UltiMaker Cura 5.0+

Gambody printing recommendations for:

DeLorean 3D Printing Model | Assembly Kit 3: Car Interior, Wheels and Platforms

Below you'll find detailed slicing settings for UltiMaker Cura 5.0+ to help you get the best results when printing this model.

These settings are optimized specifically for this 3D model and should work well in most cases. But they're not set in stone - depending on your printer, material, or even the specific part you're working with, feel free to tweak things.

Every 3D printing setup is different, so feel free to make the changes that work best for your machine. When in doubt, check your printer's manual - or reach out to our Support Team at support@gambody.com

We'll be happy to help with any questions, suggestions, or issues you may have regarding the recommended printing settings!

Quality

Layer Height	**0.12 - 0.20** mm
For better quality use 0.12 mm layer height, for fast printing use 0.2 mm layer height. For pins and the Ge connectors, use 0.2 layer height.	
Initial Layer Height	**0.20** mm
120-150% of your Layer Height	

Line Width	0.42 mm
Wall Line Width	0.40 mm
Outer Wall Line Width	0.40 mm
Inner Wall(s) Line Width	0.42 mm
Top/Bottom Line Width	0.44 mm
Infill Line Width	0.40 mm
Skirt/Brim Line Width	0.42 mm
Initial Layer Line Width	0.5 %

Walls

Wall Thickness	0.80 mm
Wall Line Count	2
To increase the strength of the print parts, use wall line count: 3	
Wall Transition Length	0.20 mm
Wall Distribution Count	1
Wall Transitioning Threshold Angle	10.0 °
Wall Transitioning Filter Margin	0.10 mm
Outer Wall Wipe Distance	0.00 mm

Optimize Wall Printing Order	✔
Wall Ordering	Inside to Outsize
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	0.10 mm
Minimum Thin Wall Line Width	0.34 mm
Horizontal Expansion	0.00 mm
Initial Layer Horizontal Expansion	0.00 mm
Hole horizontal expansion	0.00 mm
Z Seam Alignment	User Specified
Z Seam Position	Back
Z Seam X	0.0 mm
Z Seam Y	200.0 mm

Top/Bottom

Top Surface Skin Layers	0 layers

Bottom Surface Skin Layers	0 layers
Top/Bottom Thickness	1.00 mm
Top Thickness	1.00 mm
Bottom Thickness	1.00 mm
Initial Bottom Layers	4 - 5 layers
Top/Bottom Pattern	Lines
Bottom Pattern Initial Layer	Lines
Extra Skin Wall Count	1
Enable Ironing	✘

Infill

For pins and connectors use 50% Infill	
Infill Density	5.0 - 30.0 %
Infill Line Distance	6.00 mm
Infill Pattern	Triangles
Infill X Offset	0.00 mm
Infill Y Offset	0.00 mm
Infill Line Multiplier	1

Infill Overlap Percentage	**30.0** %
Infill Overlap	**0.12** mm
Infill Wipe Distance	**0.00** mm
Infill Layer Thickness	**0.20** mm
Gradual Infill Steps	**0**
Minimum Infill Area	**0.00** mm^2
Skin Edge Support Thickness	**0.00** mm
Skin Edge Support Layers	**0**
Extra Infill Lines To Support Skins	**Walls and Lines**

Material

*These parameters are for standard PLA plastic. If you are using a different type of plastic, check the printing temperature recommended by the manufacturer. Also, read the description on your filament spool. For fast printers, add **+30 °C** to the current parameters.*

Printing Temperature	**210.0** ^0C
Printing Temperature Initial Layer	**210.0** ^0C
Initial Printing Temperature	**210.0** ^0C

Final Printing Temperature	210.0 °C
Build Plate Temperature	60.0 °C
Build Plate Temperature Initial Layer	60.0 °C
Scaling Factor Shrinkage Compensation	100.0 %
Horizontal Scaling Factor Shrinkage Compensation	100.0 %
Vertical Scaling Factor Shrinkage Compensation	100.0 %
Flow	98.0 - 102.0 %

Speed

The parameters in this tab vary greatly, it all depends on the quality of your printer. For example, if you have a classic Ender3, stick to the minimum parameters, but if you have a newer printer, for example Anycubic cobra 3 v3, you can select the maximum recommended values	
Print Speed	60.0 - 160.0 mm/sec
Infill Speed	60.0 - 160.0 mm/sec
Wall Speed	30.0 - 90.0 mm/sec
Outer Wall Speed	30.0 - 90.0 mm/sec
Inner Wall Speed	40.0 - 80.0 mm/sec

Top/Bottom Speed	40.0 - 100.0 mm/sec
Travel Speed	100.0 - 250.0 mm/sec
Initial Layer Speed	15.0 - 30.0 mm/sec
Initial Layer Print Speed	15.0 - 30.0 mm/sec
Initial Layer Travel Speed	100.0 - 150.0 mm/sec
Skirt/Brim Speed	20.0 - 30.0 mm/sec
Number of Slower Layers	2.0 layers
Flow Equalization Ratio	100.0 %
Enable Acceleration Control	✔
Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise.	
Print Acceleration	500.00 - 3000.00 mm/sec²
Infill Acceleration	500.0 - 3000.0 mm/sec²
Wall Acceleration	500.0 - 2000.0 mm/sec²

Outer Wall Acceleration	500.0 - 2000.0 mm/sec²
Inner Wall Acceleration	500.0 - 3000.0 mm/sec²
Top/Bottom Acceleration	500.0 - 2000.0 mm/sec²
Travel Acceleration	500.0 - 4000.0 mm/sec²
Initial Layer Acceleration	500.0 - 1000.0 mm/sec²
Initial Layer Print Acceleration	500.0 - 1000.0 mm/sec²
Initial Layer Travel Acceleration	500.0 - 3000.0 mm/sec²
Skirt/Brim Acceleration	500.0 - 2500.0 mm/sec²

Travel

Enable Retraction	✔
Retraction Distance	0.8 mm

You need to calibrate this parameter using Gambody test models. These values are average values for a Direct Drive extruder; for a Bowden extruder, the values should be increased.	
Retraction Speed	**35.0 - 50.0** mm/sec
You need to calibrate this parameter using Gambody test models. These values are average values for a Direct Drive extruder; for a Bowden extruder, the values should be increased.	
Retraction Retract Speed	**40.0** mm/sec
Retraction Prime Speed	**40.0** mm/sec
Retraction Extra Prime Amount	**0.00** mm^3
Retraction Minimum Travel	**1.49** mm
Maximum Retraction Count	**100**
Minimum Extrusion Distance Window	**2.00 - 3.00** mm
Combing Mode	**All**
Retract Before Outer Wall	✔
Layer Start X	**0.00** mm
Layer Start Y	**0.00** mm
Z Hop When Retracted	✔
Use this value other than 0 if your nozzle catches on the internal infill during travel moves. Try to keep this value as low as possible in height.	

Cooling

Enable Print Cooling	✔
Fan Speed	100.0 %
Regular Fan Speed	100.0 %
Maximum Fan Speed	100.0 %
Regular/Maximum Fan Speed Threshold	10.0 sec
Initial Fan Speed	0.0 %
Regular Fan Speed at Height	0.5 mm
Regular Fan Speed at Layer	3 layers
Minimum Layer Time	5.0 sec
Minimum Speed	10.0 mm/sec
Cooling during extruder switch	Unchanged

Support

Generate Support	✔
Support Structure Normal	✔
Use normal supports to support large, straight surfaces (most mechanical or technical parts).	

Support Z Seam Away from Model	
Min Z Seam Distance from Model	0.80 mm
Support Placement	Everywhere
Support Overhang Angle	45.0 - 65.0 °
You have to calibrate this parameter according to the capabilities of your printer and your filament, using a Gambody test models.	
Support Pattern	Zig Zag
Support Wall Line Count	0
Use 1 instead of 0 if your supports are thin and tall. They will be harder to remove, but much stronger.	
Support Interface Wall Line Count	0
Support Roof Wall Line Count	0
Support Bottom Wall Line Count	0
Connect Support ZigZags	✔
Support Density	5.0 - 20.0 %
Support Line Distance	2.0 - 8.0 mm
Enable Support Brim	✔
Support Brim Width	4.0 mm
Support Brim Line Count	10

Support Z Distance	0.24 mm
Top Z distance = 1-1.3 layer Height. If the supports are hard to remove, try increasing this setting by 0.1-0,4 mm	
Support Top Distance	0.24 mm
Support Bottom Distance	0.24 mm
Support X/Y Distance	0.80 mm
Increase this parameter if the supports are hard to remove from walls	
Support Distance Priority	X/Y overrides Z
Support Stair Step Height	0.00 mm
Support Join Distance	2.00 mm
Support Horizontal Expansion	0.80 mm
Support Infill Layer Thickness	0.20 mm
Gradual Support Infill Steps	0
Minimum Support Area	2.0 mm^2
Enable Support Interface	✔
Enable Support Roof	✔
Enable Support Floor	✔
Support Interface Thickness	0.40 mm

Support Roof Thickness	0.40 mm
Support Floor Thickness	0.40 mm
Support Interface Density	33.33 - 33.33 %
Support Roof Density	33.33 - 33.33 %
Support Roof Line Distance	2.40 mm
Support Floor Density	33.3 - 33.3 %
Support Floor Line Distance	2.40 mm
Support Interface Pattern	Concentric
Support Roof Pattern	Concentric
Support Floor Pattern	Concentric
Minimum Support Interface Area	10.0 mm^2
Minimum Support Roof Area	10.0 mm^2
Minimum Support Floor Area	10.0 mm^2
Support Interface Horizontal Expansion	0.0 mm
Support Roof Horizontal Expansion	0.0 mm
Support Floor Horizontal Expansion	0.0 mm
Use Towers	✔
Tower Diameter	3.0 mm

Maximum Tower-Supported Diameter	3.0 mm
Tower Roof Angle	65.0 °
Support Structure Tree	✔
Use tree supports to support complex objects, such as characters.	
Maximum Branch Angle	45.0 °
You have to calibrate this parameter according to the capabilities of your printer and your filament, using a Gambody test models.	
Branch Diameter	5.0 mm
Trunk Diameter	25.0 mm
Branch Diameter Angle	7.0 °
Support Z Seam Away from Model	✔
Min Z Seam Distance from Model	0.80 mm
Support Placement	Everywhere
Preferred Branch Angle	30.0 °
Diameter Increase To Model	1.0 mm
Minimum Height To Model	3.0 mm
Initial Layer Diameter	7.5 mm
Branch Density	30.0 %

Tip Diameter	0.8 mm
Limit Branch Reach	✔
Optimal Branch Range	30.0 mm
Rest Preference	On model if required
Support Overhang Angle	45.0 °
Support Pattern	Zig Zag
Support Wall Line Count	1
Support Interface Wall Line Count	0
Support Roof Wall Line Count	0
Support Bottom Wall Line Count	0
Connect Support ZigZags	✔
Support Density	0.0 %
Support Line Distance	0.0 %
Initial Layer Support Line Distance	0.0 %
Support Infill Density Multiplier Initial Layer	1
Enable Support Brim	✔
Support Brim Width	4.0 mm

Support Brim Line Count	10
Support Z Distance	0.24 mm
Top Z distance = 1-1.3 layer Height. If the supports are hard to remove, try increasing this setting by 0.1-0,4 mm	
Support Top Distance	0.22 mm
Support Bottom Distance	0.22 mm
Support X/Y Distance	0.22 mm
Increase this parameter if the supports are hard to remove from walls	
Support Distance Priority	X/Y overrides Z
Minimum Support Area	0.00 mm^2
Enable Support Interface	✔
Enable Support Roof	✔
Enable Support Floor	✔
Support Interface Thickness	0.80 mm
Support Roof Thickness	0.80 mm
Support Floor Thickness	0.80 mm
Support Interface Density	33.33 mm
Support Roof Density	33.33 %

Support Floor Density	2.39 %
Support Interface Pattern	Grid
Support Roof Pattern	Grid
Support Floor Pattern	Grid
Minimum Support Interface Area	10.00 mm²
Minimum Support Roof Area	10.00 mm²
Support Interface Horizontal Expansion	0.00 mm
Support Roof Horizontal Expansion	0.00 mm
Support Floor Horizontal Expansion	0.00 mm
Support Interface Priority	Interface preferred

Build Plate Adhesion

Build plate type: Skirt	
Use a skirt for all parts when printing on outdated printers.	
Skirt Line Count	3.00
Skirt Height	3.00
Skirt Distance	10.00 mm
Skirt/Brim Minimum Length	250 mm
Build plate type: Brim	

Use a brim when printing thin but tall parts, as well as parts with a small bed adhesion area.	
Skirt/Brim Minimum Length	**250.0** mm
Brim Width	**10.0** mm
Brim Line Count	**25**
Brim Distance	**0.0** mm
Brim Location	**Outside Only**
Brim Avoid Margin	**1.6** mm
Smart Brim	✔
Z Offset	**0.0** mm

Best regards,

your Ge team