

SETTINGS FOR CURA

The following are images of the Cura profiles developed for printing using Colorfabb LW-PLA and regular PLA. We found that depending on the version of Cura, the profiles may or may not work. So we have created this document to aid in users being able to print with any slicer.

The images indicate the critical settings required to achieve the same standard of print as the test model.

Regular PLA

REGULAR PLA - Fine - 0.25mm 0% Off Off

Print settings

Profile: REGULAR PLA - Fine - 0.25mm

Search settings

Quality

Layer Height: 0.25 mm

Walls

Wall Thickness: 0.4 mm

Wall Line Count: 1

Print Thin Walls: ☒

Horizontal Expansion: 0.0 mm

Top/Bottom

Top/Bottom Thickness: 0.0 mm

Top Thickness: 0.0 mm

Top Layers: 0

Bottom Thickness: 3.0 mm

Bottom Layers: 12

Extra Skin Wall Count: 1

Infill

Infill Density: 0.0 %

Material

Printing Temperature: 205.0 °C

Build Plate Temperature: 45.0 °C

Flow: 100.0 %

Speed

REGULAR PLA - Fine - 0.25mm 0% Off Off

Print settings

Profile: REGULAR PLA - Fine - 0.25mm

Search settings

Enable Retraction: ☒

Retract at Layer Change: ☒

Retraction Distance: 8.0 mm

Retraction Speed: 100.0 mm/s

Retraction Extra Prime Amount: 0.1 mm³

Combing Mode: All

Layer Start X: 100.0 mm

Layer Start Y: 0.0 mm

Z Hop When Retracted: ☐

Cooling

Enable Print Cooling: ☒

Fan Speed: 100.0 %

Support

Generate Support: ☐

Build Plate Adhesion

Build Plate Adhesion Type: Skirt

Special Modes





Print Sequence: All at Once

Surface Mode: Normal

Experimental !!CRITICAL!!

Slicing Tolerance: Exclusive

Colorfabb LW-PLA


COLORFABB LW-PLA - Fine - 0.2mm

0%

Off

On

Print settings

Profile
COLORFABB LW-PLA - Fine - 0.2mm

Quality

Layer Height
0.2 mm

Walls

Wall Thickness
0.4 mm

Wall Line Count
1

Print Thin Walls
☒

Horizontal Expansion
0.0 mm

Top/Bottom

Top/Bottom Thickness
0.0 mm

Top Thickness
0.4 mm

Top Layers
0

Bottom Thickness
0.0 mm

Bottom Layers
0

Infill

Infill Density
0.0 %

Material





Printing Temperature
250.0 °C

Build Plate Temperature
60.0 °C

Flow
40.0 %

Speed

Travel


COLORFABB LW-PLA - Fine - 0.2mm

0%

Off

On

Print settings

Profile
COLORFABB LW-PLA - Fine - 0.2mm

Enable Retraction
☒

Retract at Layer Change
☒

Retraction Distance
10.0 mm

Retraction Speed
100.0 mm/s

Retraction Extra Prime Amount
0.45 mm³

Combing Mode
All

Layer Start X
100.0 mm

Layer Start Y
0.0 mm

Z Hop When Retracted
☐

Cooling

Enable Print Cooling
☒

Fan Speed
100.0 %

Support

Generate Support
☐

Build Plate Adhesion

Build Plate Adhesion Type
Brim

Special Modes

Print Sequence
All at Once

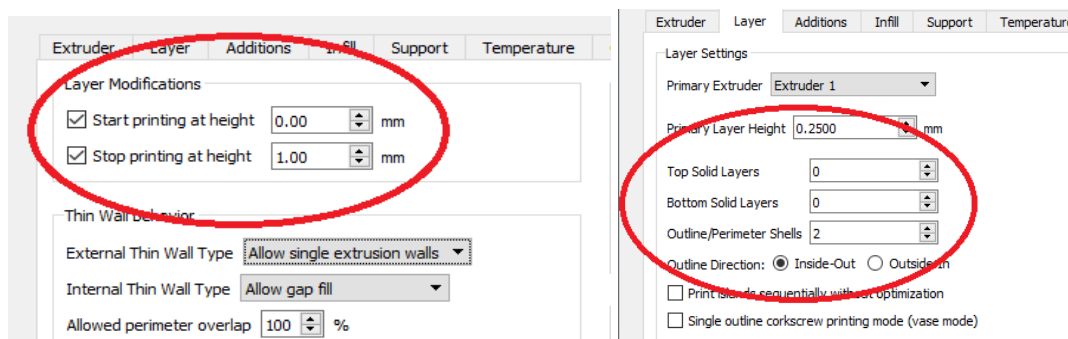
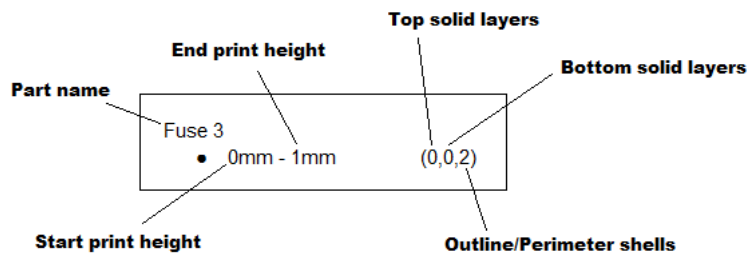
Surface Mode
Normal

Experimental

Slicing Tolerance
Exclusive

!!CRITICAL!!

In the “Print settings” document provided, the settings change at certain Z-heights throughout parts of the model. Generally the settings that will change are the number of TOP or BOTTOM layers within the region, or the WALL LINE COUNT.



Please follow the link for instructions on how to adjust the settings based on Z-height:

<https://www.youtube.com/watch?v=8e1QpC5oty8>

Please follow the link for instructions on how to support blockers (allows altered settings in selected locations):

<https://www.youtube.com/watch?v=GfcPGfsm2l0>