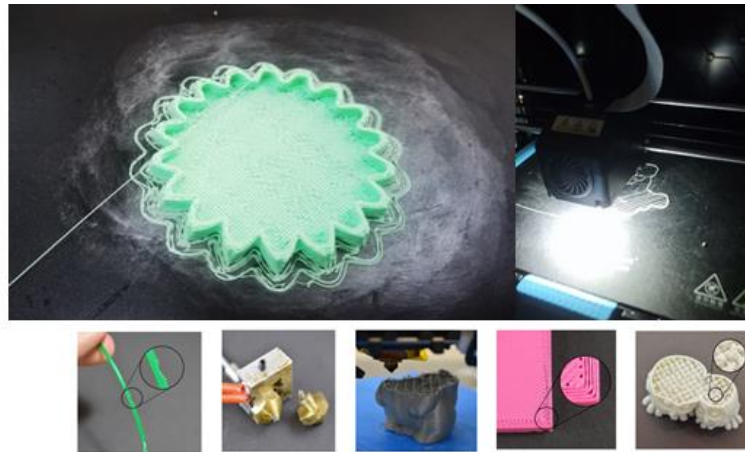


3D Printing Troubleshooting Guide

Nozzle clog & Poor extrusion output

Obvious issues (as below)



Suggested solutions:

1. When we met these situations, it is recommended that the printing temperature should raise to 230 °C, adjust printing speed to 40mm/s, reduced the layer thickness to 0.1mm, then test if it works.
2. If the first step can not solve the problem, need to exclude hardware problems: pull out the front end of the PTFE Pipe Hose to check if it is darken or not , if yes, need replace a new one with nozzle together, because the pipe is already aging.(When replacing, notice that the nozzle need to be tightened with heat)
3. Check the extruder gears, whether the bite is normal, it is recommended that in the case of normal feeding, try to loose the extruder spring

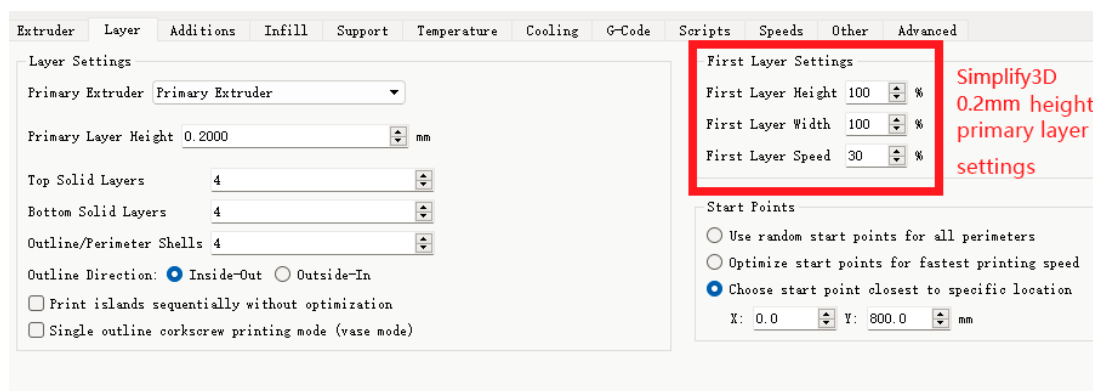
Non-stick platform , warped edges

Obvious issues (as below)



Suggested solutions:

1. It is recommended to re-level first, the nozzle is too close to the platform will cause the first layer easy to overflow, if too far the first layer can not stick to the platform; if the four corners leveling difference is too large, it may lead to warp from a farther corner
2. Adjust slicing parameters, the first layer extrusion rate is recommended to be set higher than 100%, the first layer speed set to 10mm/s (refer to the figure below), close the first layer blowing fan, the platform should set to 45-60°C, raft and brim should also adjust to improve the platform's adhesion.



Extruder Layer Additions Infill Support Temperature Cooling G-Code Scripts **Speeds** Other Advanced

Layer Settings

Primary Extruder Primary Extruder

Primary Layer Height 0.1000 mm

Top Solid Layers 4

Bottom Solid Layers 4

Outline/Perimeter Shells 4

Outline Direction: Inside-Out Outside-In

Print islands sequentially without optimization

Single outline corkscrew printing mode (vase mode)

First Layer Settings

First Layer Height 200 %

First Layer Width 100 %

First Layer Speed 30 %

**Simplify3D
0.1mm height
first layer settings**

Start Points

Use random start points for all perimeters

Optimize start points for fastest printing speed

Choose start point closest to specific location

X: 0.0 I: 800.0 mm

Generic PLA Coarse - 0.2mm 10% Off Off

Cura first layer speed settings

Print settings

Profile Coarse - 0.2mm

Speed

Print Speed	40.0	mm/s
Infill Speed	40.0	mm/s
Wall Speed	24.0	mm/s
Outer Wall Speed	20.0	mm/s
Inner Wall Speed	24.0	mm/s
Top/Bottom Speed	28.0	mm/s
Travel Speed	150.0	mm/s
Initial Layer Speed	10.0	mm/s
Initial Layer Print Speed	10.0	mm/s
Initial Layer Travel Speed	75.0	mm/s
Skirt/Brim Speed	10.0	mm/s
Z Hop Speed	10.0	mm/s
Number of Slower Layers	2	

< Recommended

Generic PLA Coarse - 0.2mm 10% Off Off

Cura 0.2mm height first layer flow settings

Print settings

Profile Coarse - 0.2mm

Infill

Material

Printing Temperature	200.0	°C
Printing Temperature Initial Layer	200.0	°C
Initial Printing Temperature	190.0	°C
Final Printing Temperature	185.0	°C
Build Plate Temperature	60	°C
Build Plate Temperature Initial Layer	60	°C
Flow	100.0	%
Wall Flow	100.0	%
Outer Wall Flow	100.0	%
Inner Wall(s) Flow	100.0	%
Top/Bottom Flow	100.0	%
Infill Flow	100.0	%
Skirt/Brim Flow	100.0	%
Prime Tower Flow	100.0	%
Initial Layer Flow	100.0	%

Speed

3. If these measures can not solve the problem, it is recommended that the platform coated with PVP solid glue or 3DLac Spray to increase material adhesion, some lattice base plate may not stick, it can be replaced with pure glass base plate, the adhesion will be better.
4. Winter temperature is too low, PLA-Type materials may also be warped, you can properly open the air conditioner or use eSUN eEnclosure to print or properly reduce the speed of the blowing fan, such as adjusting to 50%

Stringing



Suggested solutions:

1. Setting correct retraction distance and retraction compensate, retraction distance is too high, easy to cause material lack; too low, easy to stringing; Path optimization, it is recommended to open the start point alignment function when slicing, close retraction vertical lift, avoid going through the shell when empty range movement, optimize the slicing printing path, and appropriately reduce the printing speed to achieve the best printing effect

Extruder Layer Additions Infill Support Temperature Cooling G-Code Scripts Speeds Other Advanced

Extruder List
(click item to edit settings)

Primary Extruder

Primary Extruder Toolhead

Overview

Extruder Toolhead Index: Tool 0

Nozzle Diameter: 0.40 mm

Extrusion Multiplier: 1.03

Extrusion Width: Auto Manual 0.48 mm

Ooze Control

Retraction

Retraction Distance: 6.00 mm

Extra Restart Distance: 0.00 mm

Retraction Vertical Lift: 0.00 mm

Retraction Speed: 25.0 mm/s

Coast at End

Coasting Distance: 1.00 mm

Wipe Nozzle

Wipe Distance: 2.00 mm

Add Extruder

Remove Extruder

Simplify3D
close retraction vertical lift

Extruder Layer Additions Infill Support Temperature Cooling G-Code Scripts Speeds Other Advanced

Layer Settings

Primary Extruder: Primary Extruder

Primary Layer Height: 0.1000 mm

Top Solid Layers: 4

Bottom Solid Layers: 4

Outline/Perimeter Shells: 4

Outline Direction: Inside-Out Outside-In

Print islands sequentially without optimization

Single outline corkscrew printing mode (vase mode)

First Layer Settings

First Layer Height: 200 %

First Layer Width: 100 %

First Layer Speed: 30 %

Start Points

Use random start points for all perimeters

Optimize start points for fastest printing speed

Choose start point closest to specific location

X: 0.0 mm Y: 800.0 mm

Simplify3D start point settings

Generic PLA

Fine - 0.1mm 15% Off Off

Cura
Z seam Alignment settings

Print settings

Profile: Fine - 0.1mm

Search settings

Walls

Wall Thickness: 1.05 mm

Wall Line Count: 3

Outer Wall Wipe Distance: 0.2 mm

Outer Wall Inset: 0.025 mm

Optimize Wall Printing Order:

Outer Before Inner Walls:

Alternate Extra Wall:

Compensate Wall Overlaps:

Compensate Outer Wall Overlaps:

Compensate Inner Wall Overlaps:

Fill Gaps Between Walls: Everywhere

Do Not Fill Out Tiny Gaps:

Print Thin Walls:

Horizontal Expansion: 0.0 mm

Initial Layer Horizontal Expansion: 0.0 mm

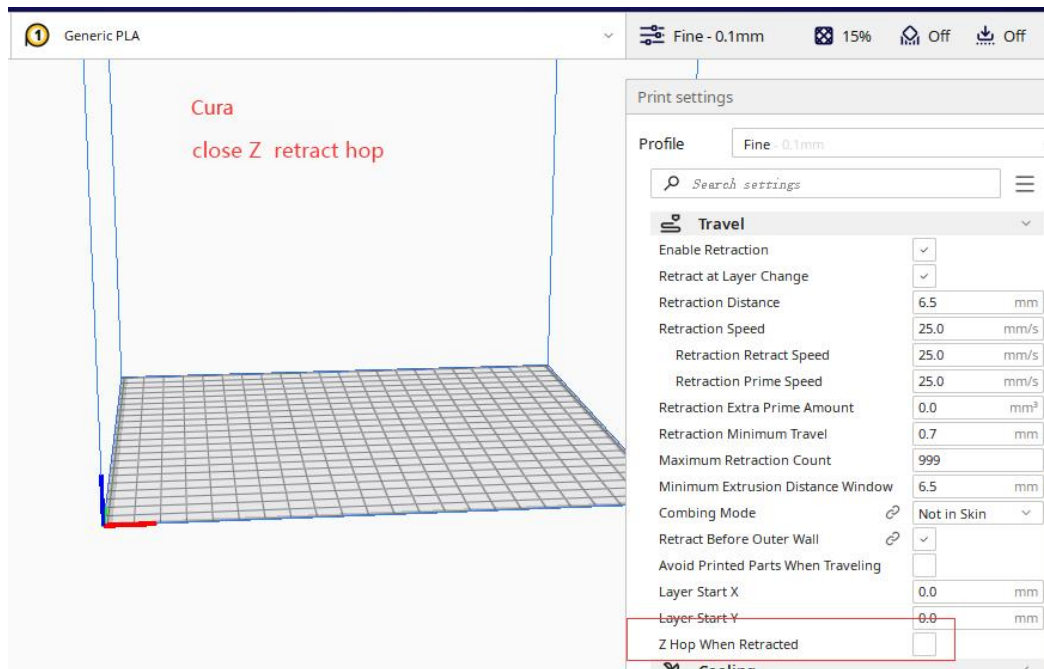
Hole Horizontal Expansion: 0.0 mm

Z Seam Alignment: Sharpest Corn...

Seam Corner Preference: Hide Seam

Top/Bottom

< Recommended



2. Higher the printing temperature is , the more likely to stringing, it is recommended that in the case of normal extrusion, reduce the printing temperature to slove sringing issues.