

GUIDE FOR BASIC EDITION OF MATERIAL PROFILES IN 3DGENCE SLICER



Spis treści

1. INTRODUCTION	3
2. EDITION OF ADVANCED OPTIONS	3
2.1. Changing the height of the layer	6
2.2. Changing the density of the infill	11
	==

1. INTRODUCTION

3DGence Slicer has been prepared for 3DGence printers with ready-made print settings for dedicated materials. Each of the profiles available in 3DGence Slicer was developed by a team of specialists and then tested for over 1000 hours on many complicated test models. Therefore, at the beginning of the adventures with 3D printing, we recommend using default profiles. The manufacturer guarantees the highest possible quality of models made using dedicated software and materials as well as pre-set default settings.

For advanced users, an option to change print settings has been made available. This option is designed to modify existing profiles to get quality improvement on materials not certified by the manufacturer or adjusting the device's operating settings. Due to the nature of parameter modification, the manufacturer does not guarantee the quality and repeatability of prints prepared in this way.

Please note that the use of materials and print settings not provided by the manufacturer is the responsibility of the customer. This means that the manufacturer does not provide support for the use of advanced profiles.

3DGence Slicer uses the modified CuraEngine engine while maintaining parameter names, also experienced users can use the extensive documentation of the Cura software community. The names and functions of the functions are identical with the Cura software.

2. EDITION OF ADVANCED OPTIONS

The following is a brief guide to the basic edition of dedicated advanced profiles.

1. Select the "PROFILE" menu on the right.



2. Select the "LOAD custom profile" option.

Module	0.4	•
Model	PLA [3DGence]	۲
Support	Same as model	T
Visual	O High	
quality	 Normal 	
	O Draft	
Durability	Normal	
	O Enhanced	

3. Select the profile you want to edit from the list, then select "Open" (it is recommended to select the profile with the latest update date).

Device name		Type and manufacturer of model material	Type and manufacturer of support material		Strength	
1		↑	↑		1	
INDUSTRY F340	PRO	ABS Verbatim	+ HIPS Spectrum	Draft	Normal	ID59ver2.profile
	\checkmark			\checkmark		\checkmark
Model ty	ype/nozzle d	iameter		Visual quality		Profile ID
	O PRO AR	S Verbatim + HIPS-X Sn	ectrum Normal Norma			
INDUSTRY F34	0 PRO AB	5 Verbatim + HIPS-X Sp	ectrum Normal Norma			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_Normal_Norma			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_Normal_Enhan			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_Normal_Enhan			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_Normal_Enhan			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_High_Normal_I			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_High_Normal_I			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_High_Normal_I			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_High_Enhanced			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_High_Enhanced			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_High_Enhanced			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_Draft_Normal_			
INDUSTRY F34	0_PRO_AB	5 Verbatim + HIPS-X Sp	ectrum_Draft_Normal_			
INDUSTRY F34	0_PRO_AB	S Verbatim + HIPS-X Sp	ectrum_Draft_Normal_			

Guide for basic edition of material profiles in 3DGence slicer | version 02.2020

INDUSTRY F340_PRO_ABS Verbatim + HIPS Spectrum_Normal_Normal_

4. After loading the profile in the "PROFILES" menu, you will be able to see the change of the profile name to "Custom". To go to the advanced edition select ADJUST PRINT SETTINGS and then select EXPERT SETTINGS.

	PR	OFILE	
Module	Custom		•
Model	Custom		Y
Support	Custom		٧
Visual quality	Custom		
Durability	Custom		
LOAD CUS	TOM PROFILE		ADJUST PRINT SETTINGS
ADJUST PR	INT SETTINGS		
Infill percent ——@)	20%	
Infill pattern Grid	¥		
Side walls)	2	
Top walls		4	
Bottom walls)	3	
Bed adhesion Skir	t 🔻		
EXPER Note: changing the p print quality and dim	r settings arameters may affi ensional accuracy	ect	
	C ()		

ATTENTION!

It should be remembered that changes to these settings may require adjustment of other parameters. The following are the basic changes that should be applied until the desired results are obtained.

As follows from the observations - the most common changes of the default profiles concern changes in the layer height and density of the infill.

The lower layer height guarantees a more accurate mapping of the model in reality. The print time will increase as the layer height decreases (assuming that we leave the remaining parameters unchanged). However, increasing the layer height reduces the time of printing at the expense of the visual effect. It is assumed that the value of the layer height should not exceed the diameter of the nozzle. The infilling of the model is mainly responsible for the durability of the print. In contrast to the side walls and the bottom and top layers of the model, the infilling can be of different density (from 0 - 100%) the more densely we generate the infilling, the more durable the model will be, and its mass and print time will increase.

2.1. Changing the height of the layer

To change the layer height, select EXTRUDER TO (model) \rightarrow QUALITY-> LAYER HIGH in the advanced settings and then enter the new layer height value in the LAYER HEIGHT window.

ATTENTION: If the Layer Height will be change on Extruder T0, the value on Extruder T1 will change automatically on the same value.

Printer	Layer Height	8 🖉 0.25 mm	
	Initial Layer Height	8 0.25 mm	
Extruder TO (model)	^ Line Width	(i) 0.4 mm	
- Quality	- Wall Line Width	8 0.4 mm	
Shell Infill	Duter Wal	.ine Width 🔗 🐵 🛛 0.4 mm	
Material	Inner Wal.	.ine Width 🖉 🔞 🛛 0.4 mm	
Travel	- Top/Bottom Line	e Width 🔗 🐵 🛛 0.4 mm	
Support Cooling	– Infill Line Width	8 🛞 🛛 0.4 mm	
Adhesion Dual Extrusion		Width 🛞 🛛 0.4 mm	
Mesh Fixes Special Modes	Support Line Wi	idth 🔗 🛞 0.35 mm	
Experimental	Support Intee I	Line Width 🔗 🛞 🛛 0.4 mm	
	- Support Ro	of Line Width 🕜 🛛 0.4 mm	
Extruder T1 (support)	~		

When changing the height of the layer from larger to smaller (e.g. from 0.25mm to 0.2mm), pay special attention to:

• Correction of retraction settings:

1st Extruder - Material - Retraction Extra Prime Amount.

If on the printed model you notice material pouring after retraction, the value in the field Retraction Extra Prime Amount should be gradually decreased by 0.01mm3 until the desired result is obtained.

	CUSTOM SETTINGS	
	Retraction Speed 50 mm/s	
Printer	 Retraction Retract Speed 50 mm/s 	
	Retraction Prime Speed (a) 30 mm/s	
Extruder IU (model)	Retraction Extra Prime Amount	
Extruder TI (support)	Retraction Minimum Travel (6) S mm	
Machine	Maximum Retraction Count 90	
Quality Shell	Minimum Extrusion Distance Window 5 mm	
Infil Material	Limit Support Retractions	
Speed Travel	Standby Temperature 60°C	
Support	Nozzle Switch Retrection Distance (b) 5.3 mm	
Adhesion	Nozzle Switch Retraction Speed 50 mm/s	
Dual Extrusion Mesh Fixes	Nozzle Switch Retract Speed 50 mm/s	
Special Modes Experimental	Nozzle Switch Prime Speed (6) 30 mm/s	
	LOAD PROFILE ACTIVATE PROFILE SA	VE PROFILE

Correction of all hotend temperatures

1st Extruder - Material - Default Printing Temperature/Printing Temperature/Printing Temperature Initial Layer/Initial Printing Temperature.

It is recommended to reduce the temperature by 2°C. By reducing the layer height, we change the flow of the filament in time, a slight lowering of the temperature gently increases the resistance of pressing the material, which results in reduced inertia and improved printing stability. The values should be reduced until the desired result is obtained.

Printer		Default Printing Temperature	200°C
	-	Printing Temperature	200°C
Extruder TO (model)	~	Printing Temperature Initial Layer	205°C
Extruder TI (support)	<u> </u>	Initial Printing Temperature	195°C
Machine	~	Final Printing Temperature	195°C
Quality		Extrusion Cool Down Speed Modifier	0.7°C/s
Infil		Default Build Plate Temperature 🤞	2°00
Material Speed		Bulld Plate Temperature	50°C
Travel		Build Plate Temperature Initial Layer 👌	60°C
Cooling Adhesion		Adhesion Tendency	0
Dual Extrusion		Surface Energy	100%
Special Modes Experimental		Shrinkage Ratio	0%
		LOAD PROFILE	ACTIVATE PROFILE SAVE PROFILE

When changing the height of the layer <u>from smaller to greater</u> (e.g. from 0.15mm to 0.2mm), pay special attention to:

• Correction of retraction settings:

1st Extruder - Material - Retraction Extra Prime Amount.

If on the printed model you notice that the material is not in perfect condition after retraction, the value in the field Retraction Extra Prime Amount should be increased gradually by 0.01mm3 until the desired result is obtained.

		CUSTOM SETTINGS		\times
		Enable Retraction		
Printer				
		Retract at Layer Change		
Extruder TO (model)	~	Retraction Distance	5 mm	
		Retraction Speed	50 mm/s	
Extruder T1 (support)	^	 Retraction Retract Speed 	50 mm/s	
Quality		Retraction Prime Speed	30 mm/s	
Shell Infil		Retraction Extra Prime Amount	01 mm ³	
 Material Speed 		Retraction Minimum Travel	5 mm	
Travel		Maximum Retraction Count	90	
Cooling		Minimum Extrusion Distance Window	S mm	
Adhesion Dual Extrusion		Limit Support Retractions	~	
Mesh Fixes Special Modes		Standby Temperature	0°0	
Experimental		Nozzle Switch Retraction Distance 🔅	5.3 mm	
		LOAD PROFILE	ACTIVATE PROFILE SAVE PROFILE	

• Correction of all hotends temperatures:

1st Extruder - Material - Default Printing Temperature/Printing Temperature/Printing Temperature Initial Layer/Initial Printing Temperature.

It is recommended to increase the temperature gradually by 2 °C. By increasing the height of the layer, we increase the flow of the filament in time, a slight increase in temperature slightly reduces the resistance of the material, which results in thinning of the material and its more accurate feeding. Values should be increased until the desired result is obtained.

	CUSTOM SETTINGS	×
Printer	Default Printing Temperature 200°C	
	Printing Temperature (e) 200°C	
Extruder TO (model) 🗸 🗸	Printing Temperature Initial Layer 💿 205°C	
Extruder TI (support)	Initial Printing Temperature	
Machine	Final Printing Temperature (b) 195°C	
Quality Shell	Extrusion Cool Down Speed Modifier 0.7°C/s	
Infil	Default Bulid Plate Temperature 🔗 60°C	
- Material Speed	Build Plate Temperature 🔗 50°C	
Travel Support	Build Plate Temperature Initial Layer 🔗 🛛 🔞 🕫 С	
Cooling Adhesion	Adhesion Tendency 0	
Dual Extrusion	Surface Energy 100%	
Mesh Fixes Special Modes Experimental	Shrinkage Ratio 0%	
	LUAD PROFILE ACTIVATE PROFILE SAVE PR	

• Correction of all printing speeds:

1st Extruder - Speed - Infil Speed/Outer Wall Speed/Inner Wall Speed/Top/Bottom Speed.

It is recommended to reduce the speed iteratively by 3 mm/s. A slight reduction in velocity allows to eliminate many errors resulting from a higher layer height, as well as to reduce the effect of "ghosting", i.e. rounding of walls, which can be more visible at a higher layer.

Printer		Prin	t Speed	60 mm/s
		-	Infill Speed 🔗 🔞	60 mm/s
Extruder TO (model)	~	-	Wall Speed	30 mm/s
Extruder T1 (support)	~		– Duter Wall Speed 🔗 😥	30 mm/s
Machine			Inner Wall Speed 🔗 座	40 mm/s
Quality Shell		-	Top Surface Skin Speed	35 mm/s
Infill Material		-	Top/Bottom Speed 🔗 🙆	35 mm/s
- Speed		-	Support Speed	60 mm/s
Travel Support			– Support Infill Speed 🔗 🕑	40 mm/s
Cooling Adhesion			- Support Iace Speed 🔗 🔞	30 mm/s
Dual Extrusion			- Support Roof Speed	30 mm/s
Special Modes Experimental			Support Floor Speed 6	30 mm/s

2.2. Changing the density of the infill

To change the infilling density, one must first select 1ST EXTRUDER \rightarrow INFILL in the advanced settings and then enter the new infill density value in the INFILL DENSITY window.

	CUSTOM SETT	INGS	×
Printer	Infill Extruder	1st Extruder 3DGenceDoubleP255	~
	Infill Density	8 20%	
Extruder TO (model) 🗸	Infill Line Distance	8 2.0 mm	
Extruder T1 (support)	Infill Pattern	8 🐵 Zig Zag	~
Machine	Connect Infill Lines	8	
Quality Shell	Connect Infill Polygons	8	
- Infil	Infill Line Directions	8	
Speed	Infill X Offset	8 0 mm	
Travel Support	Infill Y Offset	8 0 mm	
Cooling Adhesion	Infill Line Multiplier	8 1	
Dual Extrusion	Extra Infill Wall Count	8	
Special Modes Experimental	Cubic Subdivision Shell	8 0.4 mm	
	LOAD PROFILE	ACTIVATE PROFILE SAVE PR	ROFILE

When changing the infilling density from smaller to larger (eg from 20% to 40%), particular attention should be paid to:

• Correction of infill settings:

1st Extruder - Infill - Infill Overlap Percentage.

If on the printout we notice the material collapse at the bonding points of the model with the walls and the value of the Infill Overlap Percentage parameter is greater than 0, reduce this value by 2% until the desired result is obtained.

1st Extruder - Infill - Skin Overlap Percentage.

If on the printout we notice the material stacking up at the points of joining the upper walls with the model walls and the value of the Skin Overlap Percentage parameter is greater than 0, this value should be decreased by 2% until the desired result is obtained.

	Infill Y Offset		8	0 mm]
Printer	Infill Line Multiplier		8	1]
Extruder TO (model)	Extra Infill Wall Count		8	0]
	Cubic Subdivision Shell		8	0.4 mm]
Extruder T1 (support)	Infill Overlap Percentage	P	1	0%]
Machine	Infill Overlap			0.0 mm]
Shell	Skin Overlap Percentage	S	(t)	0%]
— Infill Material	Skin Overlap			0.0 mm	ו
Speed Travel	Infill Wipe Distance	d	(fz)	0.0 mm]
Support	Infill Layer Thickness		8	0.15 mm]
Adhesion	Gradual Infill Steps		8	0]
Dual Extrusion Mesh Fixes	Gradual Infill Step Height		8	5.0 mm]
Special Modes Experimental	Infill Before Walls				
	LOAD PROFILE)	A	CTIVATE PROFILE SAVE PROFILE)

When changing the infilling density from larger to smaller (eg from 40% to 20%), pay special attention to:

• Correction of the number of upper and lower layers:

1st Extruder - Shell - Top Layers/Bottom Layers.

Decreasing the infilling it is worth increasing the number of upper and lower layers. This applies in particular to the case when we assume that the infilling is not needed or we only want to minimally strengthen the inside of the printed body using, for example, a 5% infill. Increasing the upper layers is important in the case of such a thin infill, because often on such a infilling we will have to build a flat surface.

	Wall Thickness	8	1.05 mm	
Printer	Wall Line Count	A (b)	2	
Extruder T0 (model)	Duter Wall Wipe Distance	S &	0.0 mm	
Excapa to filogal	Top Surface Skin Extruder	S	1st Extruder 3DGenceDoubleP255	
Extruder T1 (support)	Top Surface Skin Layers	P	0	
Machine	Top/Bottom Extruder	θ	1st Extruder 3DGenceDoubleP255	
- Shell	Top/Bottom Thickness	8	1.5 mm	
Infil Material Speed	- Top Thickness	в	15 mm	
Travel Support	Bottom Thickness	8 . ® 8	6 15 mm	
Cooling Adhesion Dual Extrusion	Bottom Layers	8 💌	4	
Mesh Fixes	Top/Bottom Pattern	8	Lines	
Special Modes Experimental	Bottom Pattern Initial Layer	в	Lines	