# UltiMaker Cura 5.0+

Gambody printing recommendations for:

Doc Brown 3D Printer Files | Assembly

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** °C |
| **Printing Temperature Initial Layer** | **210.0** °C |
| **Initial Printing Temperature** | **210.0** °C |

| | |
|---|---|
| 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