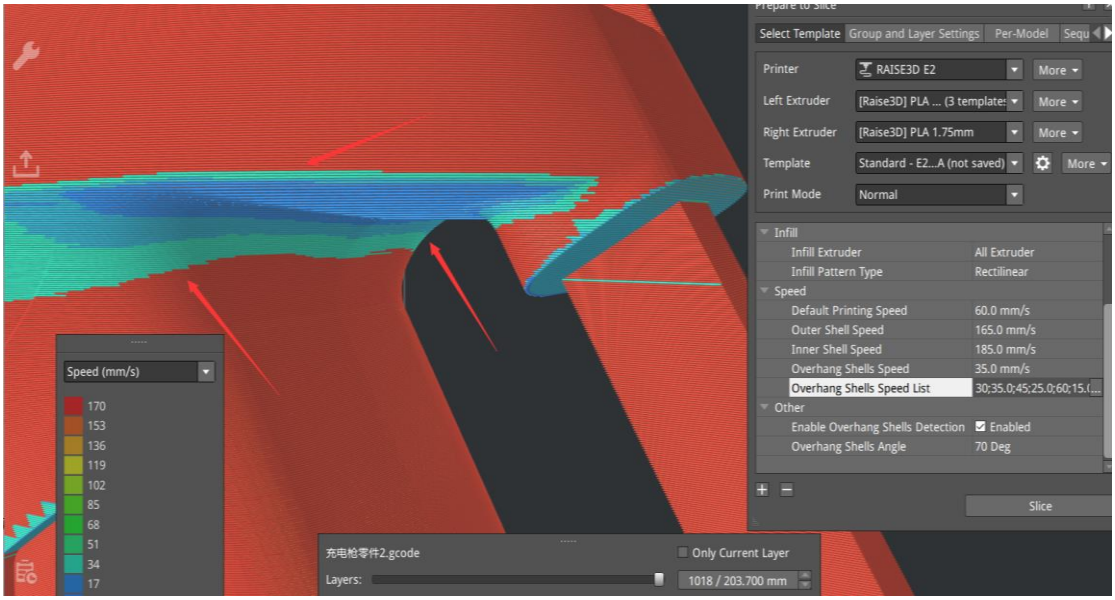
5. Built-in Templates for Non-0.4 Nozzle Official Materials with Official Printers



6. Added Slice Setting “Overhang Shell Speed List”

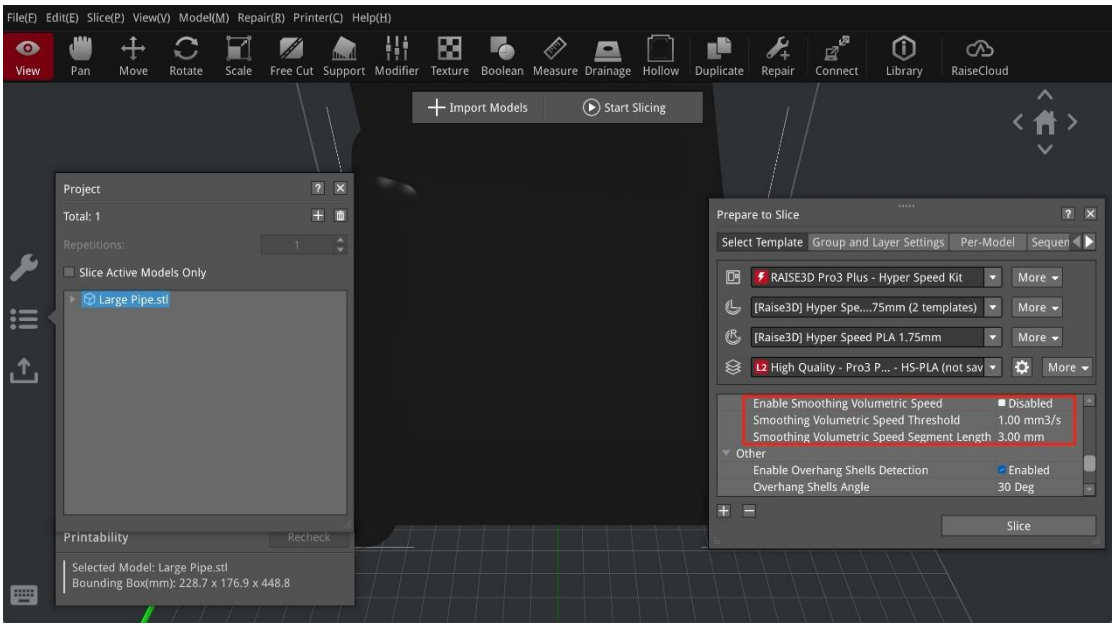
- Dynamically adjusts and uses different overhang shell speeds according to the overhang angle of different areas of the model.



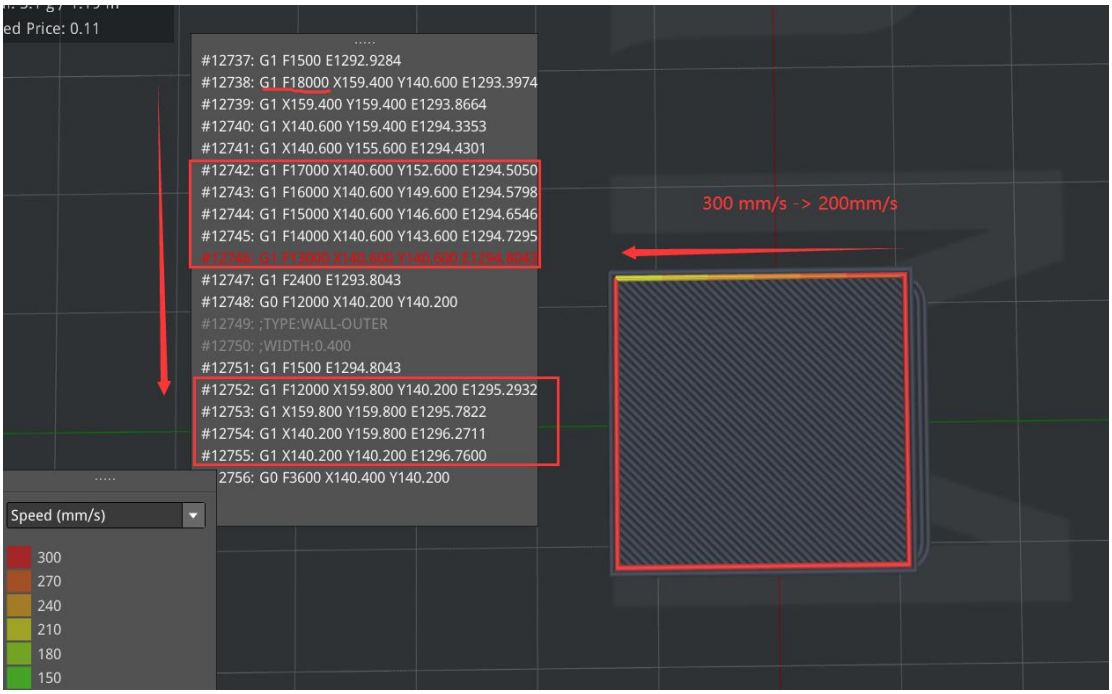


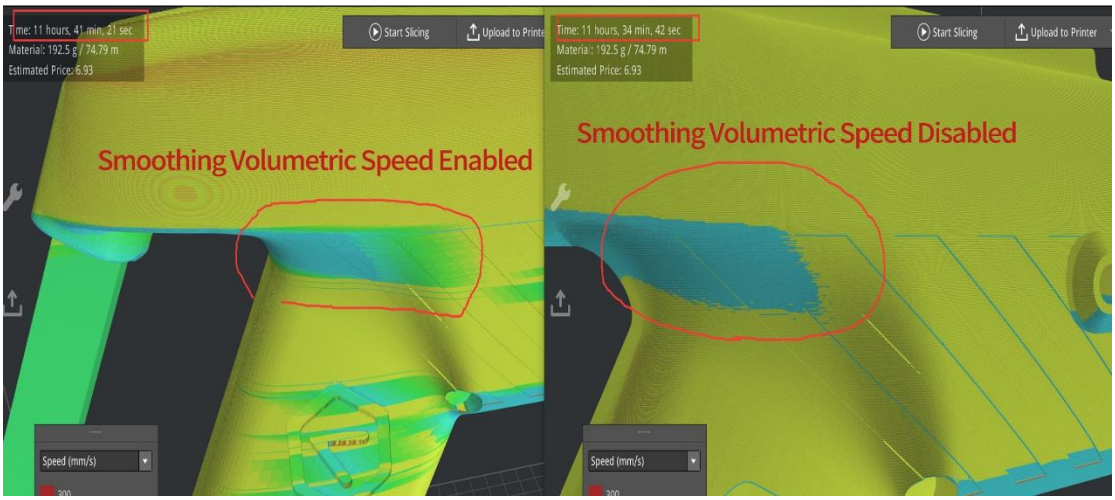
7. Added Slice Setting "Smoothing Volumetric Speed"

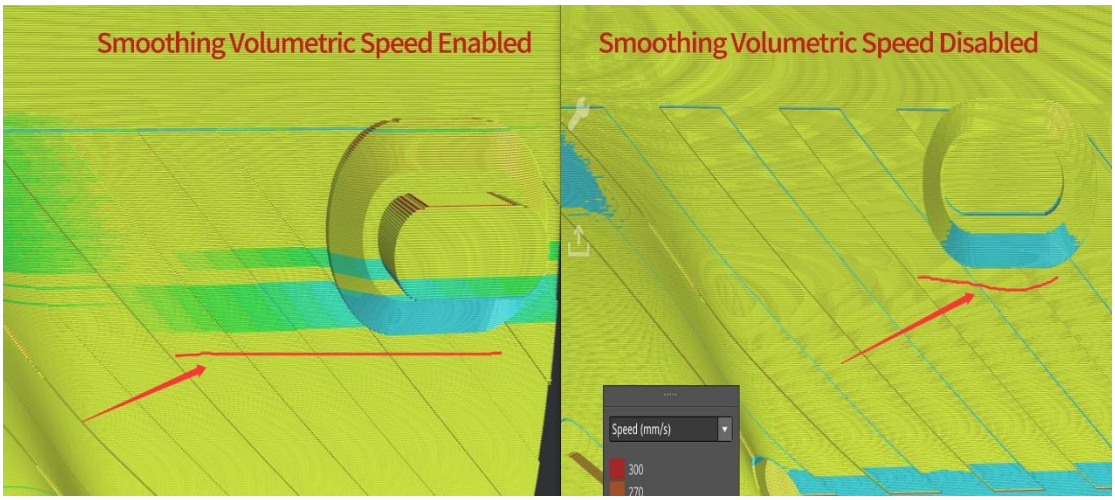
- When the difference in flow rate between different model features is too large, the printing speed is gradually decreased or increased to regulate the transition between different flow rates.

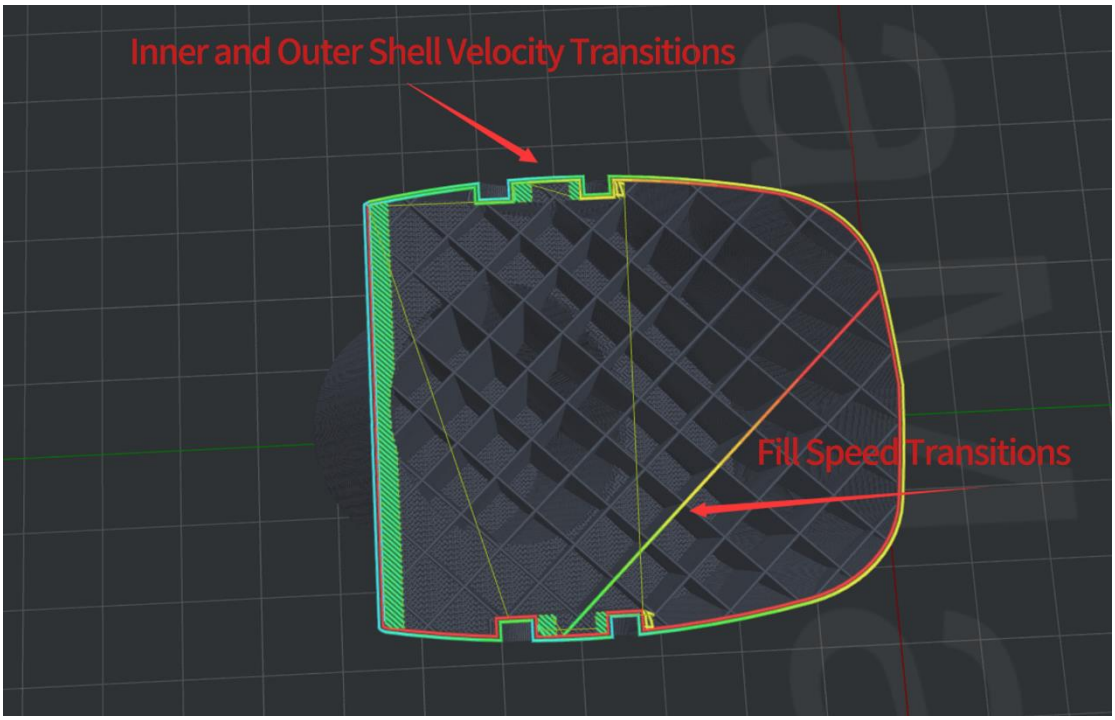


- Effect Comparison



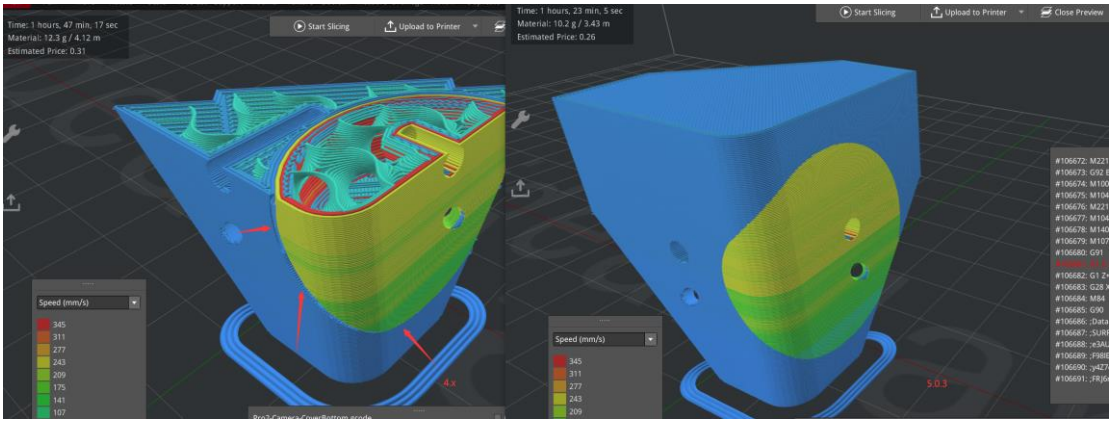

Time: 11 hours, 41 min, 21 sec
Material: 192.5 g / 74.79 m
Estimated Price: 6.93
Start Slicing
Upload to Printe
Smoothing Volumetric Speed Enabled
Speed (mm/s)
Time: 11 hours, 34 min, 42 sec
Material: 192.5 g / 74.79 m
Estimated Price: 6.93
Start Slicing
Upload to Printer
Smoothing Volumetric Speed Disabled
Speed (mm/s)

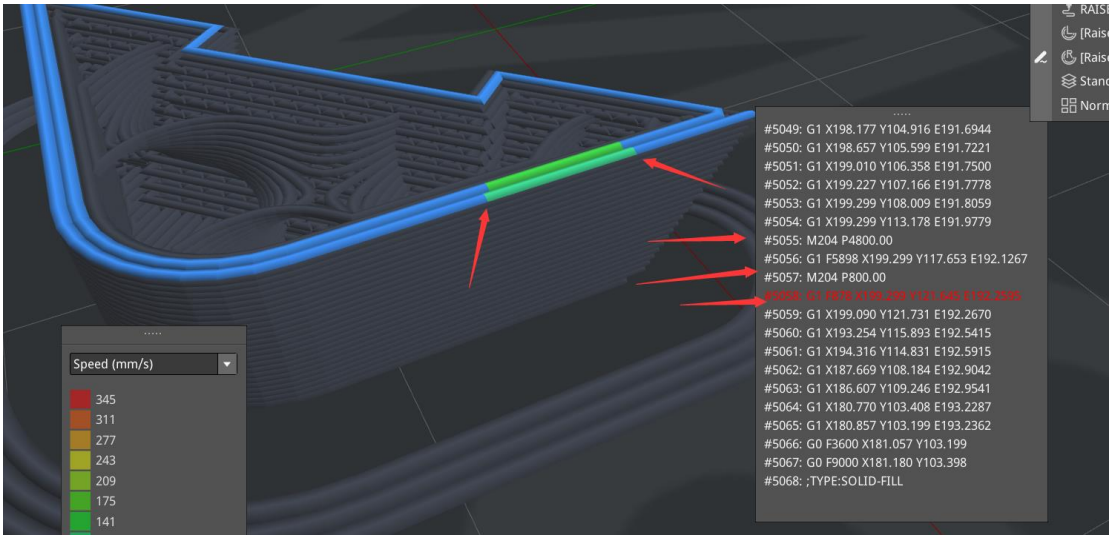

Smoothing Volumetric Speed Enabled
Smoothing Volumetric Speed Disabled
Speed (mm/s)
300
270

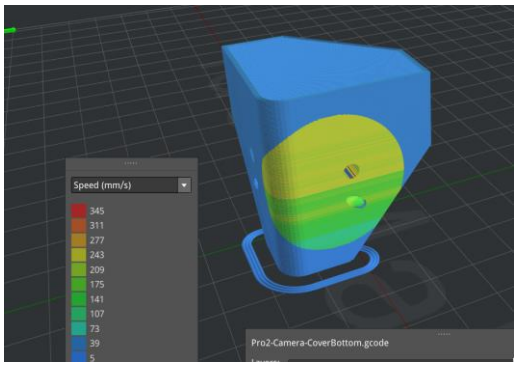


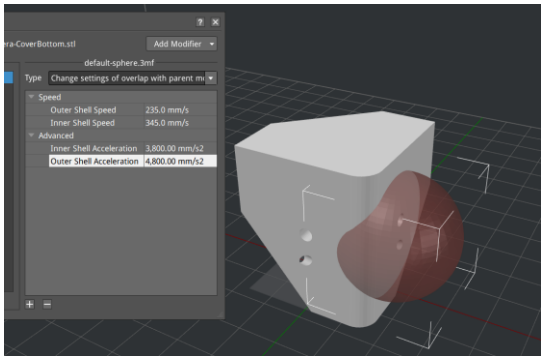
8. Modifying Outer and Inner Shell Velocities, Accelerations, and Jerks Without Generating Independent Shells

- In the new version, when adding different outer shell velocities, inner shell velocities, outer shell accelerations, inner shell accelerations, outer shell jerks, and inner shell jerks in the modifier, multiple separate shells are no longer generated. Instead, the values of velocity, acceleration, and jerk in the G-code path are varied within the same shell.
- This not only significantly improves surface quality but also greatly reduces printing time. For example, in the images below, the estimated time has been reduced by 20% in the new version compared to the old version.

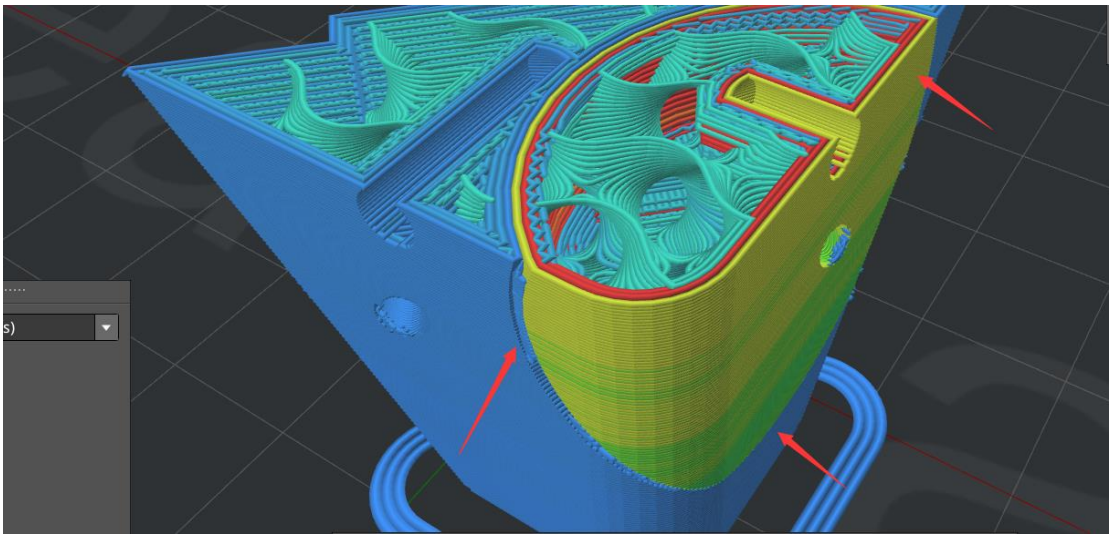






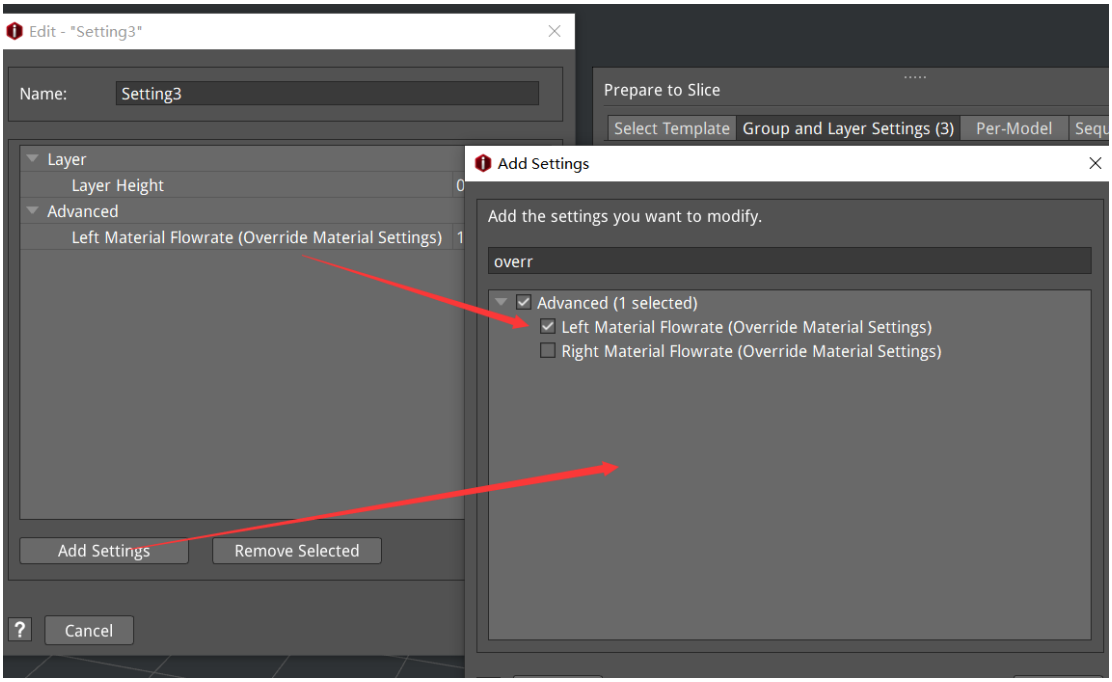


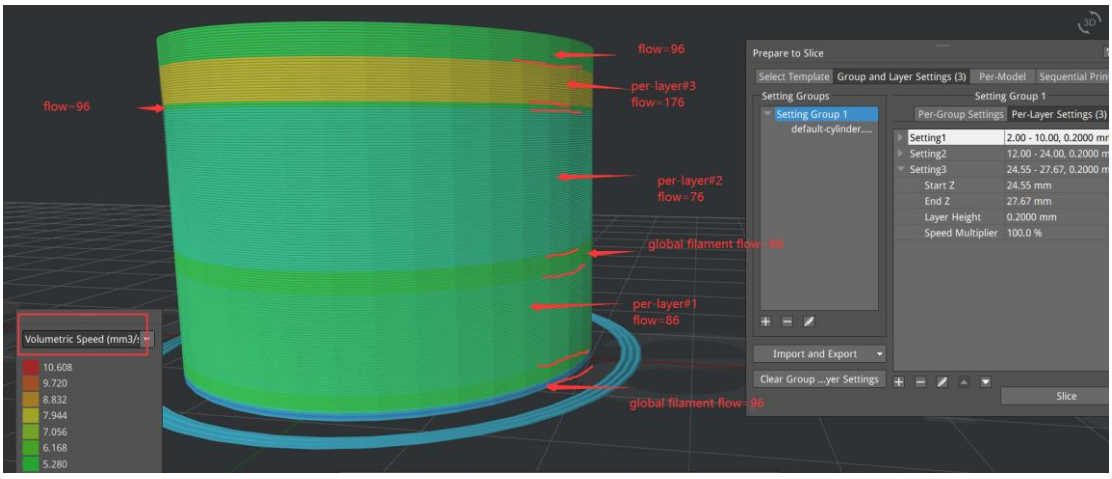
- In comparison to the old version, multiple separate shells were generated, significantly impacting surface quality.



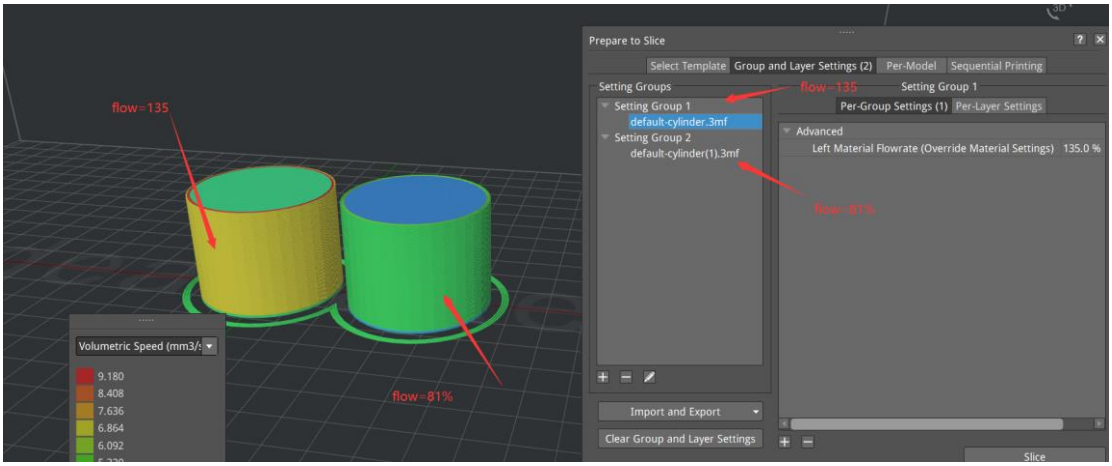
9. The "Per-Layer" and "Per-Group" Settings Allow for Increased Override Filament Flow.

- Examples of Per-Layer



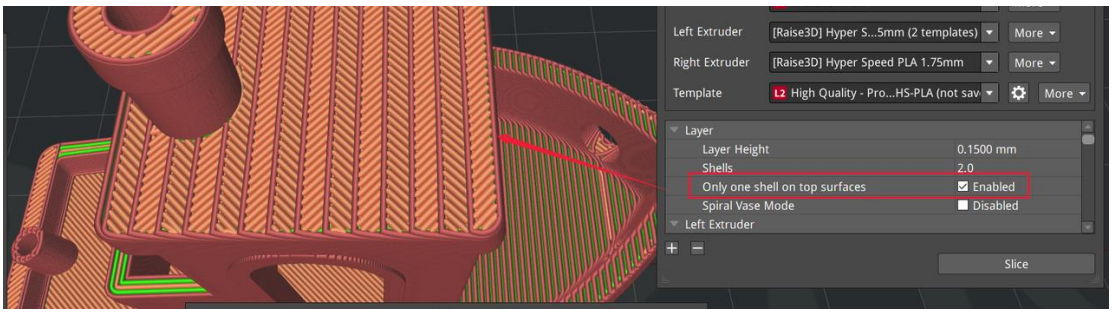


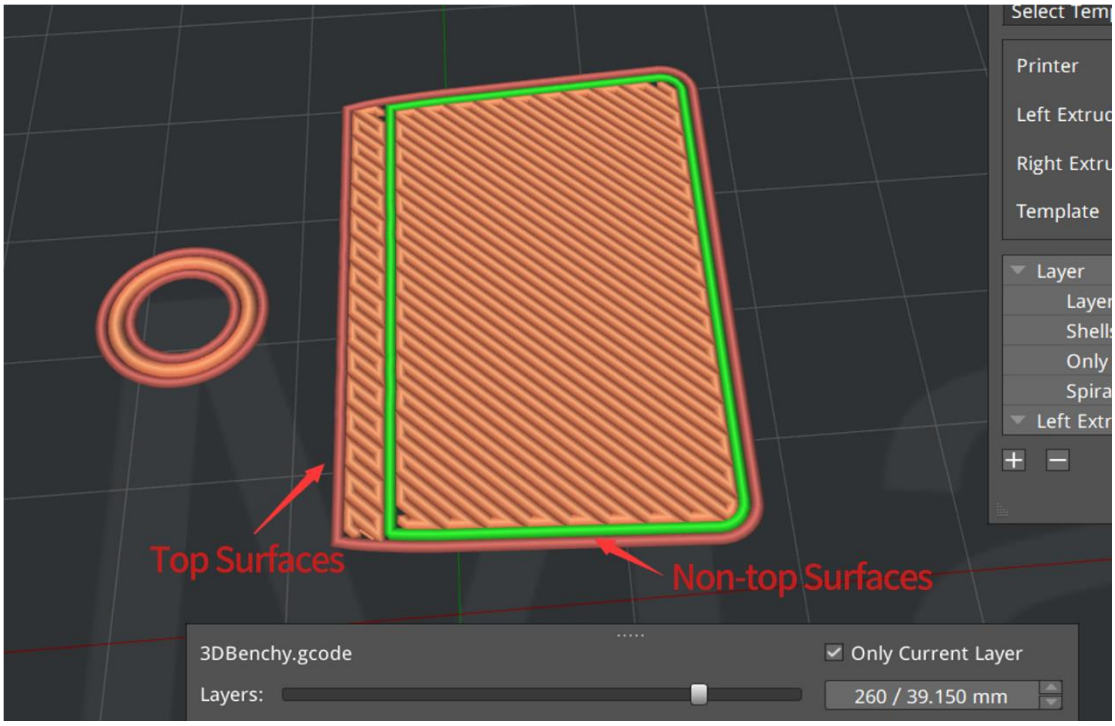
- Examples of Per-Group

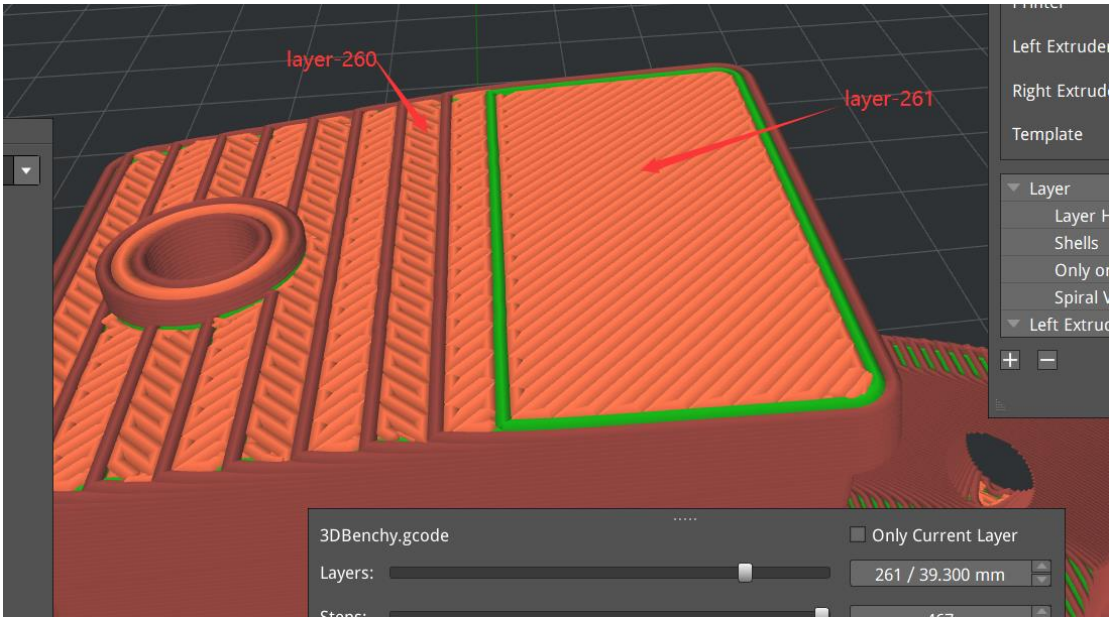


10. New Slicing Setting: Model Top Surface Uses Only One Shell

- Once enabled, all top surfaces of the model will use only one shell, while non-top surfaces will continue to use the shell settings from the original slicing template.
- If both top and non-top surfaces meeting the criteria are present on the same layer, the top surface area will use one shell, while the non-top surface area will use the original number of shells specified in the slicing template.

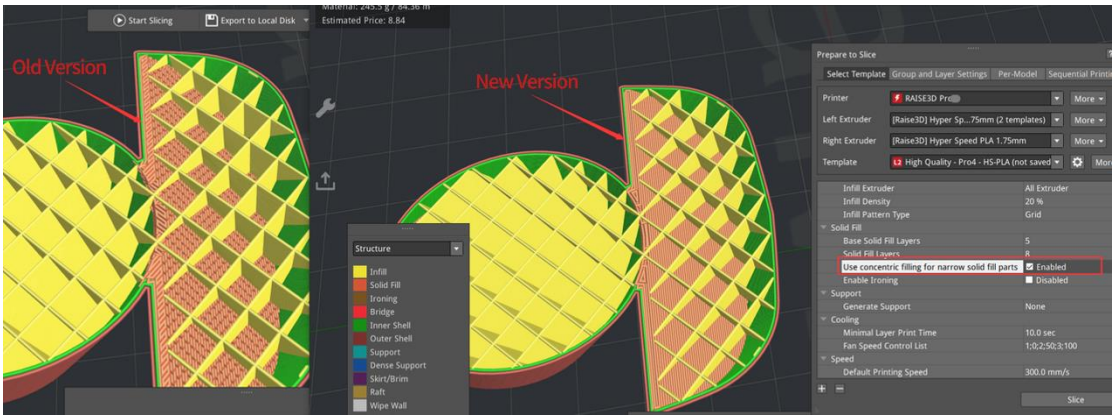




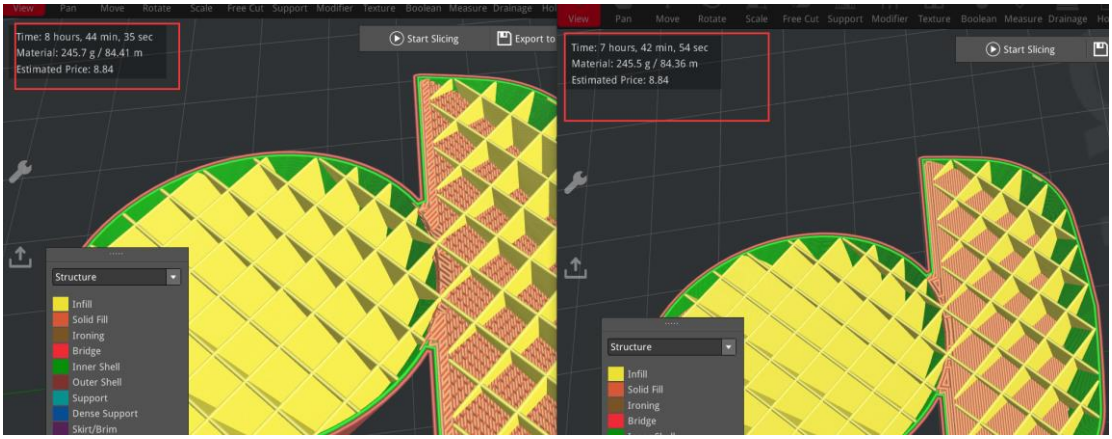


11. Using Concentric Fill Pattern for Narrow Solid Infills

- Previously, narrow areas of solid infill were printed using solid fill patterns such as lines or rectilinear. However, this often resulted in frequent nozzle movements, affecting print quality and increasing printing time. In the new version, narrow areas of solid infill are printed using concentric circles.



- Time Comparison: For models with a significant amount of internal solid infills, the new feature can significantly reduce printing time and minimize frequent nozzle shaking. The image below shows an 11% reduction in printing time after enabling the new feature.



- This only applies to the internal solid infill of the model, without modifying bottom surface, top surface, or bridging structures.

12. Added Two New Placeholders to Handle Custom Start G-code for Single Nozzle Scenarios.

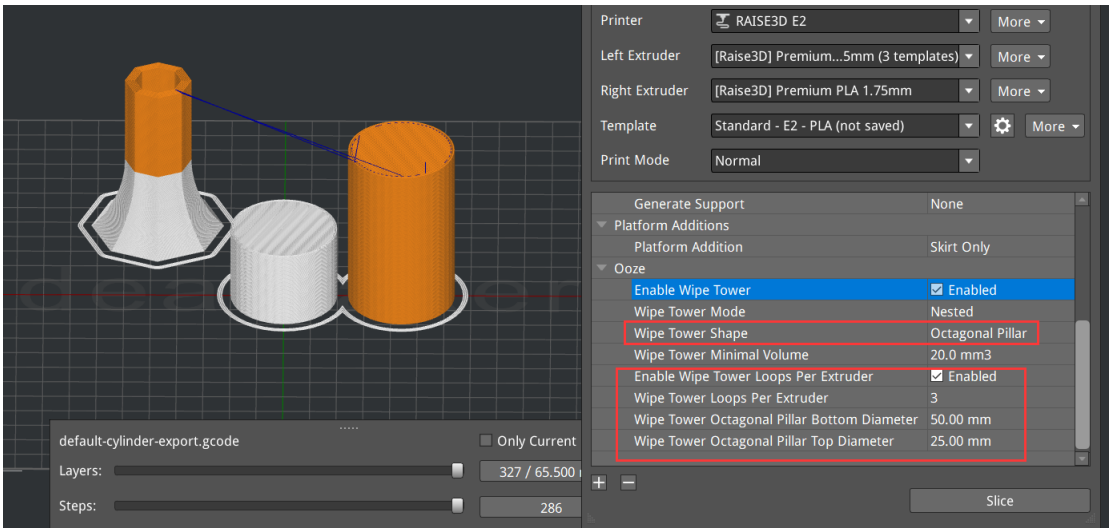
13. Removed the Limit on Texture XY Offset, Extending It to 5mm.

14. Added a New Wipe Tower Type - Octagonal Pillar.

- Added a new Wipe Tower type - Octagonal Pillar.
  - The Wipe Tower Shape in the Advanced Settings dialog has been updated to include the new

Octagonal Pillar option.

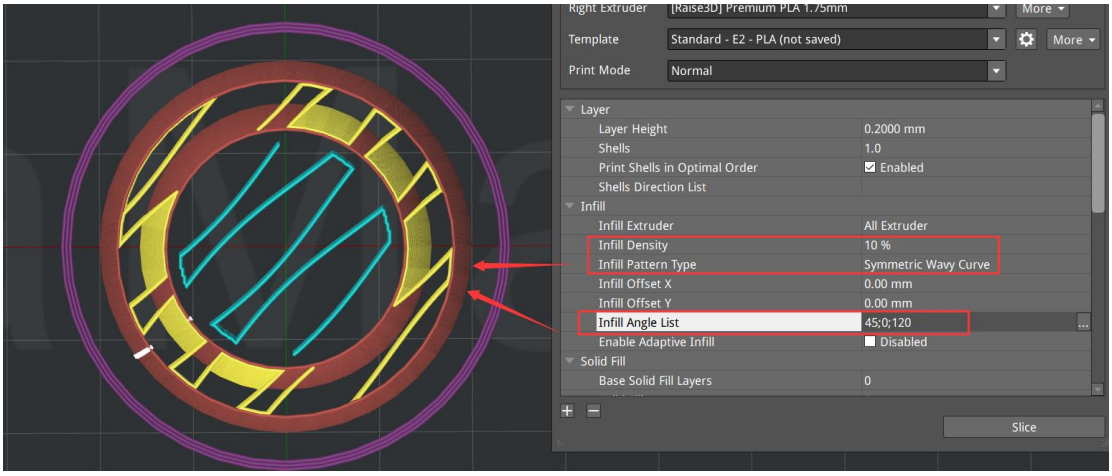
- Added settings for Wipe Tower Octagonal Pillar Bottom Diameter
- Added settings for Wipe Tower Octagonal Pillar Top Diameter.



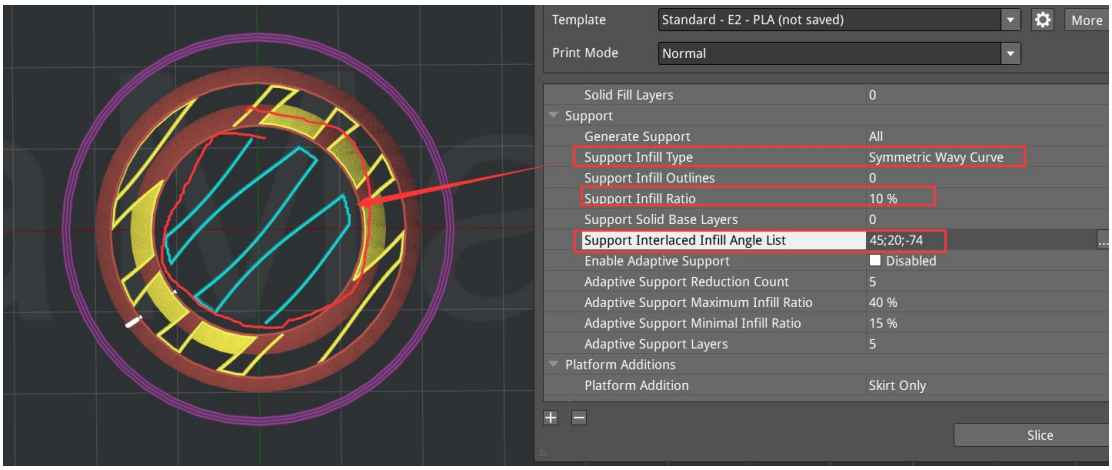
- Added Wipe Tower Loops Per Extruder option, which enforces the number of loops per extruder for the Wipe Tower (wall thickness).
  - Added settings to enable Wipe Tower Loops Per Extruder.
  - Added settings to configure Wipe Tower Loops Per Extruder.

15. Added a New Model Internal Infill and Support Infill Type - Symmetric Wavy Curve.

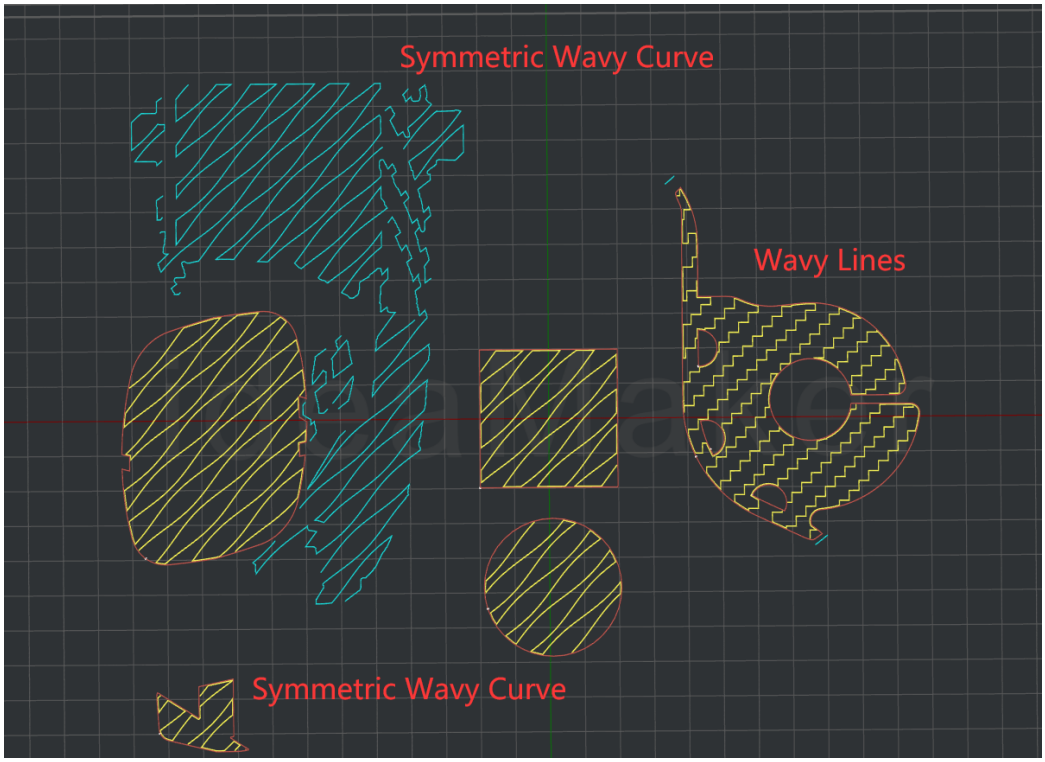
- Added a new model internal infill - Symmetric Wavy Curve.



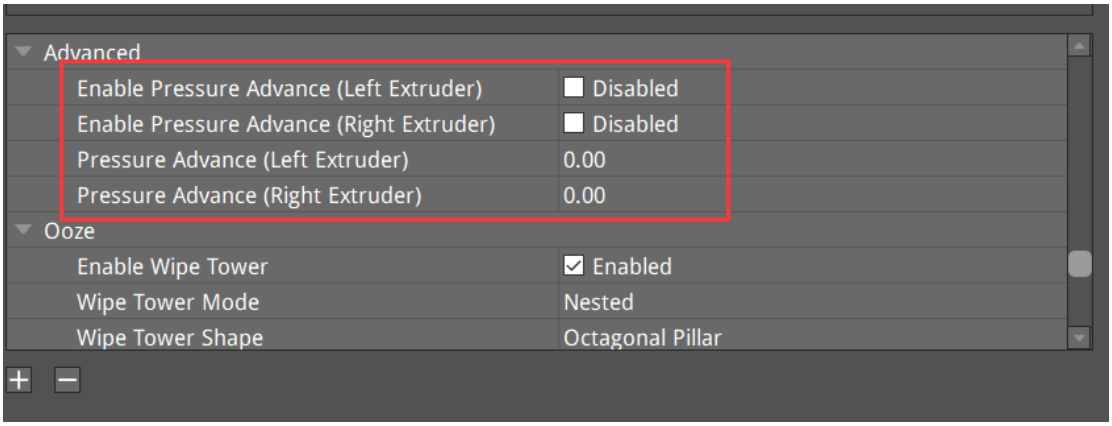
- Added a new support infill type - Symmetric Wavy Curve.



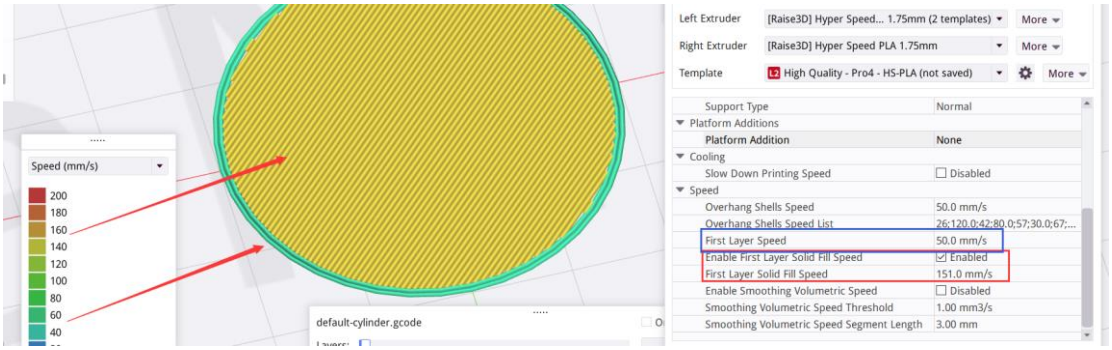
- Opened up a new model internal infill type - Wavy Lines.

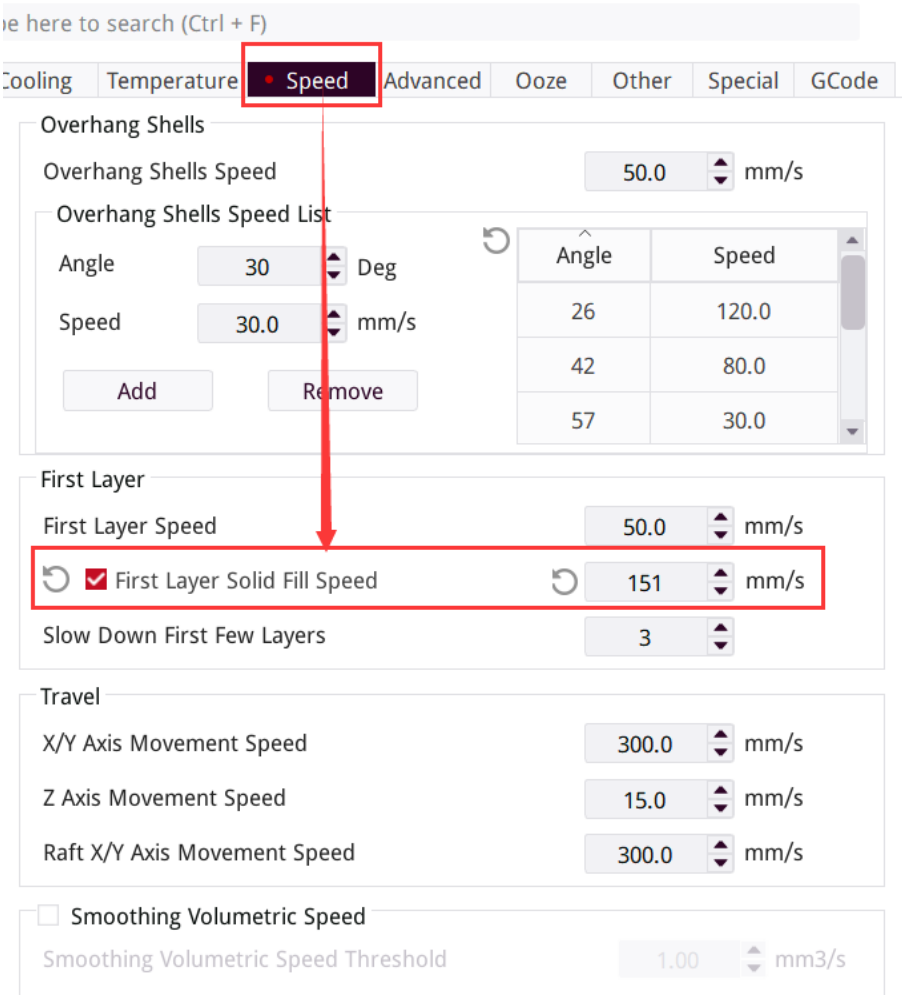


16. Added Pressure Advance Setting



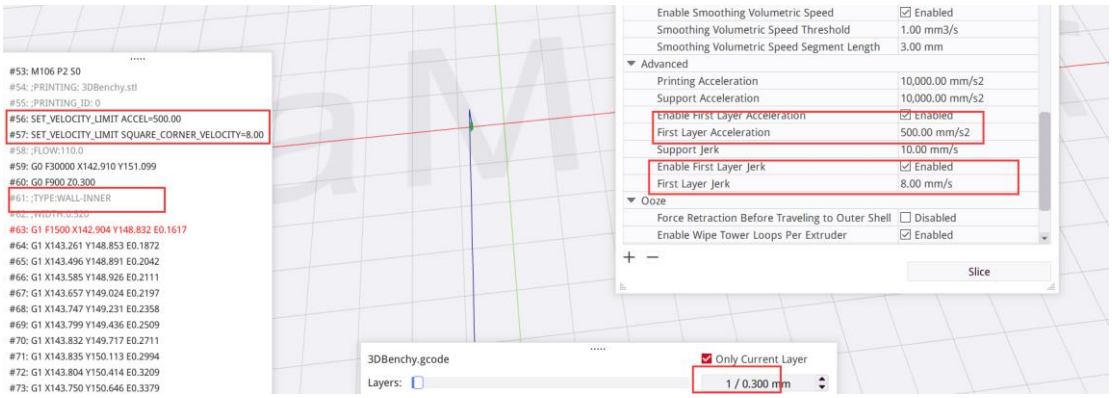
17. Added Slicing Parameter: First Layer Solid Fill Speed

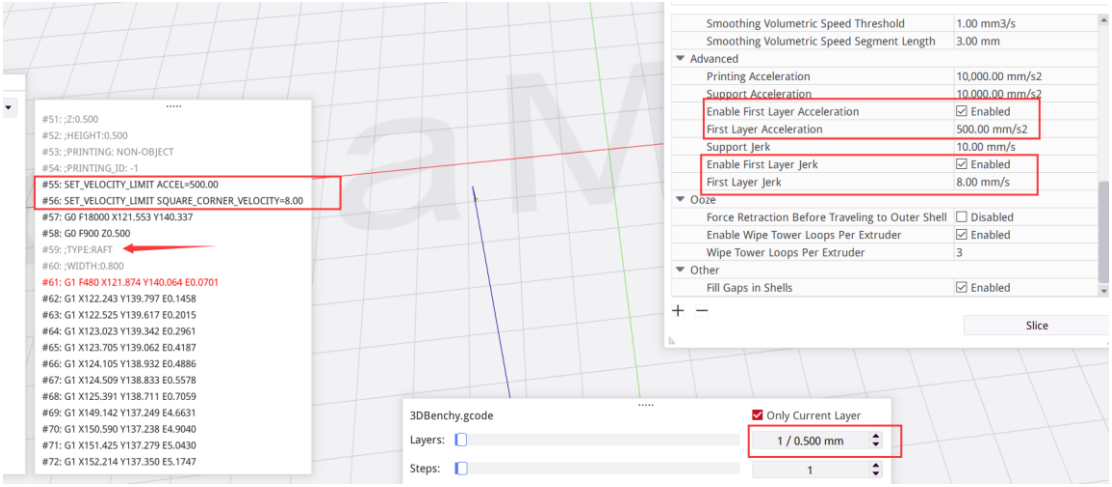




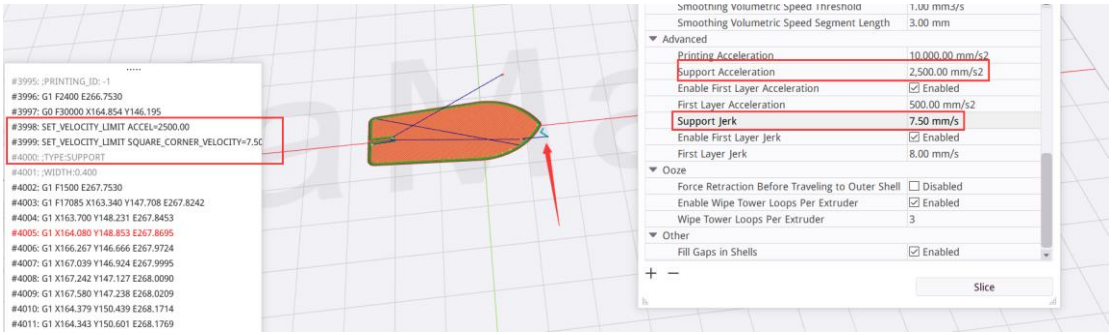
18. New Slicing Options

- First Layer Acceleration, First Layer Jerk
  - Once enabled, all features in the first layer contacting the build platform will uniformly use this acceleration and jerk.

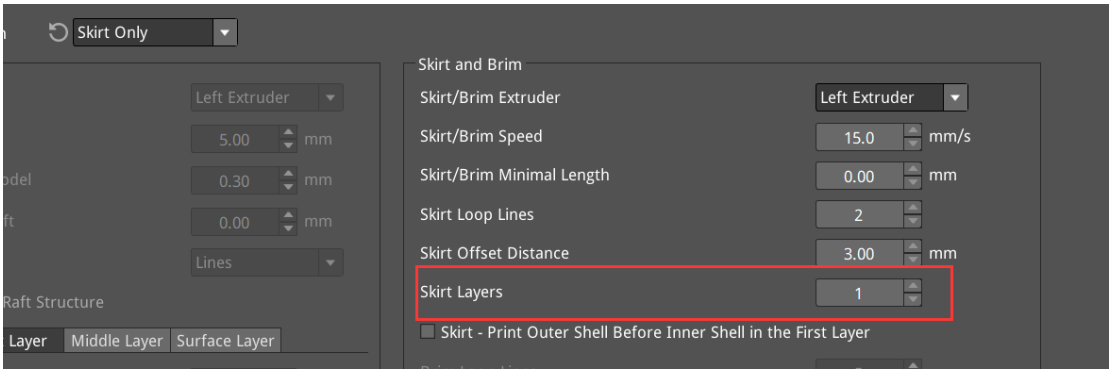




- Support Acceleration, Support Jerk
  - Control the acceleration and jerk for supports and thick supports.

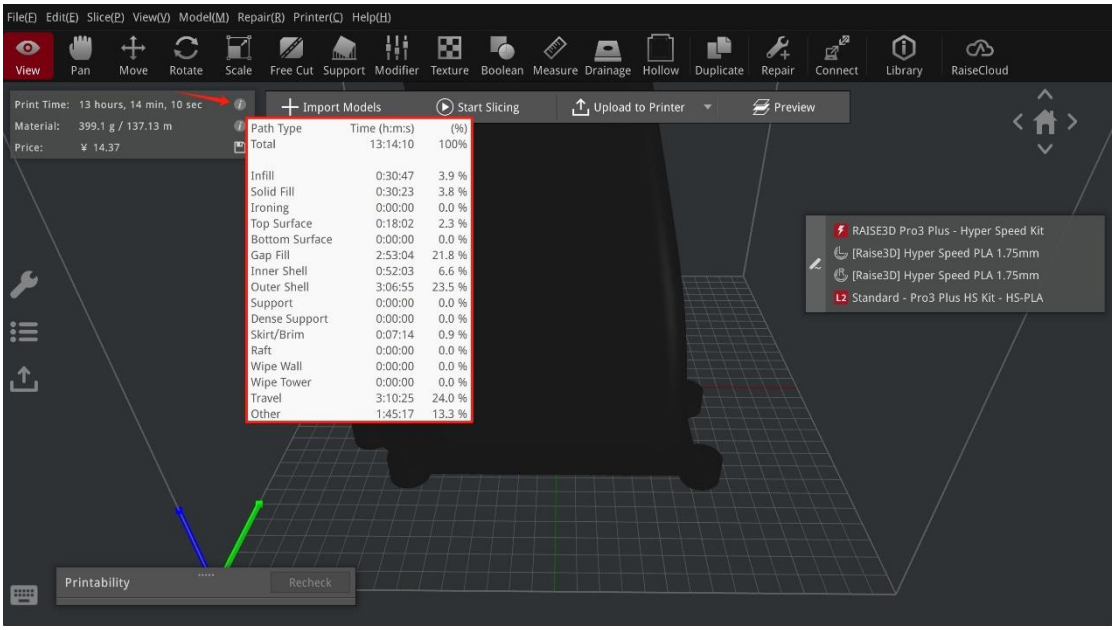


19. Added Slicing Parameter: Skirt Layers

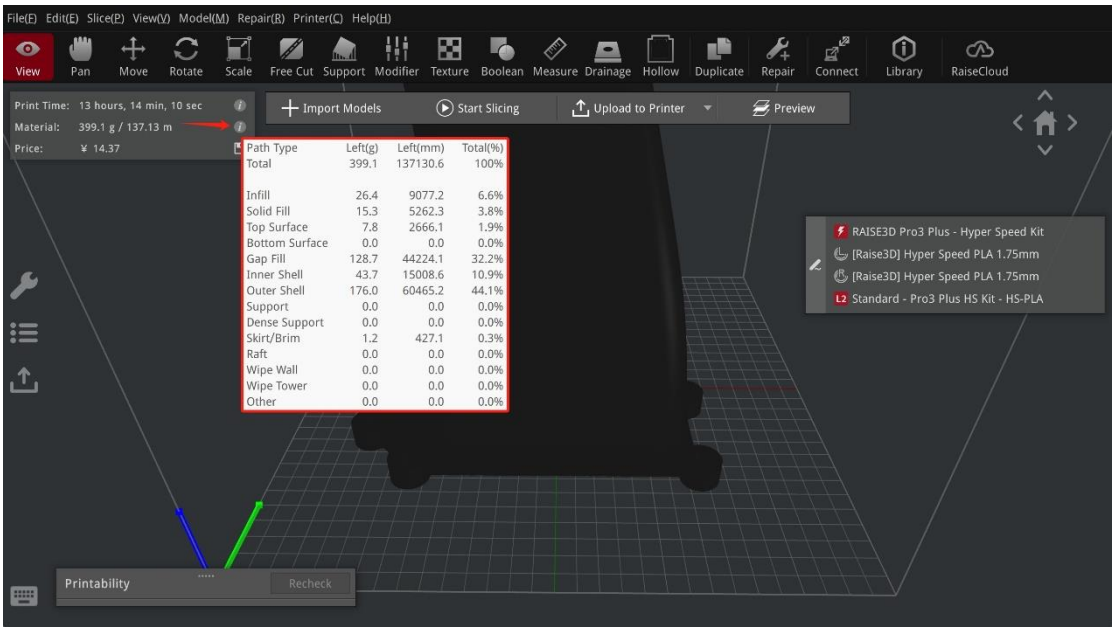


20. Slice Result Panel

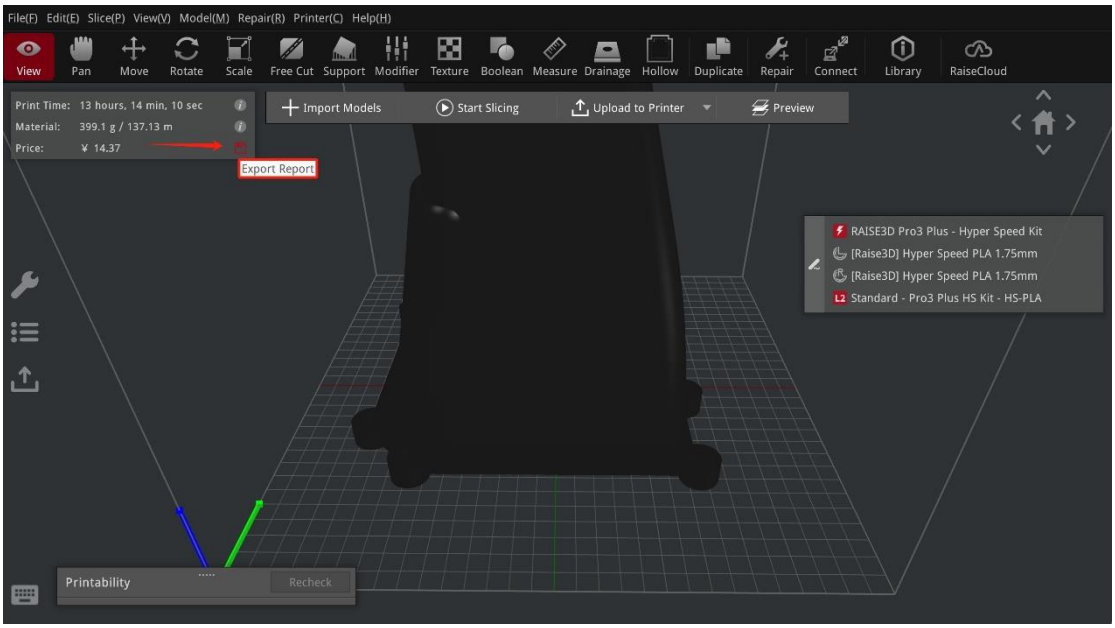
- FFF display printing time for each Path Type (preview of external gcode not displayed in interface)



- FFF display material usage for each Path Type (preview of external gcode not displayed in interface)

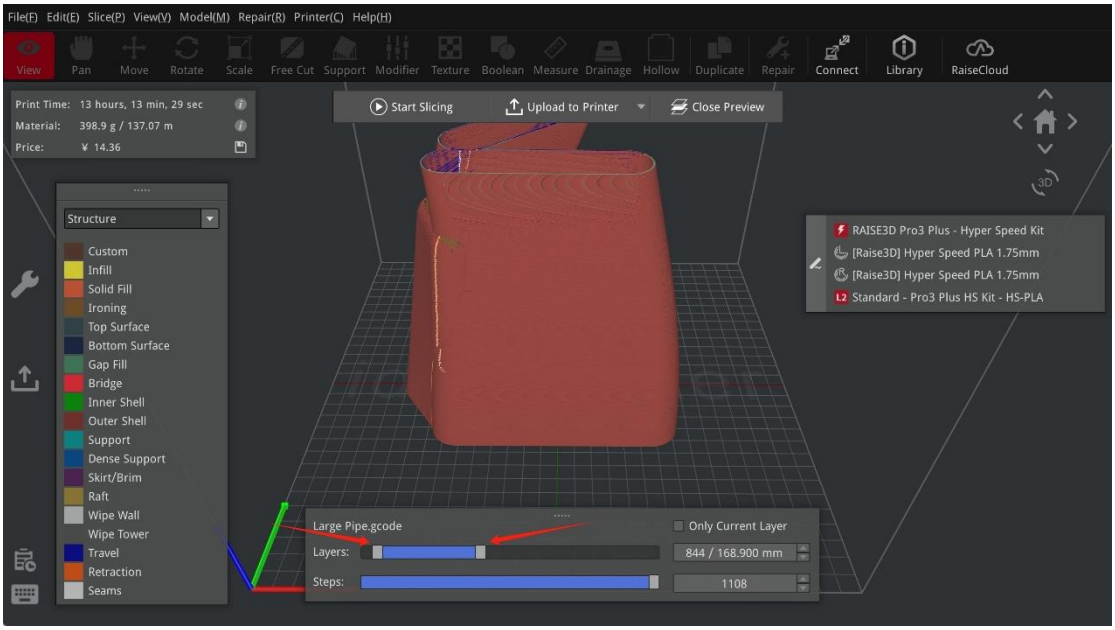


- Export slicing results report button provided on the right side of the price information (preview of external gcode, dlpcode not displayed in interface)



21. Support editing interface and optimized GCode preview scrollbar.

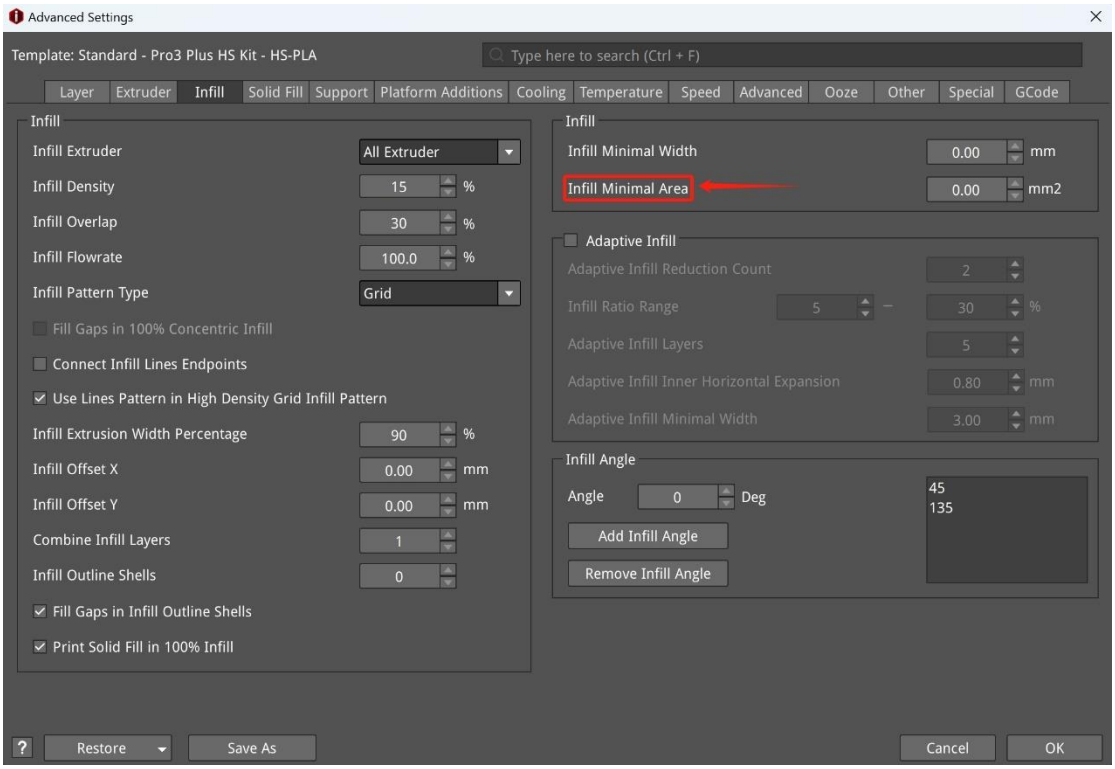
- FFF GCode preview supports simultaneous control of bottom and top layers.



22. Added the new parameter: Infill Minimal Area

- For infill regions with an area smaller than the specified value, ideaMaker will replace those infill regions with Solid Fill for printing. When used in conjunction with "use concentric filling for narrow solid fill parts," it can reduce vibrations during the printing process. This can also reduce printing time for certain model

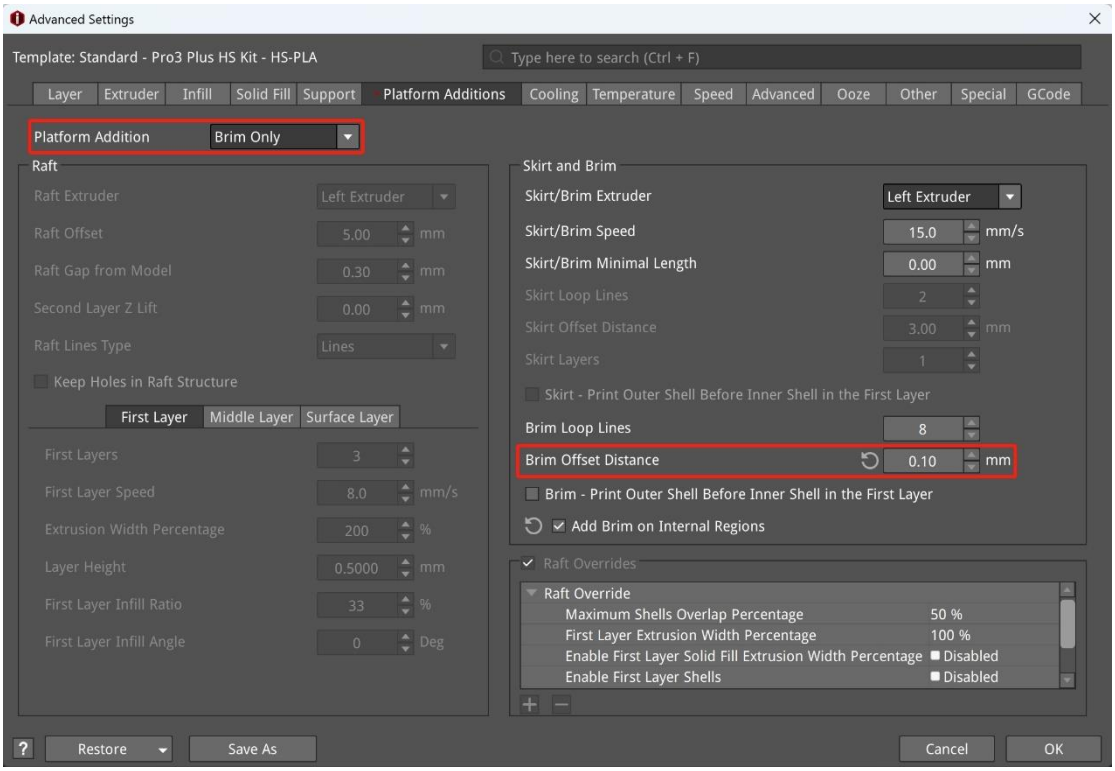
structures.

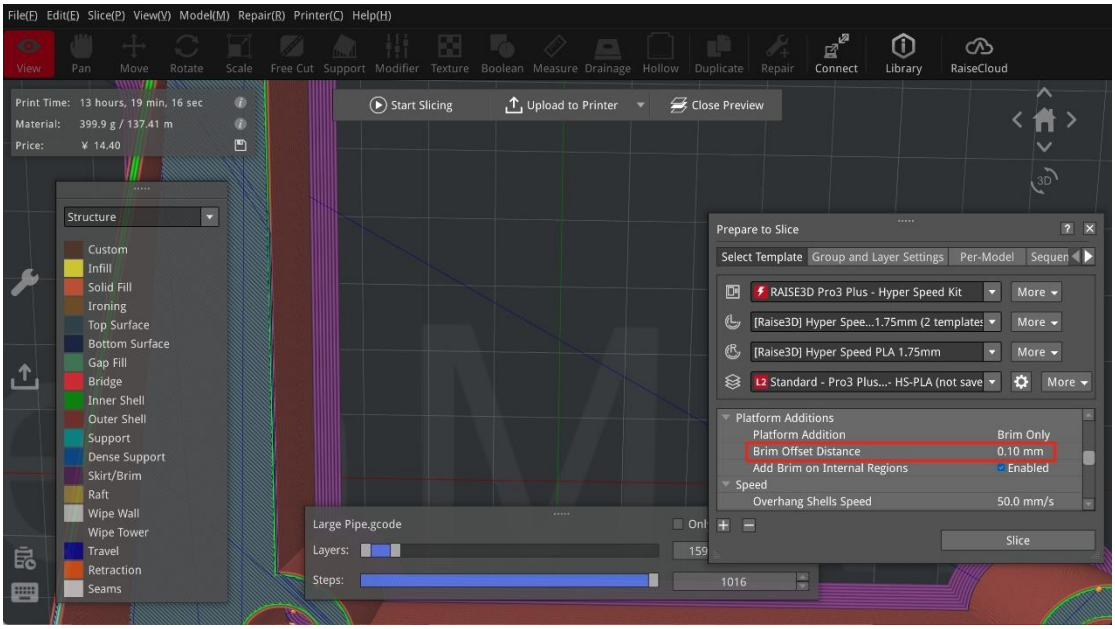


23. Added the slicing parameter: Brim Offset Distance

- Functionality: Controls the distance between the Brim and the first layer of the model, making it easier to remove the Brim from the model. Recommended values are between 0.05 to 0.1mm.

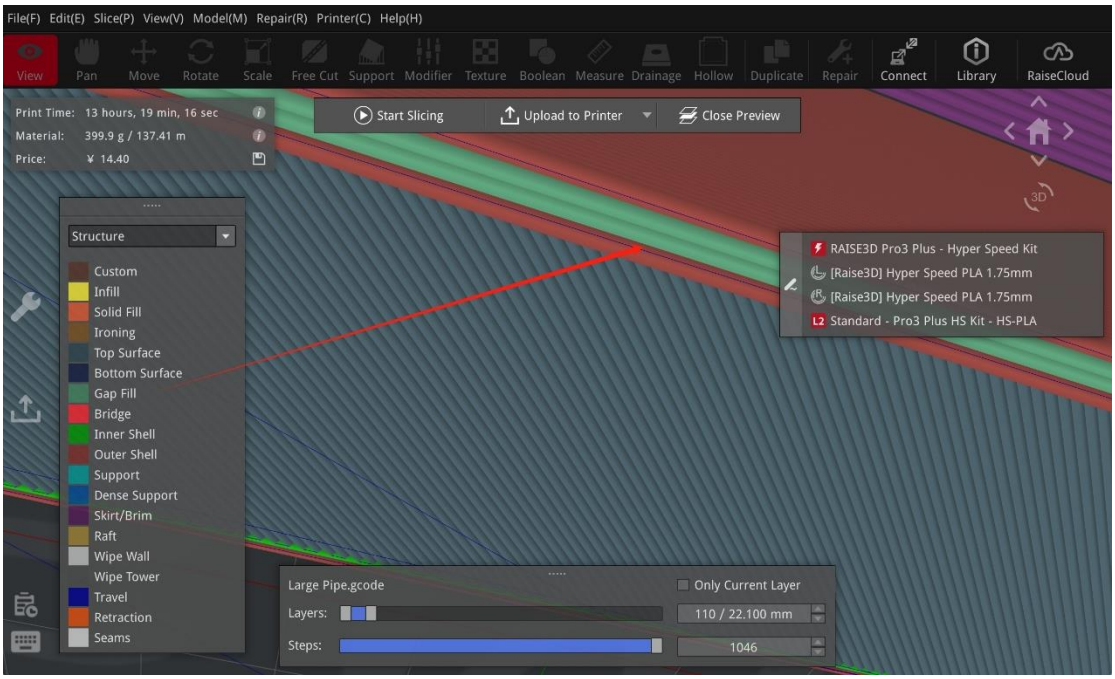
  In previous versions, the gap between the Brim and the model was 0mm, often resulting in difficulty removing the Brim completely from the model.



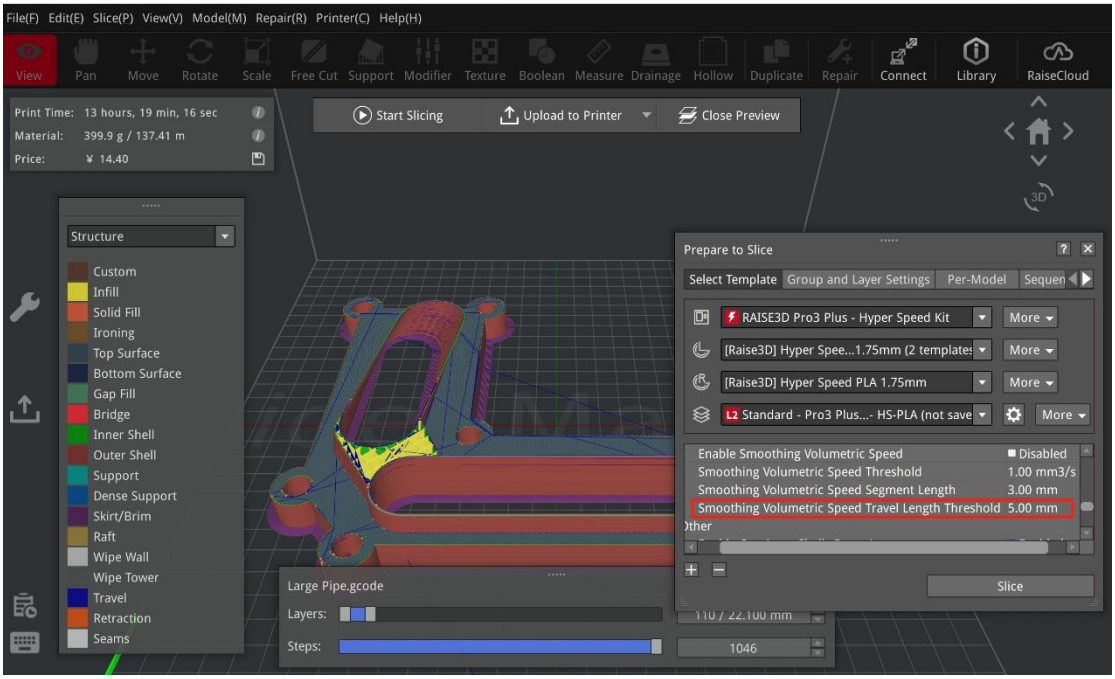


24. Improvement in GapFilling strategy to reduce vibration

- For wider gaps where a single GapFilling line cannot fill the area, use two GapFilling lines with smaller line widths to fill the gap.



25. Open Slicing Parameter: Smoothing Volumetric Speed Travel Length Threshold Option

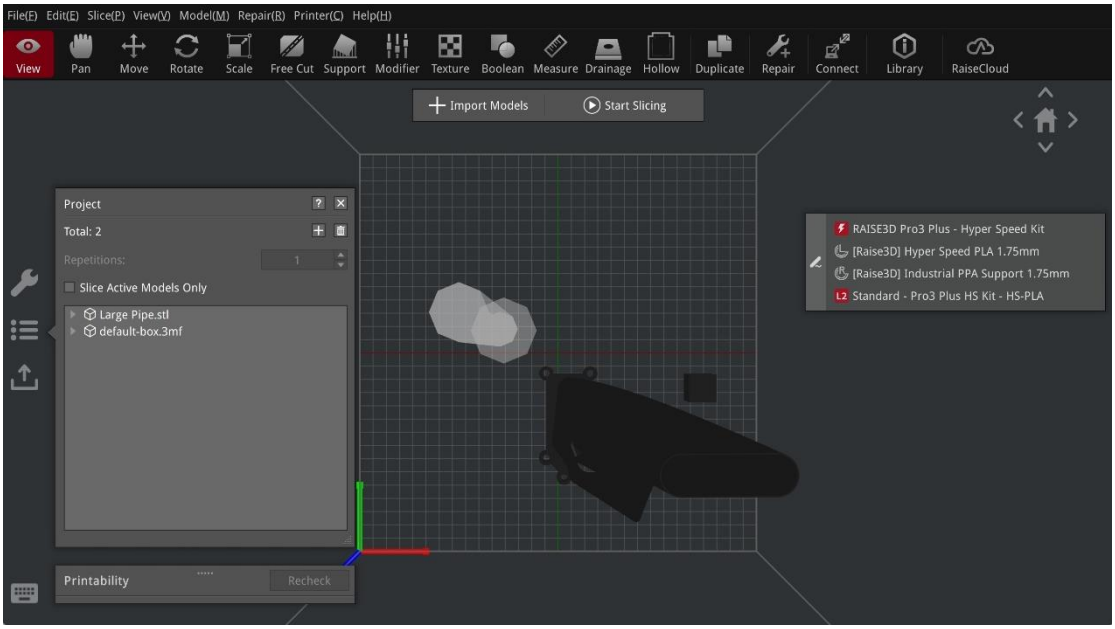


26. Slice template and some functional parameter modifications:

- In all slicing templates for the RMF500, slice settings, PET CF Support - RMF500 material override settings, and Park Position coordinates are uniformly modified to (50, 0).
- For FFF printer models, when selecting slice templates, the Standard template is prioritized whenever possible (DLP printer model rules remain unchanged, still prioritizing the Light template).
- In ideaMaker's DF2 firmware settings, the maximum Z-axis speed in old versions was 350mm/min, in the new version it is modified to 300mm/min.
- The default wall thickness for hollows has been modified to 3.0 mm.

27. Display Wipe Tower Before Slicing:

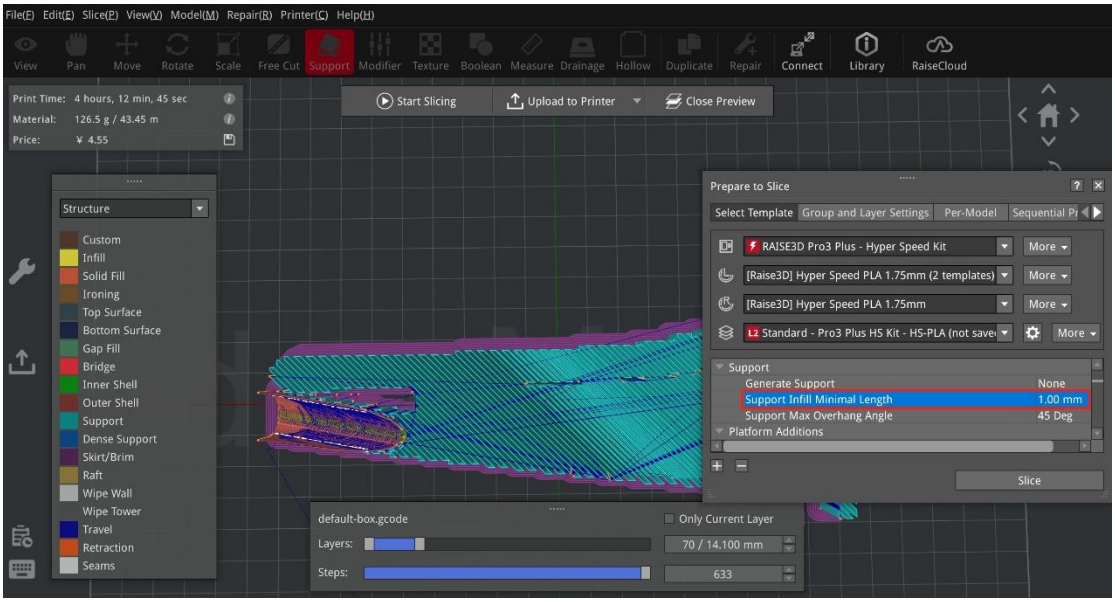
- Based on the nozzle usage status on the current platform, changes in the position and dimensions of the model, settings of the slicing template, and settings of layer grouping, the estimated position of the Wipe Tower is displayed in real-time.



28. Open slicing parameter option: Gap Filling Minimal Length

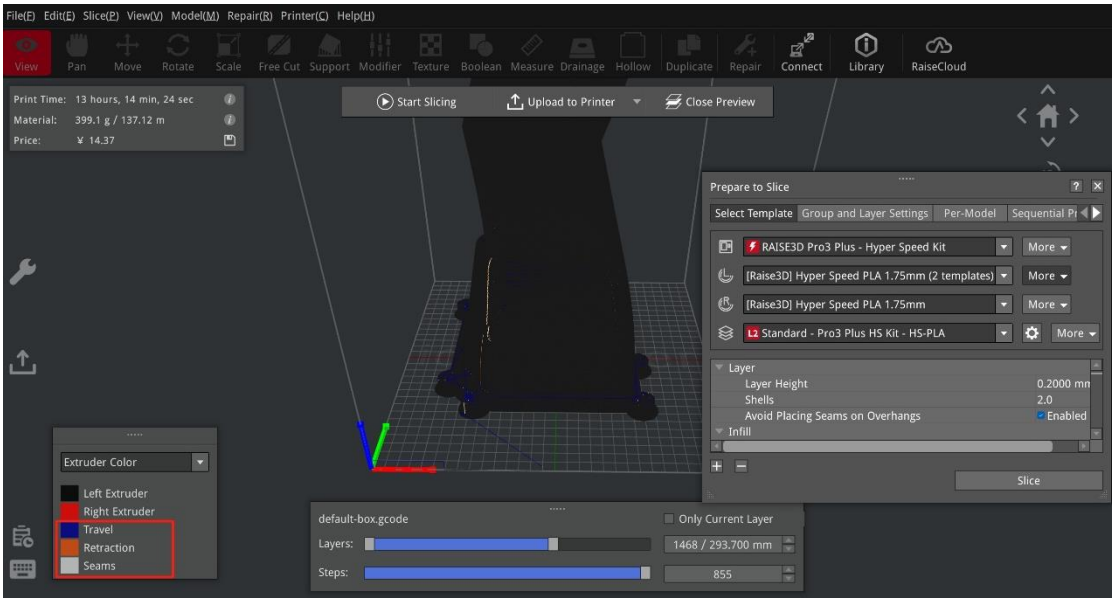
29. New slicing parameter: Minimum Support Fill Length

- Functionality: Deletes supports with fill length below the threshold value.



30. Adjustments to GCode preview:

- In Extruders mode, users can also choose to enable/disable the display of Travel, Retraction, and Seams.



New Printer Model

Resin Printers

1. RAISE3D DF2

FFF Printers

1. RAISE3D RMF500

2. No distinction between high-speed mode and normal mode. Change the models under the original high-speed mode to the following models.

- RAISE3D Pro2 - Hyper Speed
- RAISE3D Pro2 Plus - Hyper Speed
- RAISE3D Pro3 - Hyper Speed Kit
- RAISE3D Pro3 Plus - Hyper Speed Kit

3. Adjustments to RMF500 Printer Parameters

- Adjustment of RMF500 printer settings: Left Extruder -> Customize Extruder Printable Region -> X Size modified to 535mm.

New Material

Resin

- [Raise3D] Touch 2K Grey V1
- [Raise3D] High Detail Apricot V1
- [Raise3D] Standard White V1
- [Raise3D] Rigid 3K Grey V1

Filament

- New materials for RAISE3D Pro3 - Hyper Speed Kit and RAISE3D Pro3 Plus - Hyper Speed Kit
  - Raise3D Hyper Core PPA CF25 1.75mm
  - Raise3D Hyper Core PPA GF25 1.75mm
  - Raise3D Hyper Core ABS CF15 1.75mm

- New support materials

- Raise3D Industrial PPA Support 1.75mm
- Raise3D Industrial PET Support 1.75mm
- Raise3D Premium PVA+ 1.75mm

Slicing Template Updates

1. DLP

- In DF2 Standard White, a new speed-optimized template has been added. The template selection list for Standard White material is sorted as Light/Heavy/Speed, with Light selected by default.
- For the three slicing templates in DF2 Standard White resin, the UV curing time control list and the thermal curing time control list have been uniformly adjusted to 30 minutes.

2. FFF

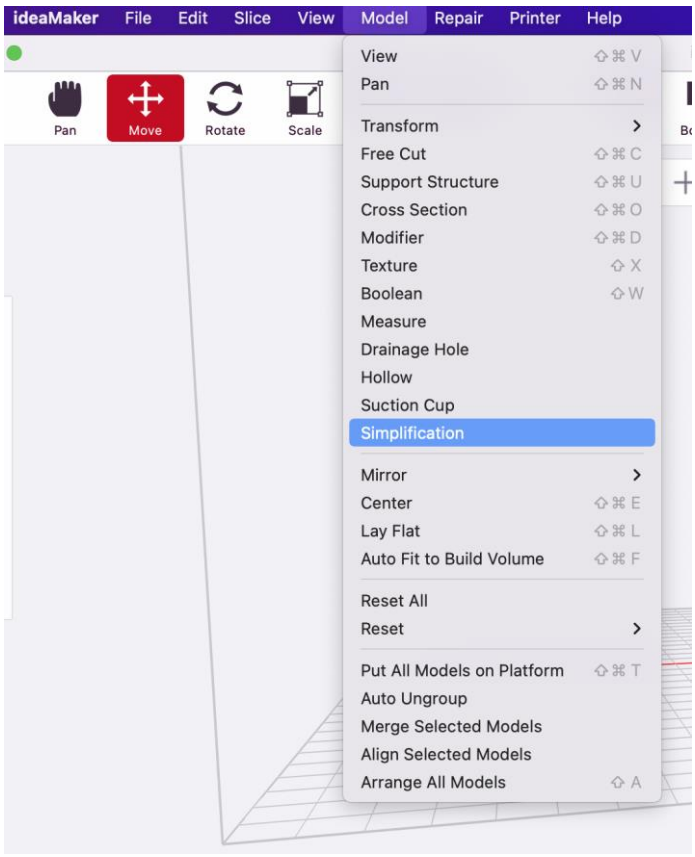
- **Updated the dual-color estimation optimization parameters in the slicing templates for Pro3 High Speed materials.**
  - ASA, PC, PETG
  - Hyper Speed ABS, Hyper Speed ABS V2
  - Hyper Core PPA CF25, PPA GF25、ABS CF15

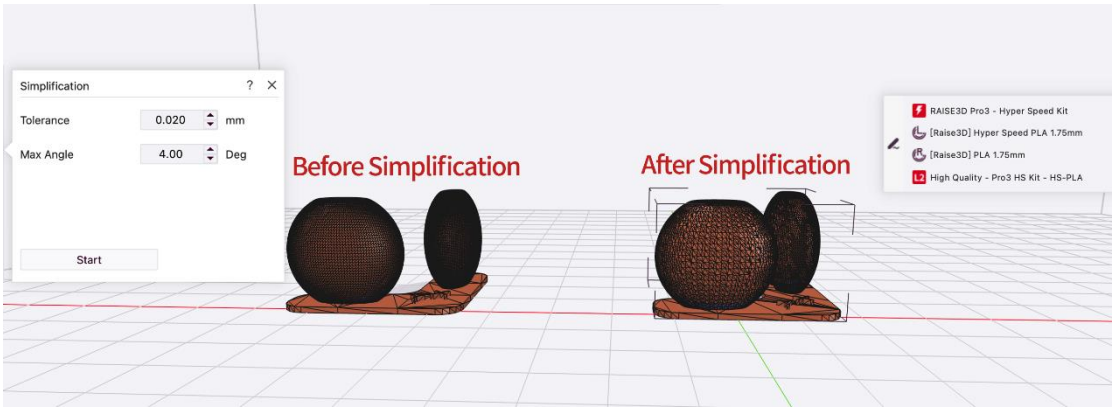
Other Features and Improvements

1. Support for Importing STEP/IGES Files.

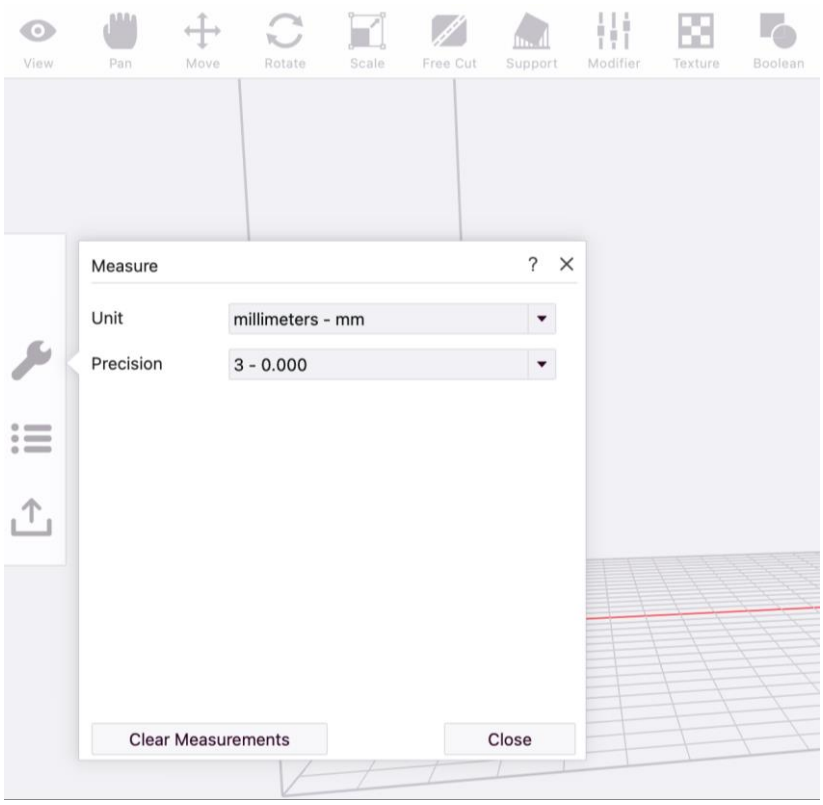
2. Added Simplification function.

- Model simplification feature to reduce the number of triangles in the model.





3. Added "Measure" Function



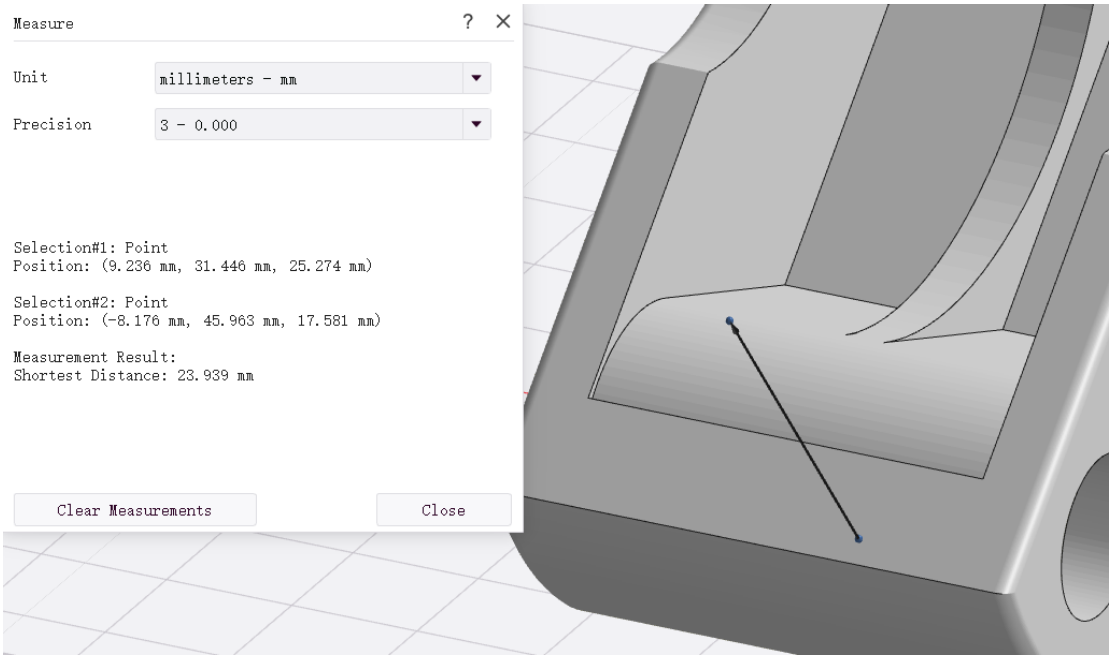
Supported Measurement Features

	Point	Line	Circle	Arc	Plane
Point	Yes	Yes	Yes	Yes	Yes
Line	Yes	Yes	Yes	Yes	Yes
Circle	Yes	Yes	Yes	No	Yes
Arc	Yes	Yes	No	No	Yes
Plane	Yes	Yes	Yes	Yes	Yes

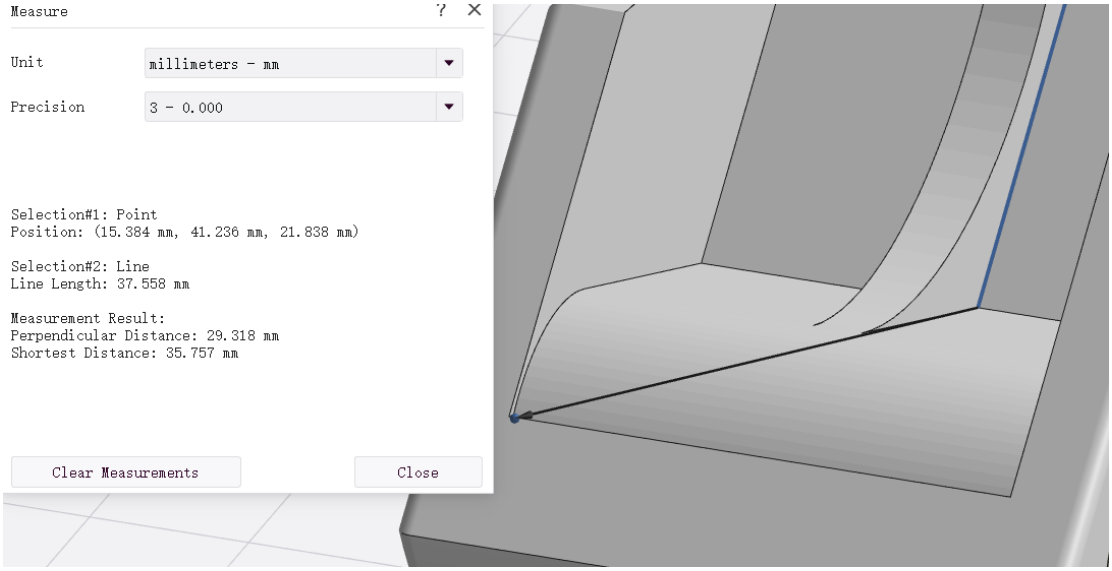
Example:

- Point - Point

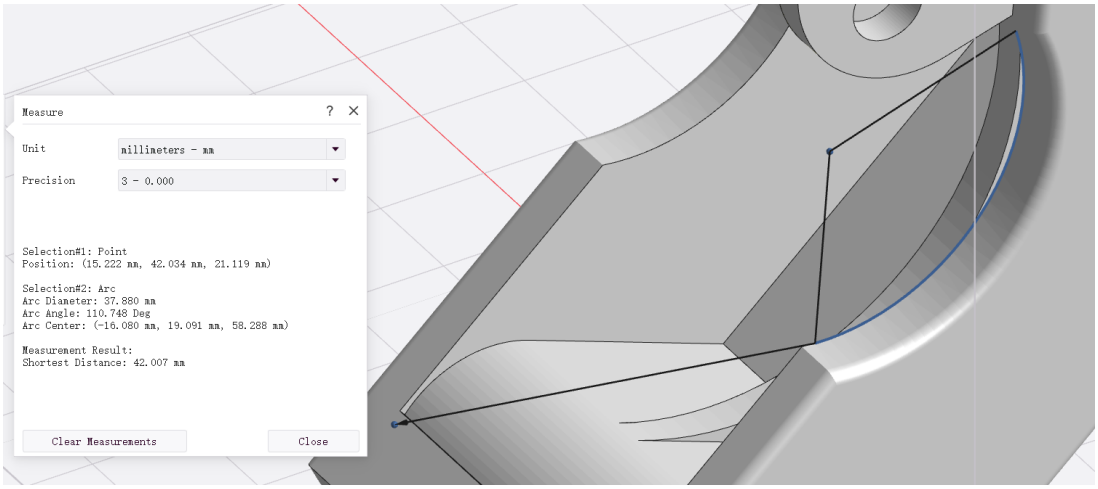
- Shortest distance from point to point



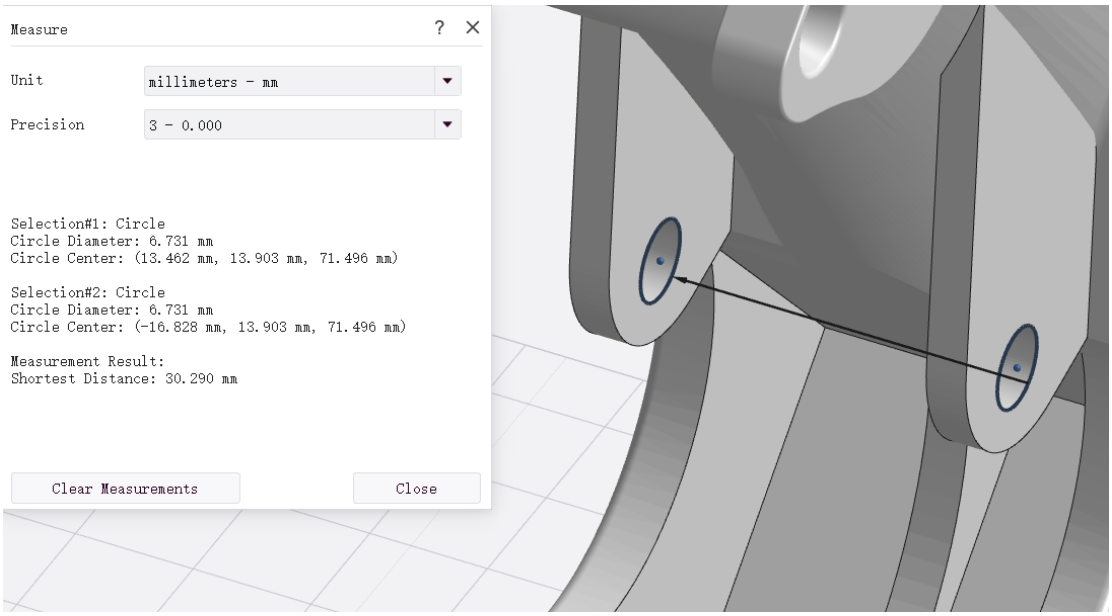
- Point – Line
  - Shortest distance from point to line, perpendicular distance from point to line (only the shortest distance indicator is displayed in the figure).



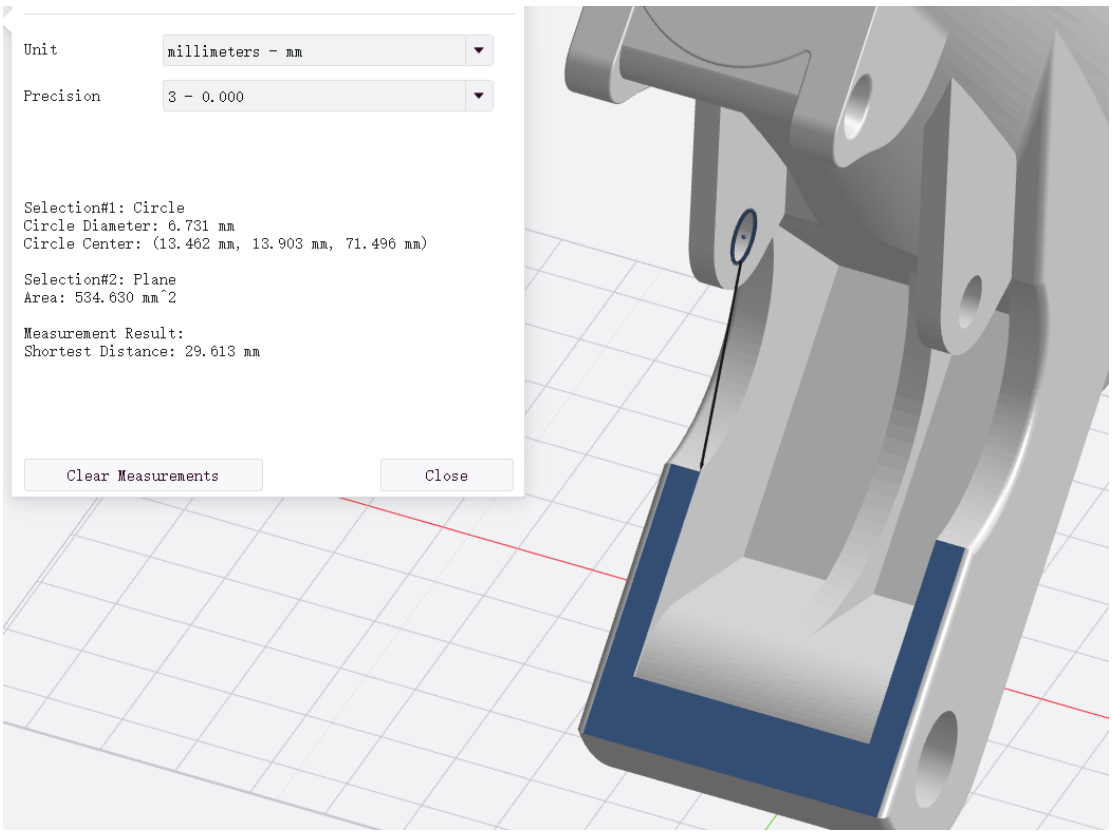
- Point – Arc
  - Shortest distance from the point to the arc.



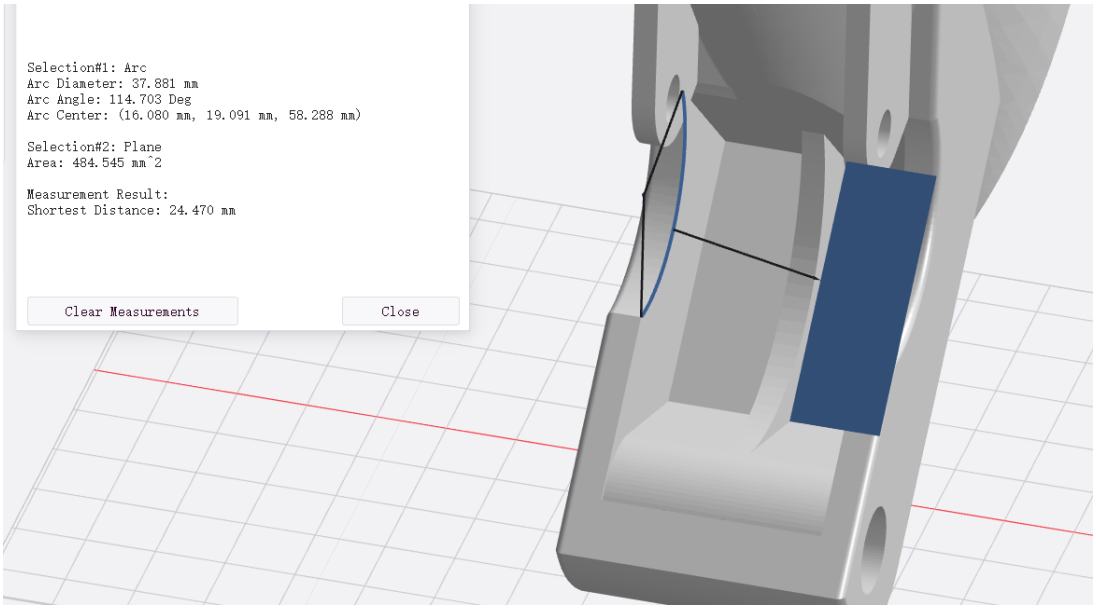
- Circle – Circle
  - Shortest distance between circles.



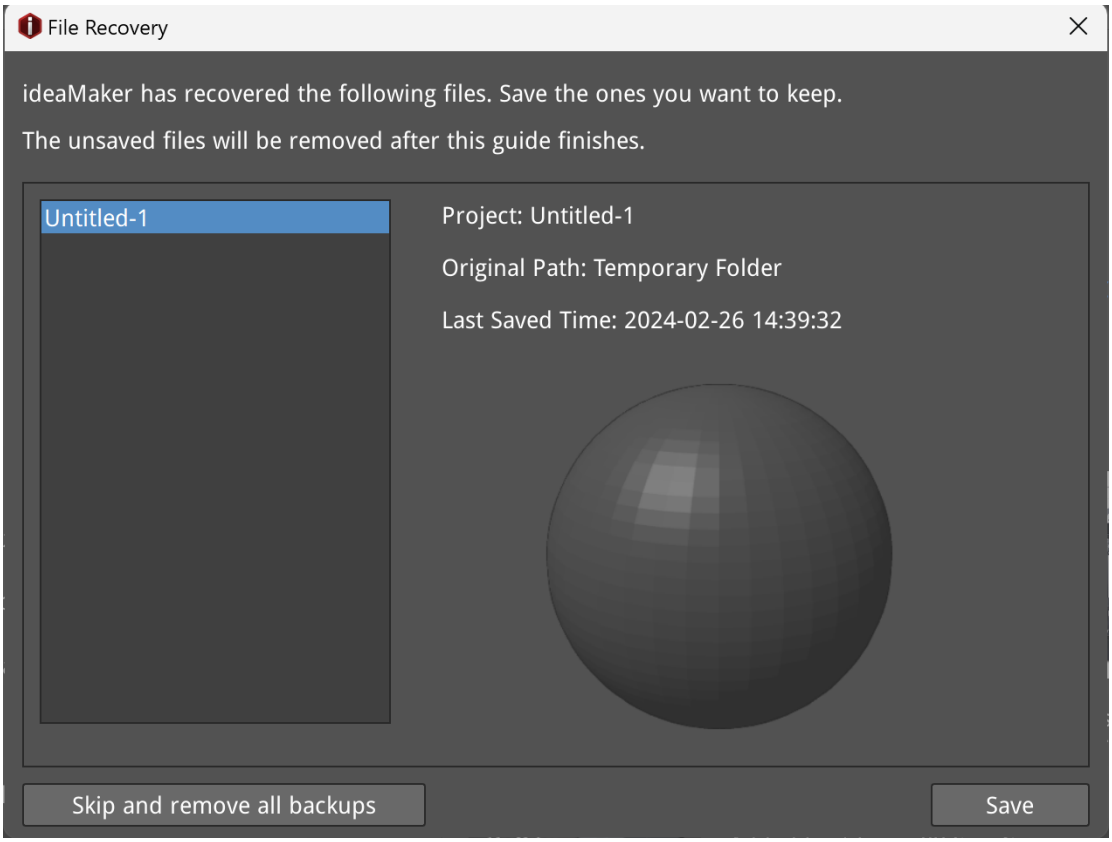
- Circle – Plane
  - Shortest distance from a circle to a plane.



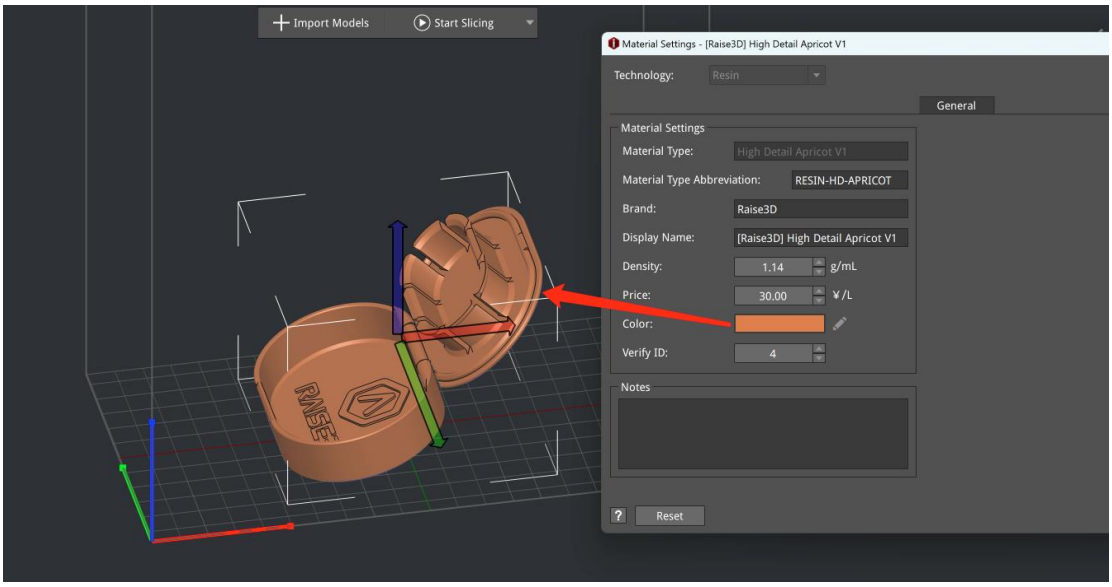
- Arc – Plane
  - Shortest distance from an arc to a plane.



4. Automatic Backup and Recovery Support for Software Crashes



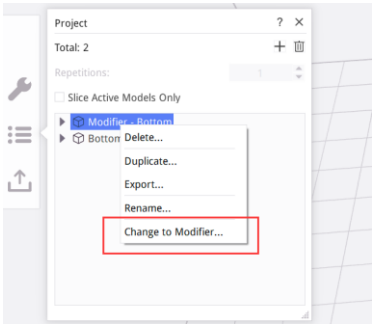
5. Material Settings Now Include a Material Color Attribute.

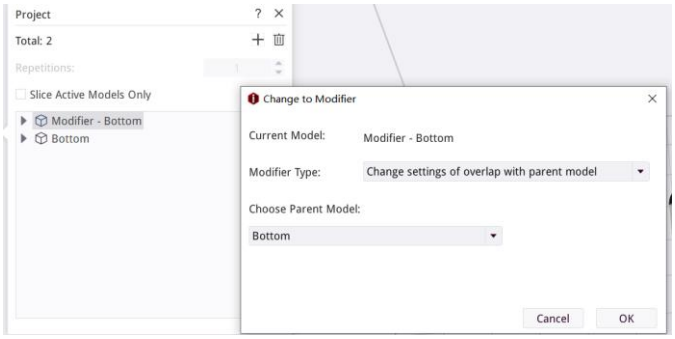


6. User-defined Slicing Templates Support Rollback of Slicing Settings.

7. Manually Select Models Already Placed on The Build Platform as Modifiers.

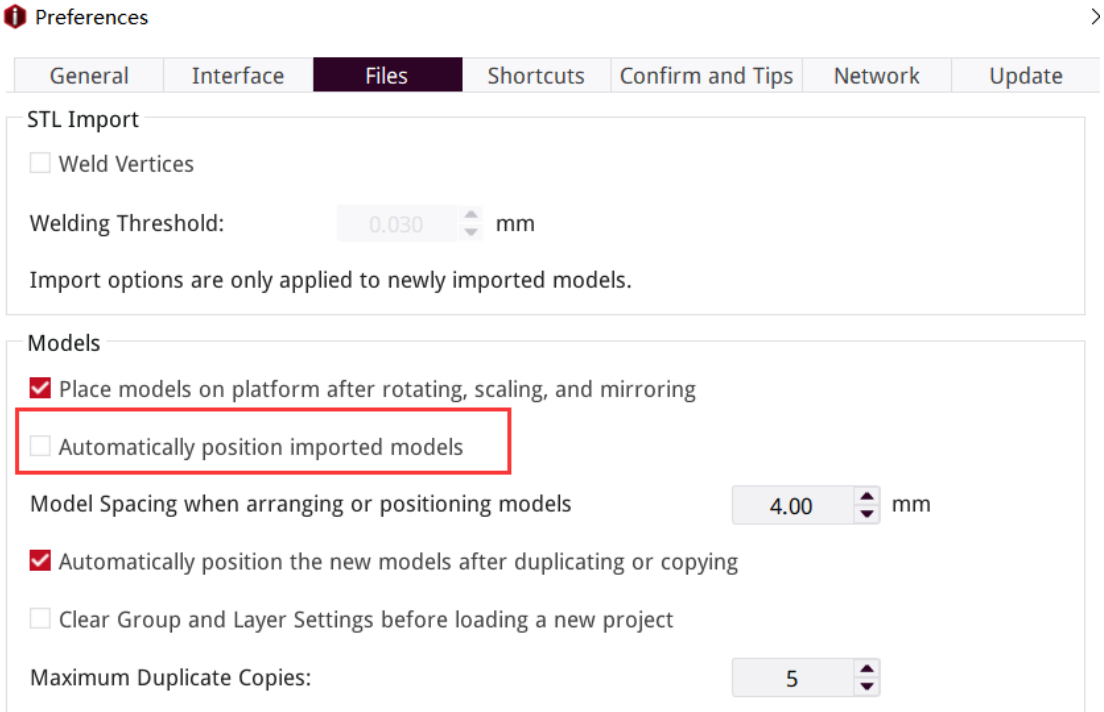
- Left panel of model list -> Select the model to be converted to a modifier -> Right-click menu -> Change to Modifier.



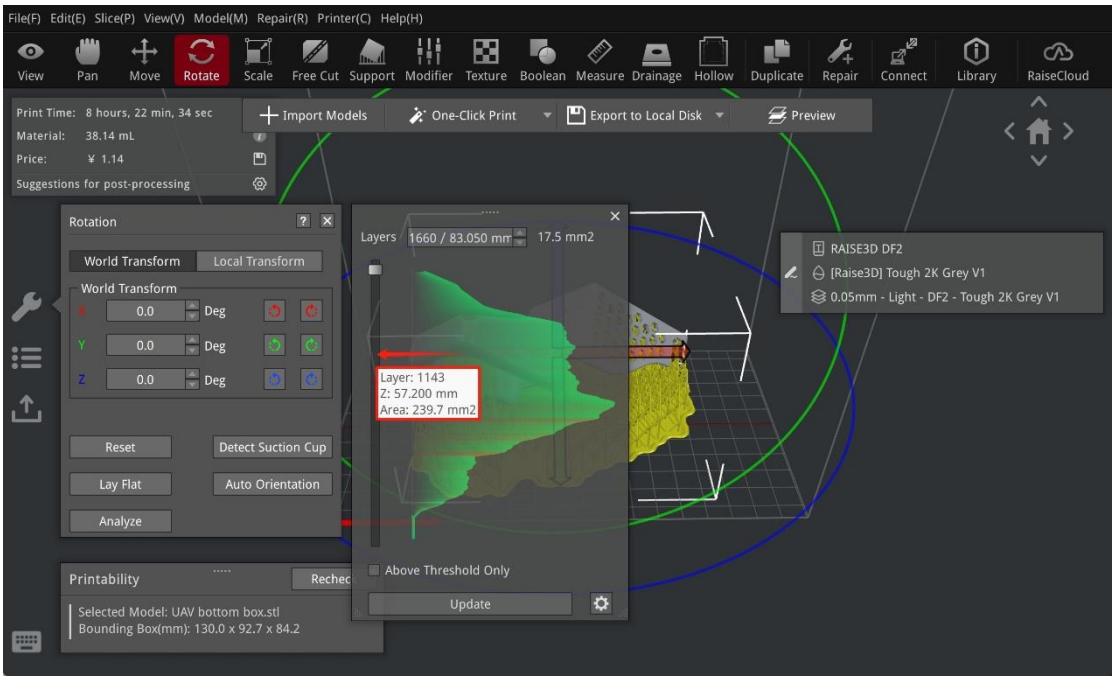


8. Imported 3MF file models maintain their absolute positions without offsetting overall.

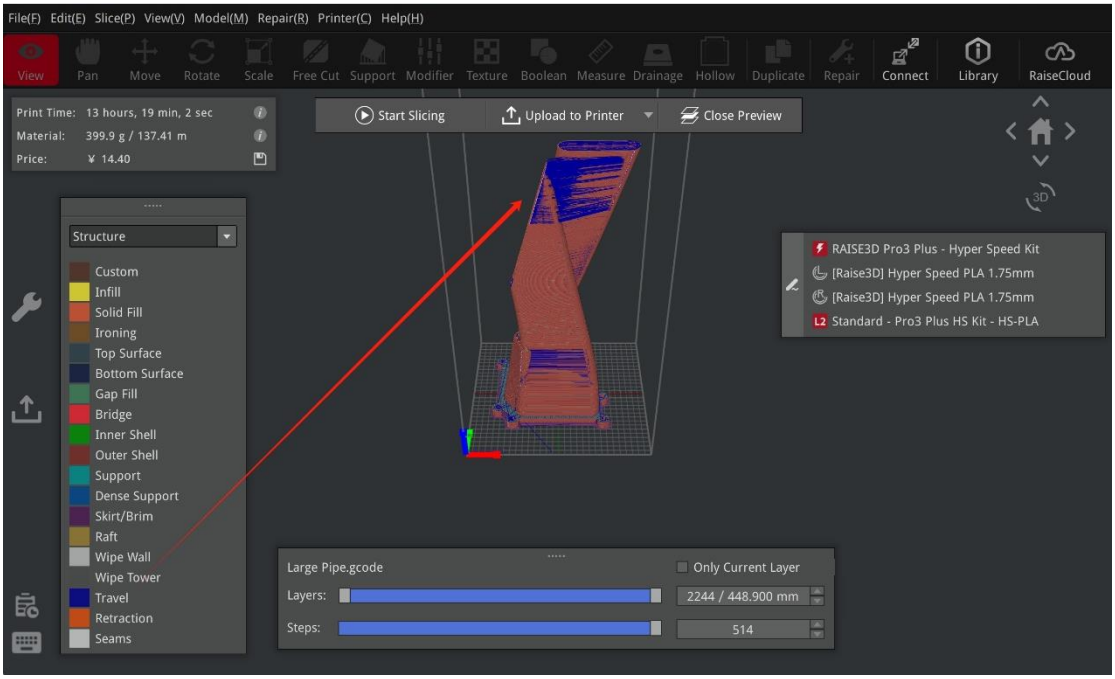
- After disabling the option in Preferences -> Files -> Automatically position imported models, imported 3MF files are not globally offset and remain in their original positions as designed in the modeling software.



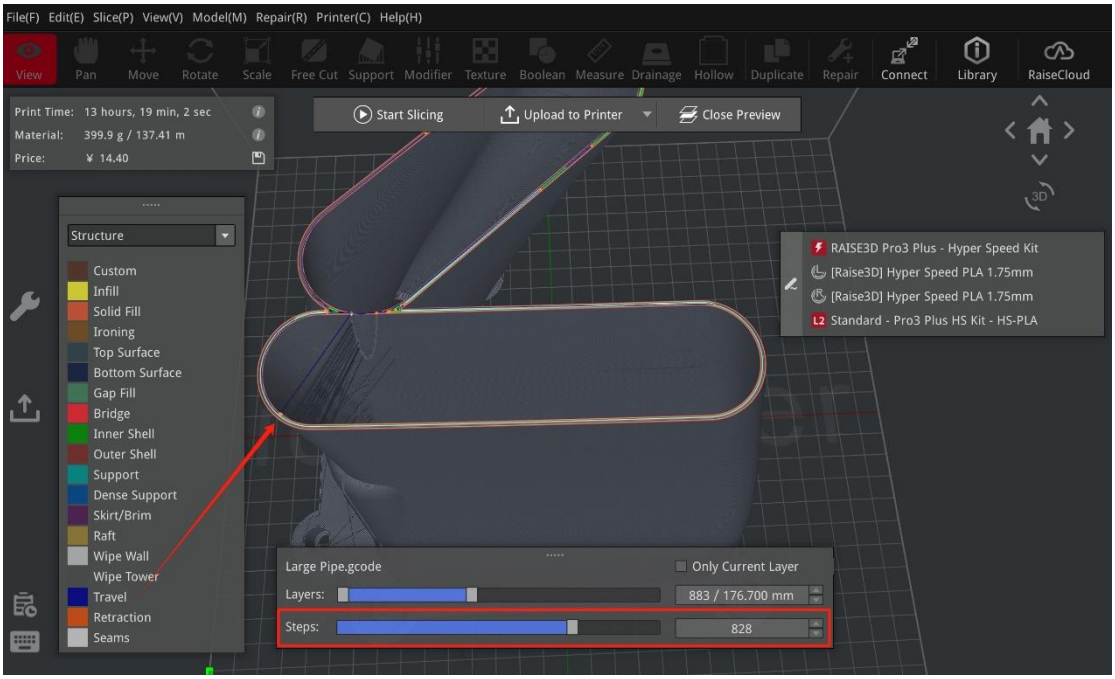
9. Optimization of cross-sectional area scrollbar: When hovering the cursor over any position on the scrollbar, it displays the corresponding layer, Z-axis height, and cross-sectional area at that position. Clicking on the cursor position directly jumps the scrollbar to the indicated location.



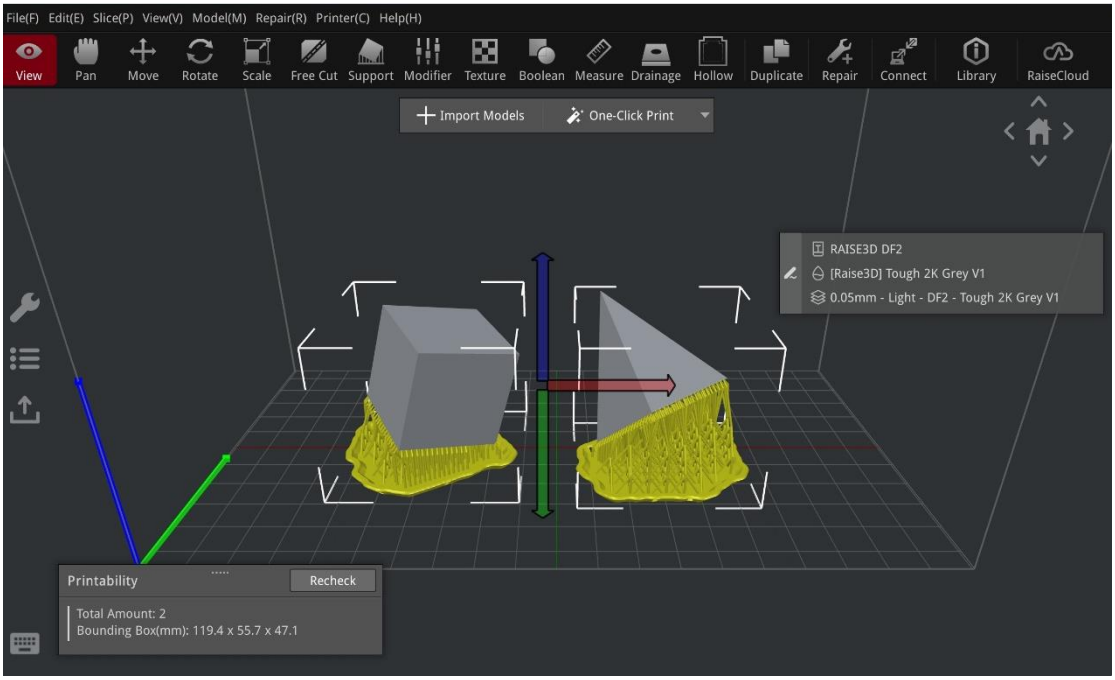
10. Shows travel on all layers under GCode preview.



11. When viewing the current layer steps, the travel for other layers turns dark gray, while only the travel for the current layer is displayed in blue.

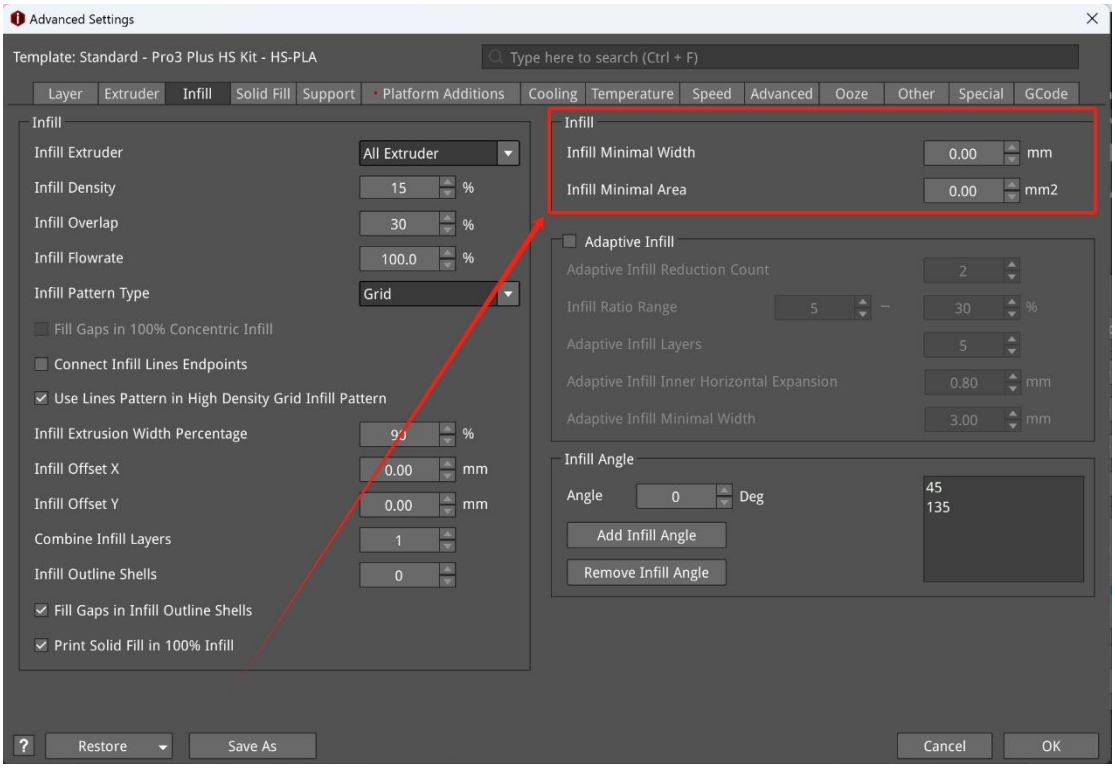


12. The bounding box lines become bold when the model is selected, distinguishing them from the edges of the model itself.



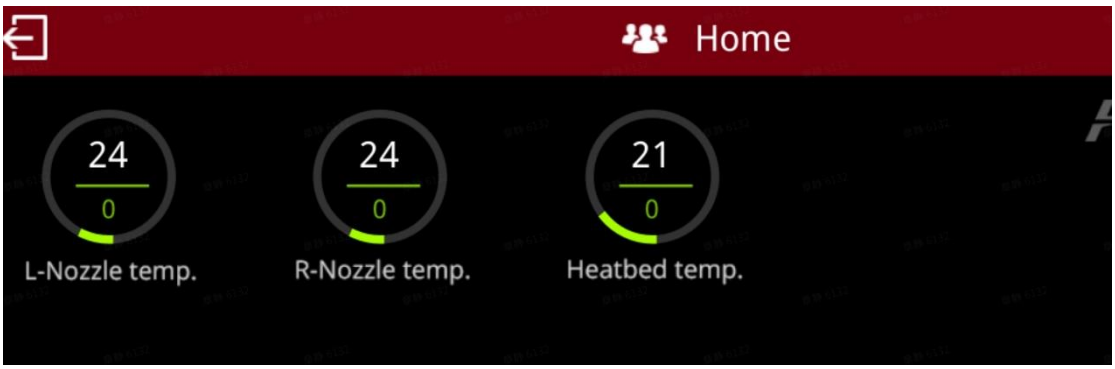
13. In the DLP slicing settings advanced dialog, the Start Gcode control has been expanded to display more area.

14. In the advanced settings dialog for slicing templates, the two options under the Infill Tab have been moved to a new location.



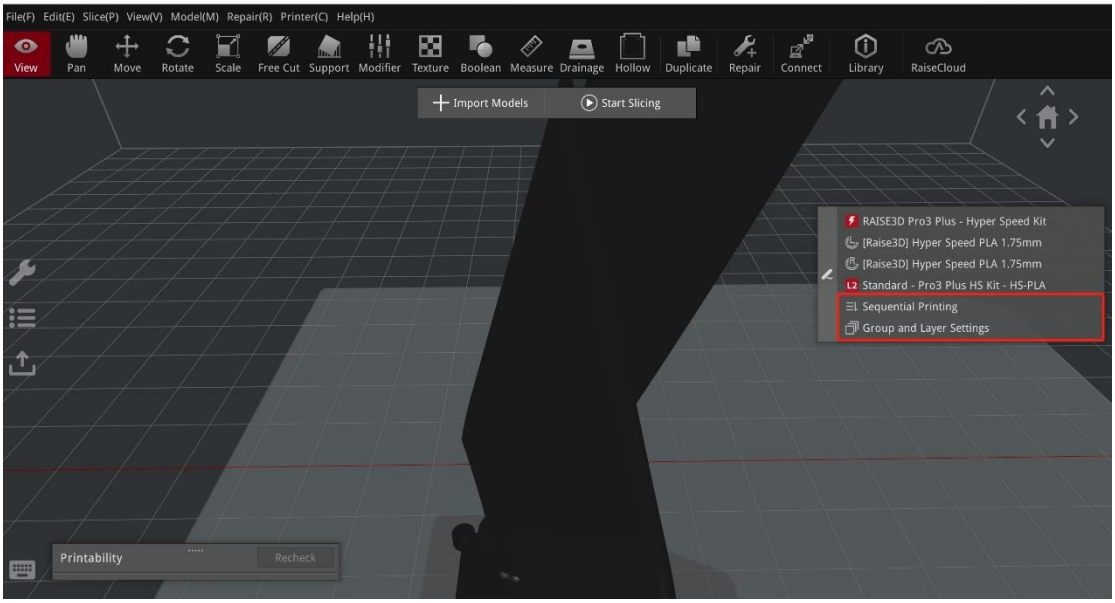
15. The Hollow and Drainage functions no longer check whether the current printer is a resin-based (DLP) printer. Regardless of whether the selected printer is FFF or DLP, both functions can be used. (Note: During FFF slicing, the Lattice Infill structure within the Hollow function will not be processed. Therefore, Lattice Infill will not appear in the GCode results after FFF slicing.)

16. In RMF500 remote control, the Home Tab provides access to set the build plate temperature.



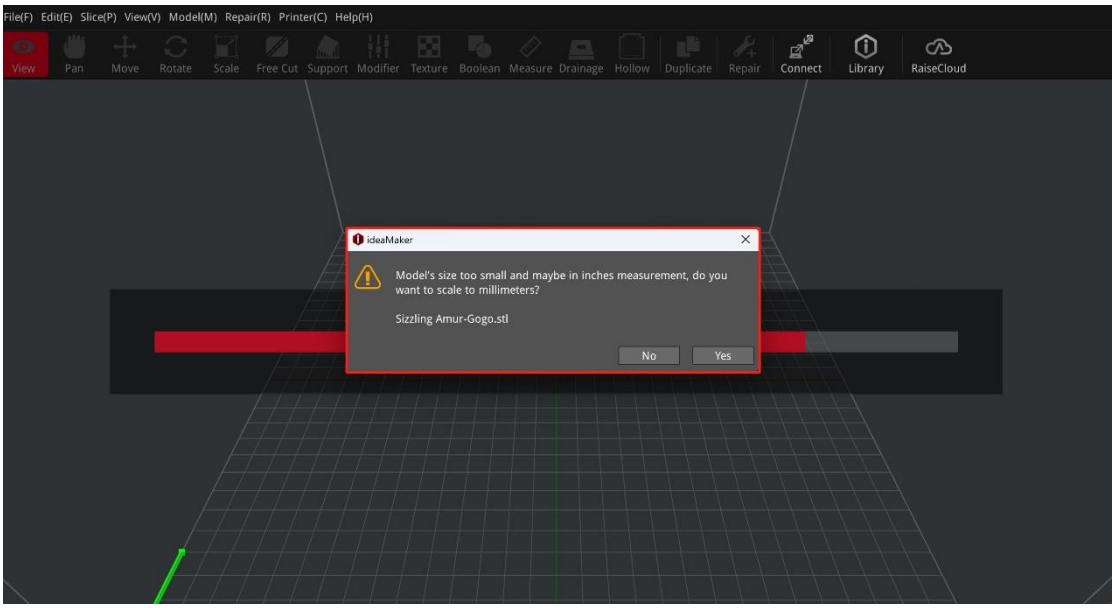
17. Prepare Slice Simplified Panel adds status display for sequential printing and group and layer

settings.

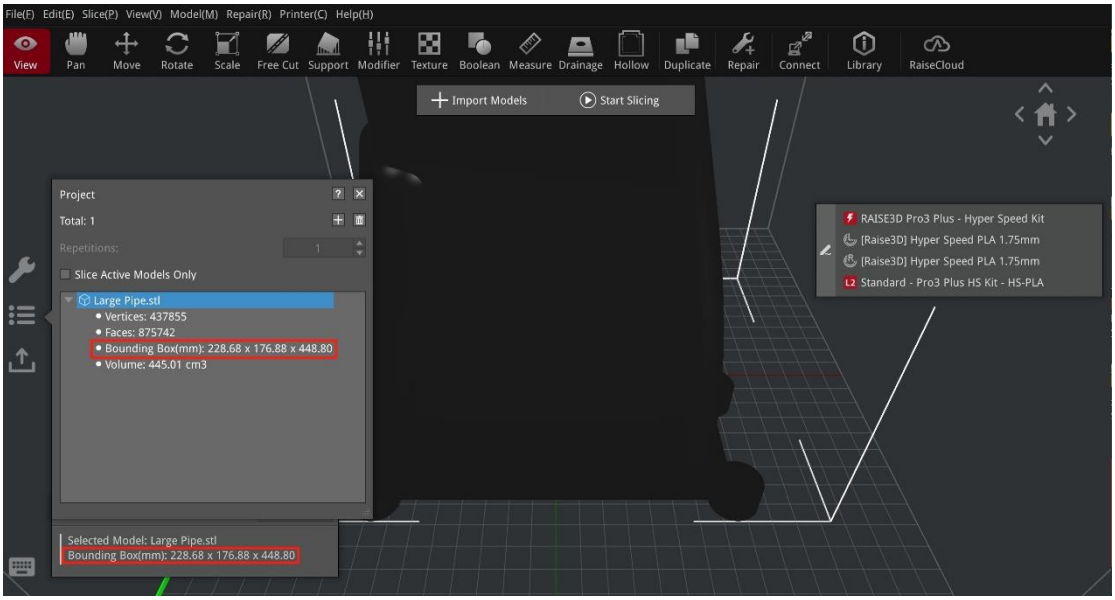


18. Scaling Issue with Importing Small-Sized Models:

- If the dimensions of the imported model are equal to or less than 0.1mm, the user is prompted to confirm whether the model was created using "meters" as the unit, and scale it accordingly using "millimeters."

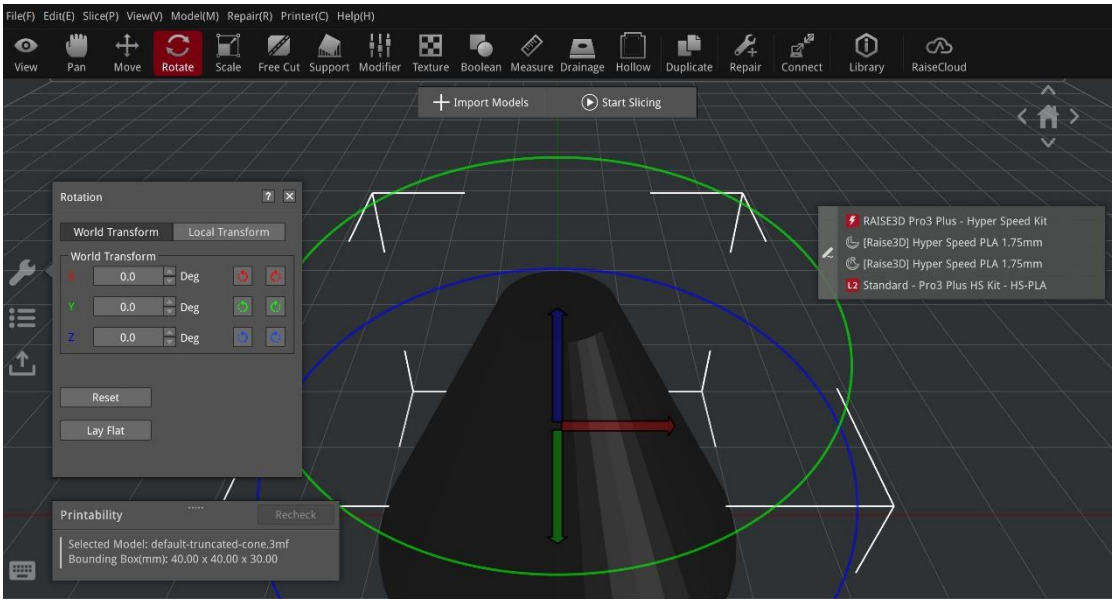


- Displays the bounding box dimensions of models in the model list and printable surface panel with two decimal places.



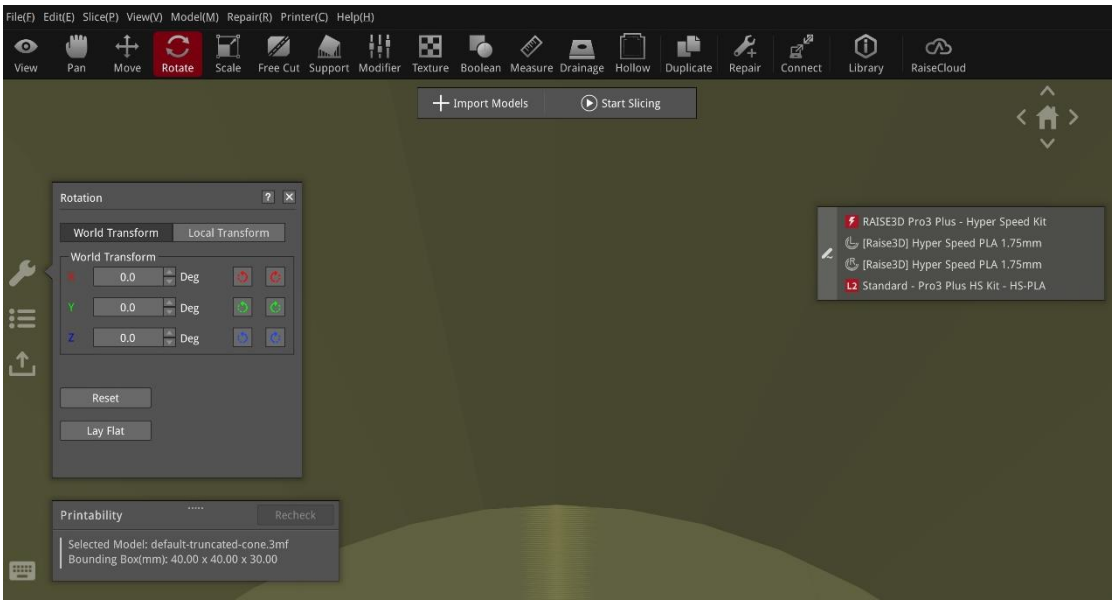
19. Adjustments to Rotation Controls:

- Regardless of whether the main interface is enlarged or reduced, the following controls should maintain their size on the interface to facilitate selection by the cursor:
  - Rotation controls
  - Rotate plane controls in the cutting function



- When the viewing angle is close to the model, the following will no longer be displayed:
  - Rotation controls

- Rotate plane controls in the cutting function



20. After slicing a model printed with the right nozzle, the new model will inherit the nozzle settings from the original model.