Cura 5.2.1 settings

By Gambody v 1.0

These are averaged settings which were tested in the Cura 5.2.1 slicer. Test models were printed on Anycubic Vyper, Creality Ender 3 Pro with PLA filament.

Disclaimer: The following printing settings are a recommendation, not an obligation. The parameters can vary depending on the peculiarities of your 3D printer, the material you use, and especially the particular assembly part you are working with. Each part that any model comprises often needs preliminary review, and you are free to tweak the settings the way you find suitable.

Note:
- You can scale up the model (downscaling is not recommended!);
- All connectors should be printed at 100% Infill;

To avoid printing problems, we recommend the following settings:

Quality

Layer Height*		0.12 - 0.14 mm
Initial Layer Height		0.2 mm
Line Width		0.4 mm
Wall Line Width:		0.4 mm
Outer Wall Line Width		0.4 mm
Inner Wall(s) Line Width		0.4 mm

Top/Bottom Line Width:		0.4 mm
Infill Line Width		0.4 mm
Skirt/Brim Line Width		0.4 mm
Support Line Width		0.4 mm
Support Interface Line Width		0.4 mm
Support Roof Line Width		0.4 mm
Support Floor Line Width		0.4 mm
Initial Layer Line Width		120%

** You can also increase Layer Height to 0.2 to reduce printing time (printing quality will decrease).*

Walls

Wall Thickness	f_x	1.2 mm
Wall Line Count	f_x	3
Wall Transition Length		0.2 mm
Wall Distribution Count		1
Wall Transitioning Threshold Angle		10.0
Wall Transitioning Filter Margin		0.1
Outer Wall Wipe Distance		0.2 mm
Outer Wall Inset		0 mm
Optimize Wall Printing Order		✔
Wall Ordering		Inside To Outside
Minimum Wall Line Width		0.34 mm
Minimum Even Wall Line Width		0.34 mm

Minimum Odd Wall Line Width		0.34 mm
Print Thin Walls		✔
Minimum Feature Size		0.1
Minimum Thin Wall Line Width		0.34
Horizontal Expansion		0 mm
Initial Layer Horizontal Expansion		0 mm
Hole horizontal expansion		0 mm
Z Seam Alignment		User Specified
Z Seam Position		Back
Z Seam X		0
Z Seam Y	f_x	200
Seam Corner Preference		None

Top/Bottom

Top Surface Skin Layers		0
Top/Bottom Thickness		1.2
Top Thickness		1.2
Top Layers	f_x	
Bottom Thickness		1.2
Bottom Layers	f_x	
Initial Bottom Layers	f_x	
Top/Bottom Pattern		Lines

Bottom Pattern Initial Layer		Lines
Extra Skin Wall Count		1
Skin Overlap Percentage	*fx*	10%
Skin Overlap		0.04 mm
Skin Removal Width		1.2 mm
Top Skin Removal Width		1.2 mm
Bottom Skin Removal Width		1.2 mm
Skin Expand Distance		1.2 mm
Top Skin Expand Distance		1.2 mm
Bottom Skin Expand Distance		1.2 mm
Maximum Skin Angle for Expansion		90
Minimum Width Angle for Expansion		0.0

Infill

Infill *		12%
Infill Density		10.0
Infill Line Distance	*fx*	Triangles
Connect Infill Lines	*fx*	✔
Infill Line Directions		[ ]
Infill X Offset		0.0
Infill Y Offset		0.0
Infill Line Multiplier		1
Extra Infill Wall Count		0

Infill Overlap Percentage	fx	15.0
Infill Overlap		0.06
Infill Wipe Distance		0.1
Infill Layer Thickness		0.14
Gradual Infill Steps		0
Infill Before Walls		✔
Minimum Infill Area		0.0
Skin Edge Support Thickness		0.0
Skin Edge Support Layers		0

** For small parts and all parts with connectors use 100% Infill.*

Material

Printing Temperature *	fx	195 °C
Printing Temperature Initial Laye	fx	195 °C
Initial Printing Temperature	fx	195 °C
Final Printing Temperature	fx	195 °C
Build Plate Temperature		60 °C
Build Plate Temperature Initial Layer		60 °C
Scaling Factor Shrinkage Compensation		100 %
Horizontal Scaling Factor Shrinkage Compensation		100 %
Vertical Scaling Factor Shrinkage Compensation		100 %
Flow		98 - 102%

** For PLA Plastic. If you are using a different type of plastic, check the printing temperature suggested by the manufacturer.*

Speed

Print Speed *		50 mm/s
Infill Speed	fx	50 mm/s
Wall Speed	fx	50 mm/s
Outer Wall Speed	fx	25 mm/s
Inner Wall Speed	fx	25 mm/s
Top/Bottom Speed	fx	25 mm/s
Support Speed	fx	35.0 mm/s
Support Infill Speed		35.0 mm/s
Support Interface Speed		30.0 mm/s
Support Roof Speed		30.0 mm/s
Support Floor Speed		30.0 mm/s
Travel Speed	fx	100 mm/s
Initial Layer Speed	fx	
Initial Layer Print Speed	fx	10 mm/s
Initial Laver Travel Speed	fx	10 mm/s
Skirt/Brim Speed	fx	15mm/s
Z Hop Speed		5.0
Number of Slower Layers		2
Flow Equalization Ratio		100%
Enable Acceleration Control **		✔

** For straight and simple objects you can increase the speed by 50%, for small parts you need to decrease the speed by 25% - 50%*

*** For straight and simple objects you need to set all acceleration parameters to 500 mm/s, for small parts you need to decrease the speed by 50% - 70%.*

Travel

Enable Retraction		✔
Retraction Distance	*fx*	6 mm
Retraction Speed		40mm/s
Retraction Retract Speed		40mm/s
Retraction Prime Speed		40mm/s
Retraction Extra Prime Amount: 0 mm3		0 mm^3
Retraction Minimum Travel		1.5 mm
Maximum Retraction Count		90
Minimum Extrusion Distance Window		6 - 10 mm
Combing Mode		All
Max Comb Distance With No Retract		30 mm
Retract Before Outer Wall		✔
Avoid Printed Parts When Traveling		✔
Avoid Supports When Traveling		✔
Travel Avoid Distance		0.625
Layer Start X		0.0
Layer Start Y		0.0
Z Hop When Retracted		✔

Z Hop Only Over Printed Parts		✔
Z Hop Height		3 mm

Cooling

Enable Print Cooling		✔
Fan Speed	*fx*	100%
Regular Fan Speed		100%
Maximum Fan Speed	*fx*	100%
Regular/Maximum Fan Speed Threshold		10 s
Initial Fan Speed		0%
Regular Fan Speed at Height	*fx*	0.5 mm
Regular Fan Speed at Layer	*fx*	3
Minimum Layer Time		5 s
Minimum Speed		10 mm/s

Support

Generate Support		✔
Support Structure *		Normal
Support Placement		Everywhere
Support Overhang Angle **		60°
Support Pattern		Zig Zag
Support Wall line count ***		0

Support Density	*fx*	10%
Support Line Distance		4 mm
Support Bottom Distance		4 mm
Support Infill Line Directions		[ ]
Enable Support Brim	*fx*	✔
Support Brim Width		8 mm
Support Brim Line Count		17
Support Z Distance ****		0.2 mm
Support Top Distance		0.2 mm
Support Bottom Distance	*fx*	0.12 mm
Support X/Y Distance		1 mm
Support Distance Priority		Z overrides X/Y
Minimum Support X/Y distance	*fx*	1.0
Support Stair Step Height		0.3 mm
Support Stair Step Maximum Width		5.0 mm
Support Stair Step Minimum Slope Angle		10 °
Support Join Distance		2 mm
Support Horizontal Expansion		0.2 mm
Support Infill Layer Thickness		0.2 mm
Gradual Support Infill Steps		0
Minimum Support Area		2 mm
Enable Support Interface		✔
Enable Support Roof	*fx*	✔
Enable Support Floor		✔

Support Interface Thickness		0.8 mm
Support Roof Thickness		0.8 mm
Support Floor Thickness		0.8 mm
Support Interface Resolution		0.2 mm
Support Interface Density		50 -100%
Support Roof Density		50 -100%
Support Roof Line Distance		0.8 mm
Support Floor Density		50 -100%
Support Floor Line Distance		0.4mm
Support Interface Pattern		Concentric
Support Roof Pattern *****		Concentric
Support Floor Pattern		Concentric
Minimum Support Interface Area		10 mm
Minimum Support Roof Area		10 mm
Minimum Support Floor Area		10 mm
Support Interface Horizontal Expansion		0.0 mm
Support Roof Horizontal Expansion		0.0 mm
Support Floor Horizontal Expansion		0.0 mm
Fan Speed Override		✔
Supported Skin Fan Speed		100%
Use Towers		✔
Tower Diameter		4 mm
Maximum Tower Supported Diameter		3.0 mm

Tower Roof Angle		65°

** You could try using Tree structure if you have difficulty printing models.*

*** You can set this parameter from 30° to 70°.*

**** You can also use 1 - the supports will be stronger but harder to remove.*

***** You can also increase this parameter for easier support removal, but keep in mind that it might spoil the surface quality.*

****** This parameter should be different from the Interface Pattern.*

Build Plate Adhesion

Build Plate Adhesion Type*		Brim
Skirt/Brim Minimum Length		250 mm
Brim Width*		6
Brim Line Count		12
Brim Distance		0.0
Brim Replaces Support		✔
Brim Only on Outside		✔

**We recommend using a Brim for better adhesion to the table, but if you normally use glue for better adhesion then you can use a Skirt.*

Special Modes

Print Sequence		All at Once
Surface Mode		Normal

Dear customer, if you have any questions or suggestions for the printing settings for the CURA slicer v 5.2 , you can email our ***Support Team*** *- support@gambody.com. We will be happy to assist you!*

Best regards,

Your team