Cura 5.8.1 settings

By Gambody v 1.0

These are averaged settings which were tested in the Cura 5.8.1 slicer. Test models were printed on Anycubic Cobra 2, Creality Ender 3 S1 with Anycubic PLA grey 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
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	*fx*	1.2 mm
Wall Line Count	*fx*	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 mm
Outer Wall Inset		0 mm
Optimize Wall Printing Order		✔
Wall Ordering		Inside To Outside
Alternative Extra Wall		none
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 mm

Minimum Thin Wall Line Width		0.34 mm
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	fx	200
Seam Corner Preference		Smart Hiding
Z Seam Relative		

Top/Bottom

Top Surface Skin Layers		0
Top/Bottom Thickness		1.2
Top Thickness		1.2
Top Layers	fx	
Bottom Thickness		1.2
Bottom Layers	fx	
Initial Bottom Layers	fx	
Top/Bottom Pattern		Lines
Bottom Pattern Initial Layer		Lines
Monotonic Top/Bottom Order		✔

Top/Bottom Line Direction		[]
Small Top/Bottom width		0.8 mm
Small Top/Bottom On Surface		none
No Skin in Z Gaps		none
Extra Skin Wall Count		1
Enable Ironing		none
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 *		5 %
Infill Density		10.0
Infill Line Distance	fx	Gyroid
Connect Infill Lines	fx	✔
Randomize infill Start		none
Infill X Offset		0.0

Infill Y Offset		0.0
Infill Line Multiplier		1
Extra Infill Wall Count		0
Infill Overlap Percentage	fx	30
Infill Overlap		0.12
Infill Wipe Distance		0.1
Infill Layer Thickness		0.2
Gradual Infill Steps		0
Infill Before Walls		✔
Minimum Infill Area		1.0
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	210 ºC
Printing Temperature Initial Laye*	fx	210 ºC
Initial Printing Temperature*	fx	210 ºC
Final Printing Temperature*	fx	210 ºC
Build Plate Temperature*		60 ºC
Build Plate Temperature Initial Layer*		60 ºC
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. Also read the description on your filament roll and increase these parameters for fast printers

Speed

Print Speed *		120 mm/s
Infill Speed		120 mm/s
Wall Speed		60 mm/s
Outer Wall Speed		60 mm/s
Inner Wall Speed		60 mm/s
Top Surface Outer Wall Speed		60 mm/s
Top Surface Inner Wall Speed		60 mm/s
Top/Bottom Speed		60 mm/s
Support Speed		60 mm/s
Support Infill Speed		60 mm/s
Support Interface Speed		60 mm/s
Support Roof Speed		60 mm/s
Support Floor Speed		60 mm/s
Travel Speed	*fx*	120 mm/s
Initial Layer Speed		20 mm/s
Initial Layer Print Speed		20 mm/s
Initial Laver Travel Speed		100 mm/s
Skirt/Brim Speed	*fx*	20mm/s
Z Hop Speed		5.0

Number of Slower Layers		2
Flow Equalization Ratio		100 %
Enable Acceleration Control **		✔
Enable Travel Acceleration		✔

* For small parts or old printers you need to decrease the speed by 50% -70%

** 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		✔
Retract at layer Change		none
Retraction Distance*		0.8 mm
Retraction Speed*		50 mm/s
Retraction Retract Speed*		50 mm/s
Retraction Prime Speed*		50 mm/s
Retraction Extra Prime Amount:		0 mm³
Retraction Minimum Travel		1.5 mm
Maximum Retraction Count		100
Minimum Extrusion Distance Window		2 - 3 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		none
Z Hop Only Over Printed Parts		✔
Z Hop Height		3 mm

* The settings are intended for modern fast printers with a direct extruder; if you use older printers, then see the settings from previous versions

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
Small Layer Printing Temperature		200

Cooling during extruder Switch		Unchanged

Support

Generate Support			✔
Support Structure *			Normal
Support Z Seam Away from Model			✔
Min Z seam Distance from Model			1 mm
Support Placement			Everywhere
Support Overhang Angle **			60°
Support Pattern			Lines
Support Wall line count ***			0
Support Interface Wall Line Count ***			0
Support Roof Wall Line Count ***			0
Support Bottom Wall Line Count ***			0
Connect Support Lines			none
Support Density		*fx*	5%
Support Line Distance			8 mm
Support Bottom Distance			8 mm
Support Infill Density Multiplier Initial Layer			1
Support Infill Line Directions			[ ]
Enable Support Brim		*fx*	✔
Support Brim Width			6 mm
Support Brim Line Count			9

Support Z Distance ****		0.2 mm
Support Top Distance		0.2 mm
Support Bottom Distance	*fx*	0.15 mm
Support X/Y Distance		1 mm
Support Distance Priority		X/Y overrides Z
Support Stair Step Height		0.0 mm
Minimum Support X/Y distance	*fx*	1.0
Support Stair Step Height		0.3 mm
Support Join Distance		2 mm
Support Horizontal Expansion		0.5 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 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			✔
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 5.8 , you can email our **Support Team** - support@gambody.com. We will be happy to assist you!*

Best regards,
Your Ge *team*