

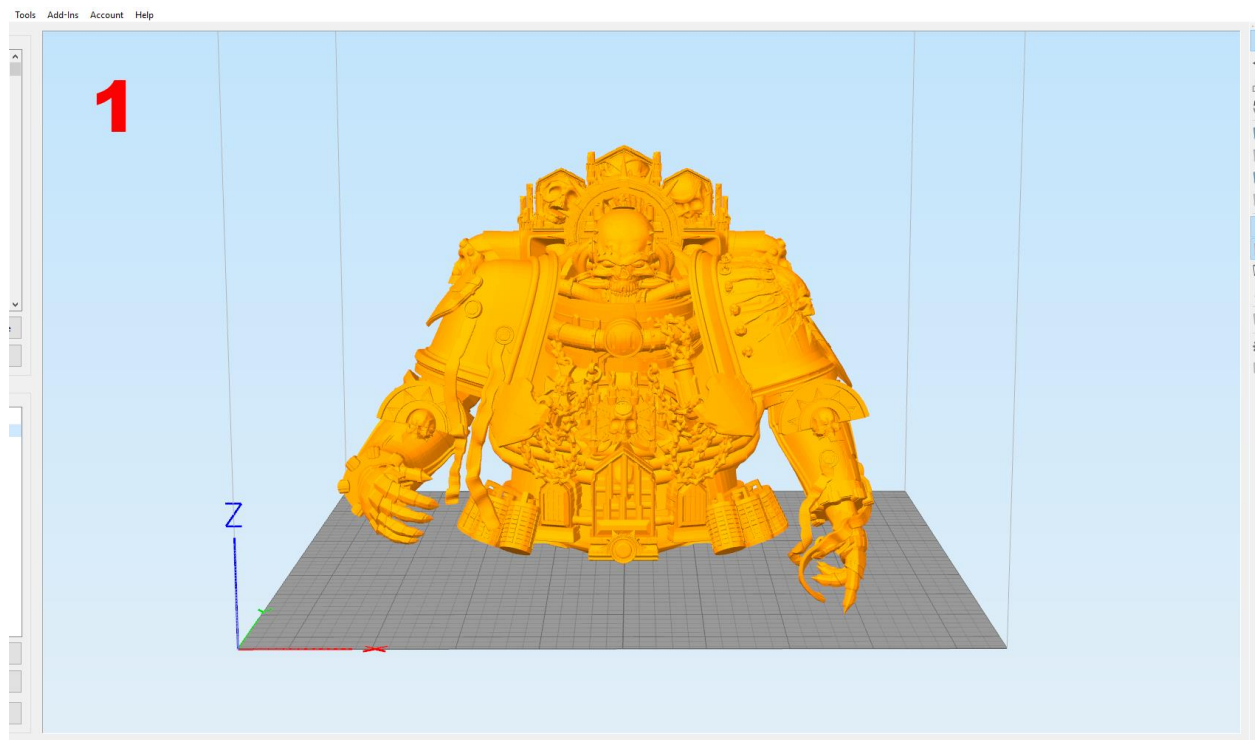
Simplify3D User Guidelines

These guidelines will help you configure correctly your Simplify3D slicing program. We will walk you through all the steps.

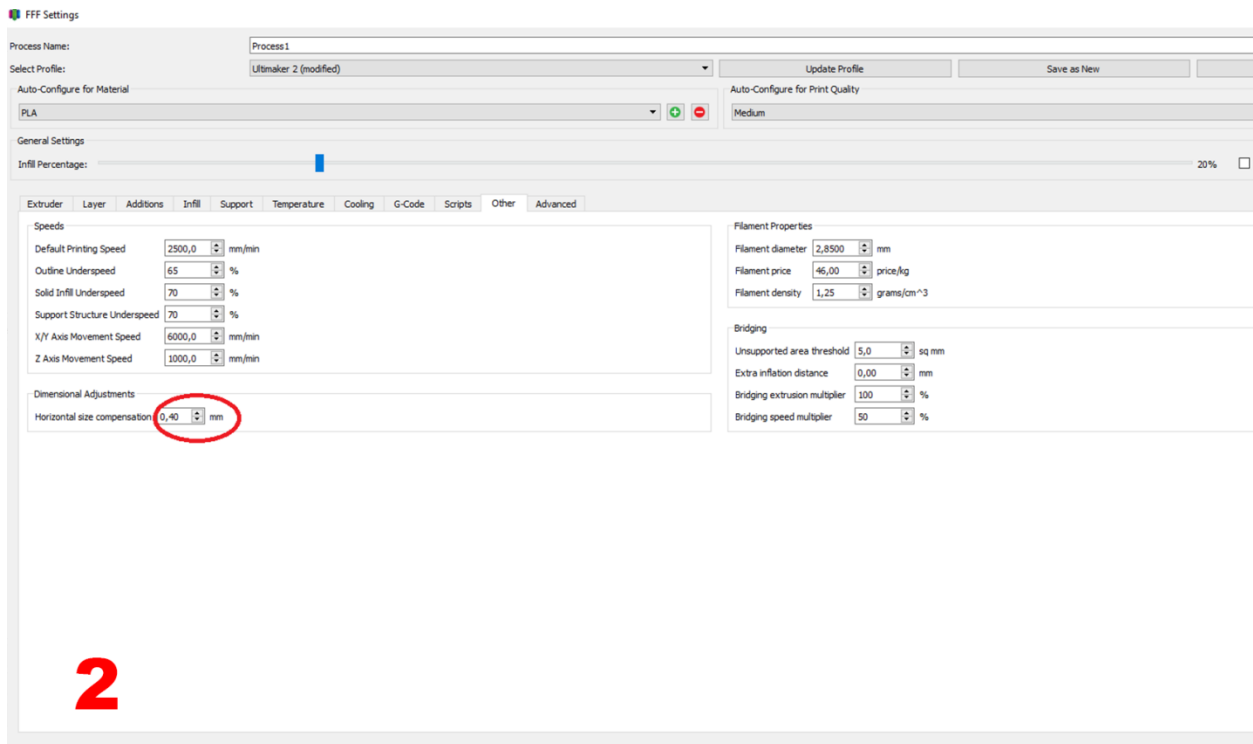
We have recently started receiving a lot of messages and emails concerning the printing of a 3D model using Simplify3D software.

These guidelines will assist you in configuring your slicing program correctly and will ensure an enjoyable 3D printing experience.

Step 1. Upload or drag your 3D model into the slicing program.



Step 2. Set *Horizontal size compensation* at **0.40mm**.



The screenshot shows the 'FFF Settings' window with the 'Advanced' tab selected. The 'Horizontal size compensation' field under 'Dimensional Adjustments' is set to '0,40 mm' and is circled in red. A large red number '2' is overlaid on the left side of the window.

Process Name: Process1
Select Profile: Ultimaker 2 (modified)
Auto-Configure for Material: PLA
Auto-Configure for Print Quality: Medium
General Settings: Infill Percentage: 20%
Extruder | Layer | Additions | Infill | Support | Temperature | Cooling | G-Code | Scripts | Other | Advanced

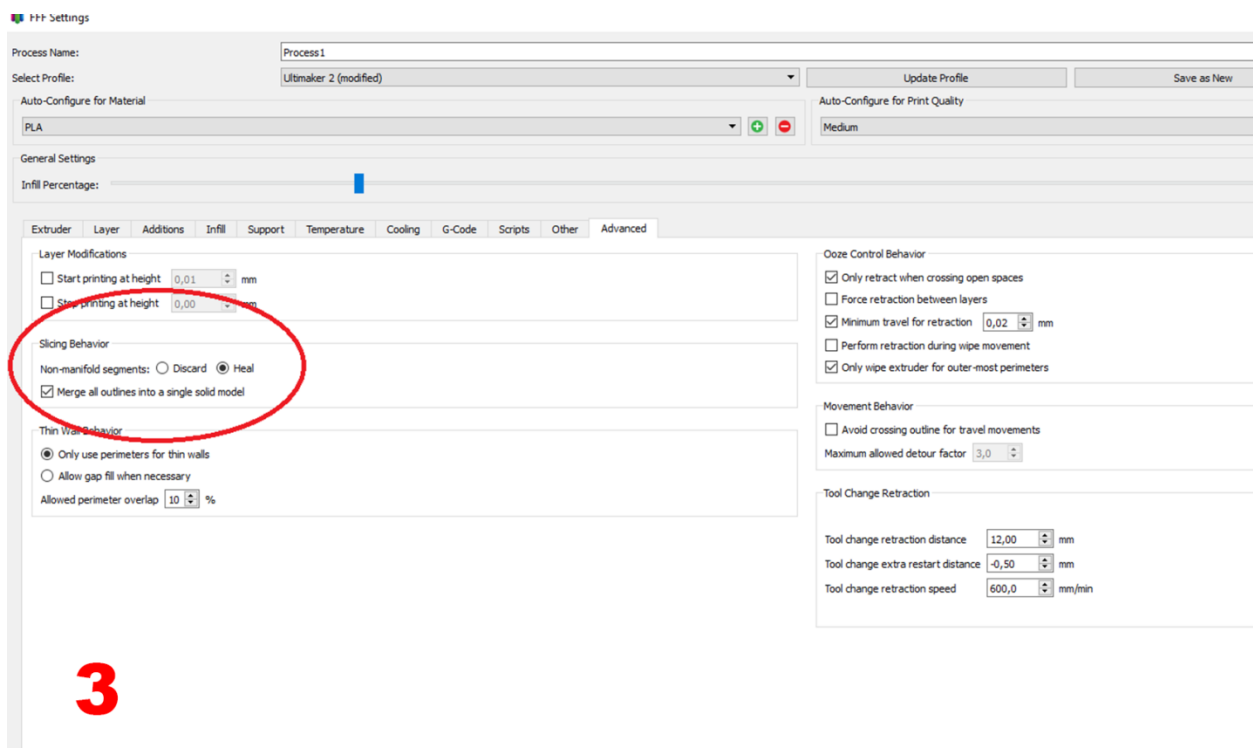
Speeds
Default Printing Speed: 2500,0 mm/min
Outline Underspeed: 65 %
Solid Infill Underspeed: 70 %
Support Structure Underspeed: 70 %
X/Y Axis Movement Speed: 6000,0 mm/min
Z Axis Movement Speed: 1000,0 mm/min

Dimensional Adjustments
Horizontal size compensation: 0,40 mm

Filament Properties
Filament diameter: 2,850 mm
Filament price: 46,00 price/kg
Filament density: 1,25 grams/cm³

Bridging
Unsupported area threshold: 5,0 sq mm
Extra inflation distance: 0,00 mm
Bridging extrusion multiplier: 100 %
Bridging speed multiplier: 50 %

* **Step 3.** Click *Edit Process Settings*, choose *Advanced*, and put the check mark in the box next to *Merge all outlines into a single solid model*. Press *OK*.



The screenshot shows the 'FFF Settings' window with the 'Advanced' tab selected. The 'Merge all outlines into a single solid model' checkbox under 'Slicing Behavior' is checked and circled in red. A large red number '3' is overlaid on the left side of the window.

Process Name: Process1
Select Profile: Ultimaker 2 (modified)
Auto-Configure for Material: PLA
Auto-Configure for Print Quality: Medium
General Settings: Infill Percentage: 20%
Extruder | Layer | Additions | Infill | Support | Temperature | Cooling | G-Code | Scripts | Other | Advanced

Layer Modifications
☐ Start printing at height: 0,01 mm
☐ Stop printing at height: 0,00 mm

Slicing Behavior
Non-manifold segments: ☐ Discard ☒ Heal
☒ Merge all outlines into a single solid model

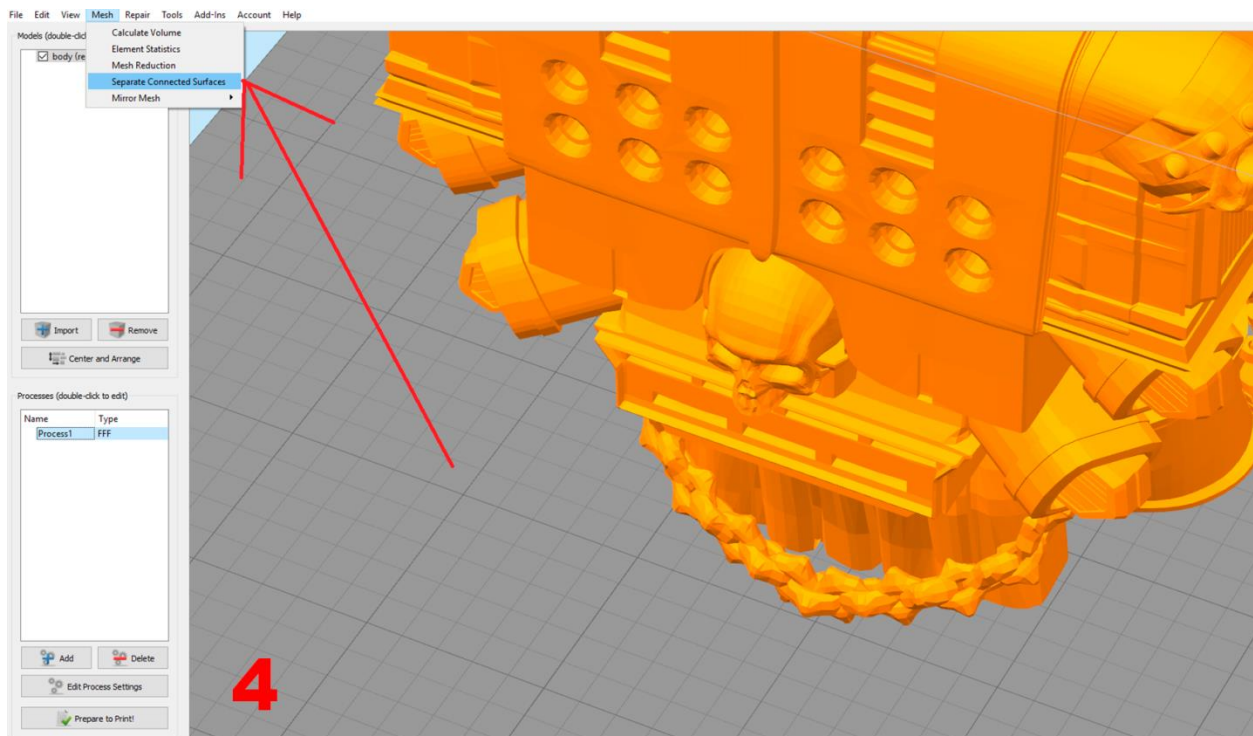
Thin Wall Behavior
☒ Only use perimeters for thin walls
☐ Allow gap fill when necessary
Allowed perimeter overlap: 10 %

Ooze Control Behavior
☒ Only retract when crossing open spaces
☐ Force retraction between layers
☒ Minimum travel for retraction: 0,02 mm
☐ Perform retraction during wipe movement
☒ Only wipe extruder for outer-most perimeters

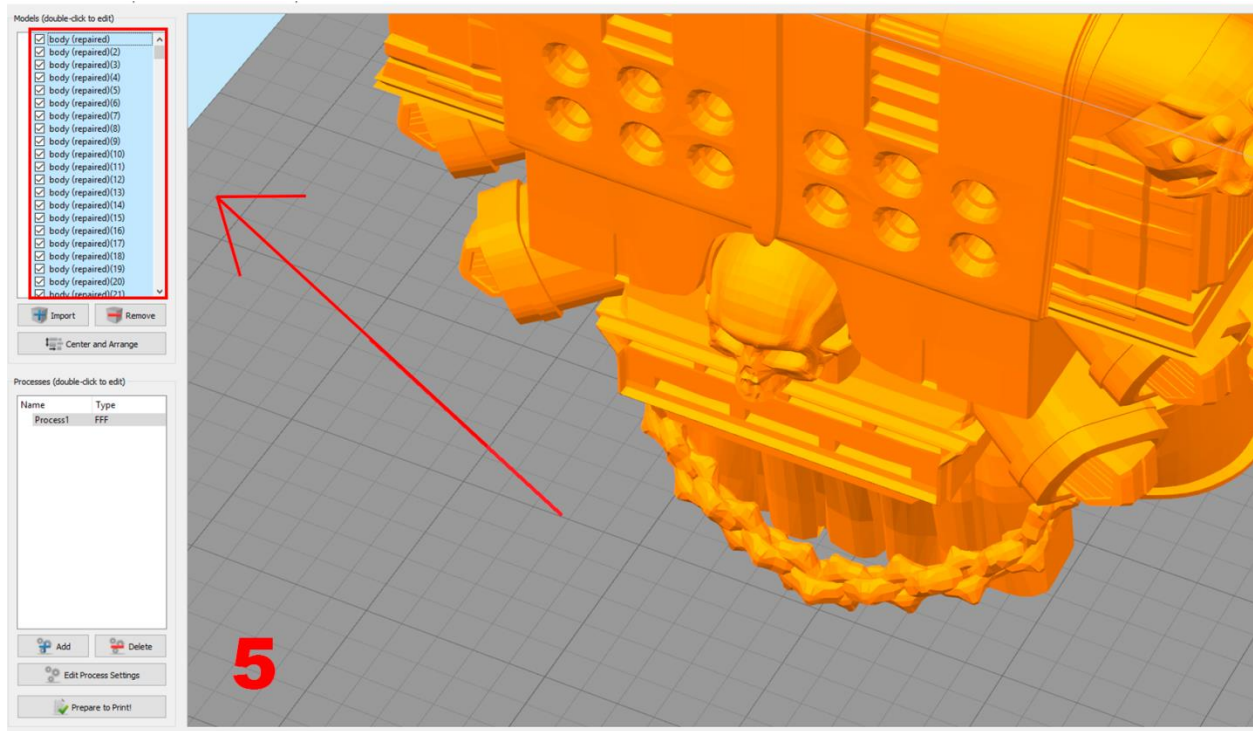
Movement Behavior
☐ Avoid crossing outline for travel movements
Maximum allowed detour factor: 3,0

Tool Change Retraction
Tool change retraction distance: 12,00 mm
Tool change extra restart distance: -0,50 mm
Tool change retraction speed: 600,0 mm/min

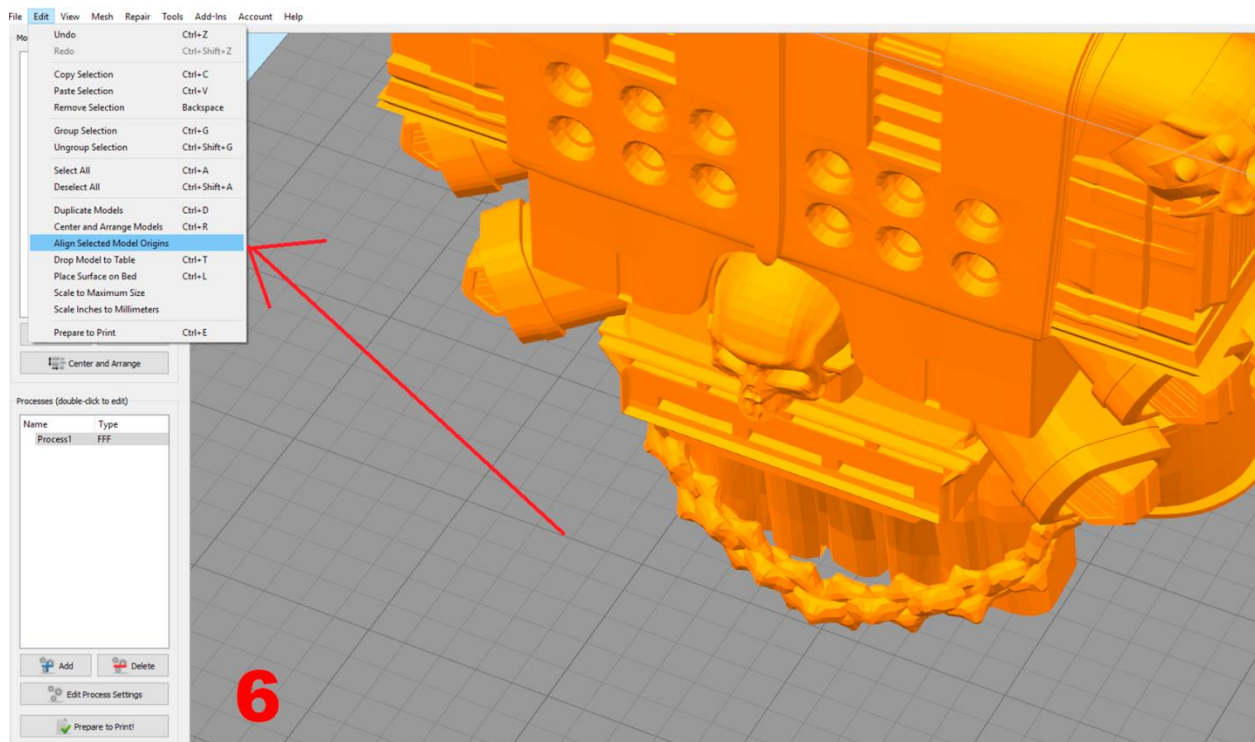
Step 4. Select *Mesh > Separate Connected Surfaces*



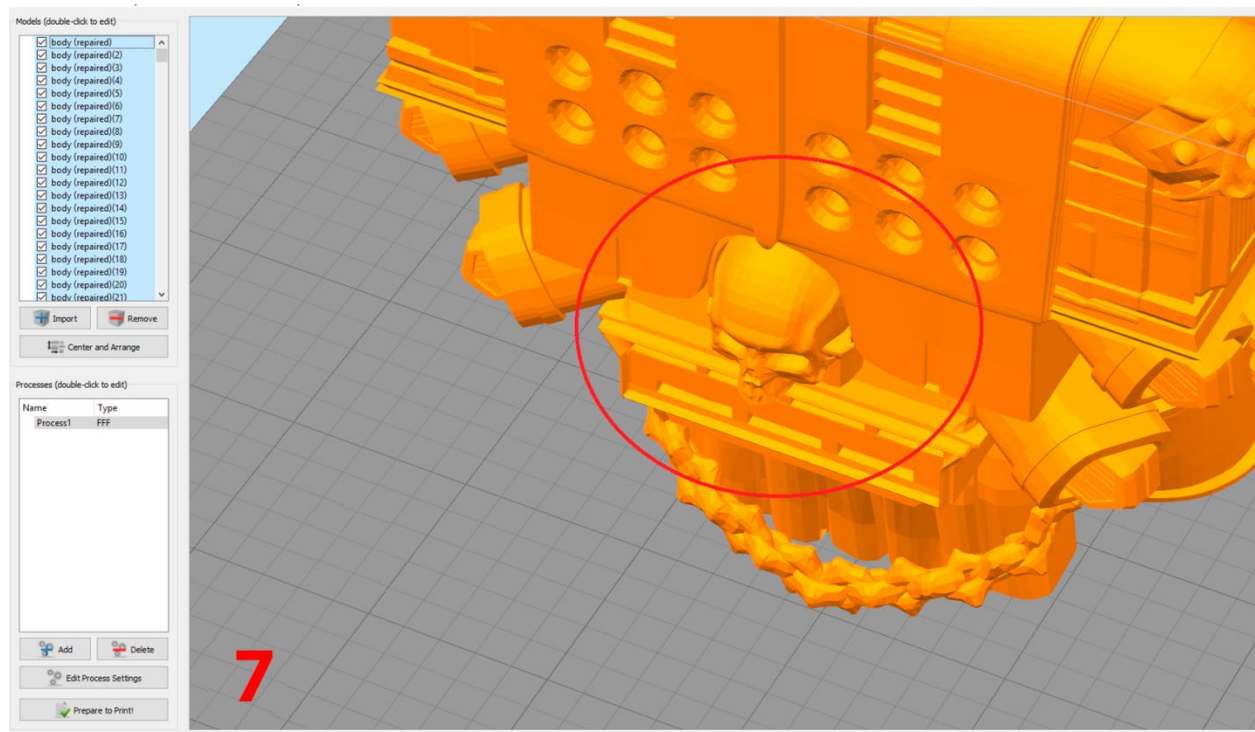
Step 5. Select all edits from the first through the last.



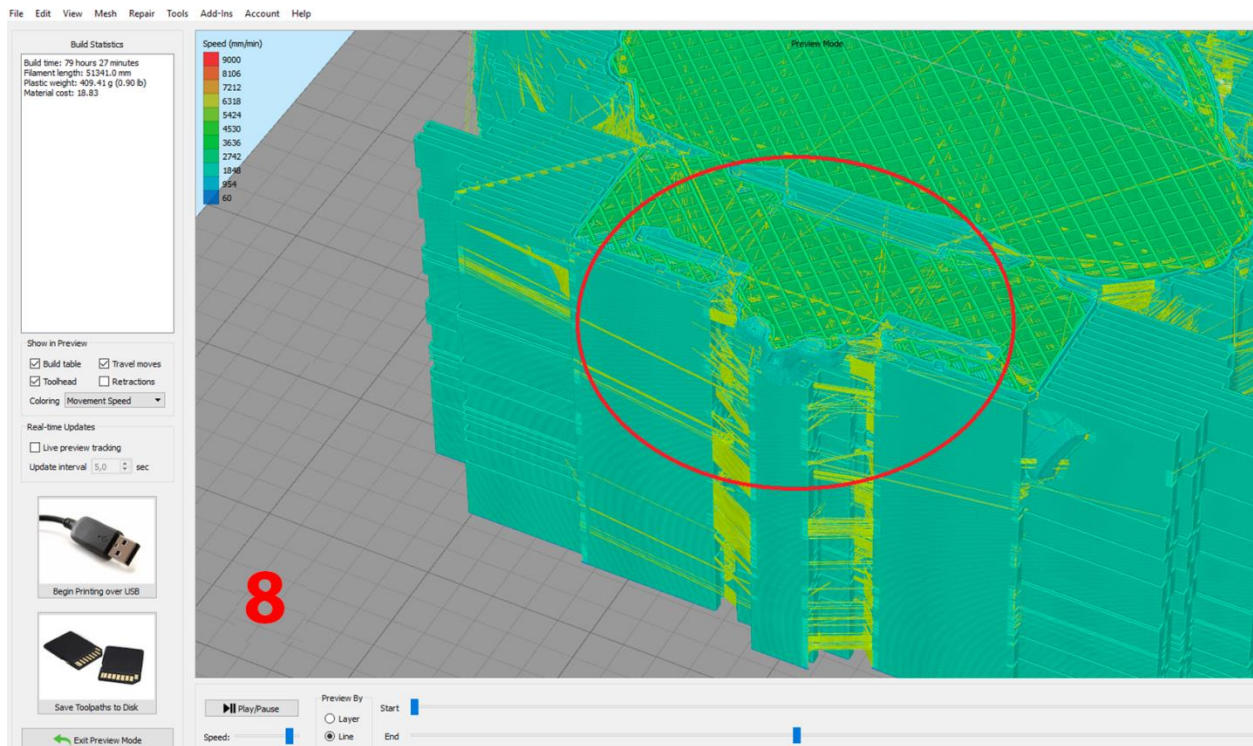
Step 6. Go to *Edit > Align Selected Model Origins*



Step 7. Result



Step 8. Result



* For some 3D printing models, avoid putting the check mark in the box next to the *Merge all outlines into a single solid model*. To know exactly for which 3D models the box must be checked or not, please test both with and without the check mark before sending the 3D model to the printer.