

## Print Settings

### Transform

slice

Option	Description	Example	Unit	Range
Scale	Enlarge or reduce your STL model	1.2 will scale the object to 120% of the original size eg 1= original size	file extension.stl	N/A
Rotate	Turns STL file on the printer plate	45' will turn the object 45' from original position.	angle in degrees	0' – to 360'
Copies along X	Creates copies of STL file on the X axis	Using "2" will produce 2 copies of the object on the X axis. 1 produced 1 print only.	Number of print copies	
Copies along Y	Creates copies of STL file on the Y axis	Using "3" will produce 2 copies of the object on the Y axis. 1 produced 1 print only.	Number of print copies	
Distance between copies	How much space (in mm) will be between copies.	Using "20" will space the objects 20mm apart (from the edge of each object)	mm	1 to ... 6 is a good setting

### Accuracy

Option	Description	Example	Unit	Range
Layer height (mm)	Defines the height of every layer	An object that is 5mm high and printed at 0.5mm layer height will be printed in 10 layers. With every layer 0.5mm high	mm	0.4mm nozzle has a range of 0.1 to 0.3mm
First Layer Height Ratio	This defines the ratio (percentage) at which the first layer will be printed	A ratio of 0.75 with layer height set at 0.4mm would mean a first layer printed at 0.3mm	%	0.2mm to 0.
Infill Every N Layers	Define how often to infill, rather than every layer.	2 would infill on every other layer.	number	1 to...

### Skirt and Brim

Option	Description	Example	Unit	Range
Loops	Defines the number of loops to print surrounding the object.	2 loops would mean that before printing the object, the printer will print 2 complete loops around the object. This is done to prime the extruder nozzle- so it delivers a smooth output- prior to printing the object.	number	0 (no loop) to ...

Distance From Object (mm)	The amount of distance between the loops and the object that will be printed.	10 would mean that the loops will be printed 10mm (1cm) away from the object.	mm	0 to ... (6mm default)
Skirt Height (Layers)	The amount of layers that the skirt will be printed for.	5 would mean the skirt is printed for the first 5 layers of the object.	mm	0 to ...

## Support Material

Option	Description	Example	Unit	Range
Overhang threshold	Threshold for overhangs in degrees.	45 is reasonable for most prints. If the setting is at 45' than the support will be added to the print automatically. 45' become the support adding trigger point.	Degrees	Below 90'
Pattern	Selects the support material pattern.			
		Honeycomb- prints the support in hexagonal columns.	N/A	N/A
		Rectilinear is a very common choice. It prints the support of your object as rectangular columns.	N/A	N/A
Pattern Spacing	Selects the spacing between instances of the print support	2.5 mm should produce support patterns which are 2.5mm apart.	N/A	N/A
Pattern Angle	Angle at which the pattern is drawn	Typically this setting is left at 0 degrees which prints vertical 90' columns.	N/A	N/A

## Print Settings

Option	Description	Example	Unit	Range
Perimeters [print settings]	The number of print layer for a single layer height.	eg. layer height = 0.3mm, Print_setting_perimeter = 2, than $0.3\text{mm}/2 =$ every layer is 0.15mm heigh. 2 layers will be printed. The benefit of using a larger perimeter number [than 1], is obtaining a smoother surface finish. The disadvantage is it will take more time to slice and more time to print.	mm	1 default range to be determined via test

<b>Solid Layers</b>	<b>The amount of layers that will be printed on the top and on the bottom of the object.</b>	<b>5 will produce 5 solid layers on the top and 5 layers on the bottom of the object.</b>	<b>mm</b>	<b>0 = no top or bottom layer. Likewise 3 will print 3 top and bottom layers</b>
<b>Fill Density</b>	<b>Is the amount of infill to be printed inside your object.</b>	<b>0.5 will fill 50% off of object inside with print plastic. Do not go above 80%. 25% make a soft object 80% a very hard object.</b>	<b>%</b>	<b>0.1 (no fill) to 0.6 is best. 1 = solid</b>
<b>Fill angle (°)</b>	<b>The angle at which the infill will rotated inside the object</b>	<b>45 degrees will start to draw the infill at a 45 degree angle between Y and X axis. Use 45' for round objects and 90' for square objects.</b>	<b>Degrees</b>	<b>0' to 360'</b>
<b>Fill Pattern and Solid Fill Pattern</b>	<b>The algorithm used to calculate the infill pattern</b>	<b>This is the style of infill used, normally Rectilinear is used, but here are some examples of what the others look like - they are all 37mm high and 25% infill, 3 outlines.</b>	<b>N/A</b>	<b>N/A</b>
		<b>Rectilinear – best for square or oblong objects</b>	<b>N/A</b>	<b>N/A</b>
		<b>Concentric – best for round objects</b>	<b>N/A</b>	<b>N/A</b>
		<b>Line</b>	<b>N/A</b>	<b>N/A</b>
		<b>Hilbert curve (slow)</b>	<b>N/A</b>	<b>N/A</b>
		<b>Archimedean cord (slow)</b>	<b>N/A</b>	<b>N/A</b>
		<b>octagramspirl (slow)</b>	<b>N/A</b>	<b>N/A</b>

## Retraction

<b>Option</b>	<b>Description</b>	<b>Example</b>	<b>Unit</b>	<b>Range</b>
<b>Length (mm)</b>	<b>The amount of filament to be retracted before it arrives at a window.</b>	<b>1 will retract 1mm of filament before the window is reached.</b>	<b>mm</b>	<b>0 = no retraction. 0 to ... best ????</b>
<b>Lift Z (mm)</b>	<b>The distance the Z-axis will move up before it moves to a new point of extrusion</b>	<b>5 will move the Z-axis up 5mm every time the extruder moves to a new point of extrusion</b>	<b>mm</b>	<b>0 to ...</b>
<b>Speed (mm/s)</b>	<b>The speed at which the Z-axis moves up and down</b>	<b>30 will move the Z-axis up and down at 30mm per second.</b>	<b>mm/s</b>	<b>0 to ...</b>

<b>Extra Length On Restart</b>	Extrude this amount of print filament in mm, after printing resumes on the far side of a window.	1 would add 1mm of extrusion.	mm	0 to ...
<b>Minimum Travel After Retraction (mm)</b>	The minimum amount for the extruder travel before it resumes printing on the far	2 means that the extruder will start printing 2mm before it reaches the far side of a window.	mm	0 to ...

## Printer and Filament

### Printer

<b>Option</b>	<b>Description</b>	<b>Example</b>	<b>Unit</b>	<b>Range</b>
<b>Nozzle Diameter</b>	The diameter of the nozzle you use.	0.4 means that your nozzle has a 0.4mm hole from which the plastic is extruded.	mm	0.2mm to 0.75mm
<b>Printer Center</b>	The center of the X-axis and Y-axis	On a print plate sized 200x200mm, X: 100 and Y: 100 means that the center of your print bed is 100 right and 100 back. The center of the print object will be aligned with this center point.	mm	According to print plate.
<b>Use Relative E Distances</b>	Specify extruder movements relative to the previous position, rather than in absolute terms	This is advanced stuff. Say you have printed a box. And you want to add another STL file, a semi sphere to the top of your box, then you need to use the relative XYZ position to place the semi sphere on the top of your box. Your start/homing position becomes the actual XYZ location of your semi sphere. There may also be some odd firmware designed that use this method of printing. Our firmware does not.	mm	N/A
<b>G-code flavour</b>		We use regrab/marlin	N/A	N/A
<b>Extrusion Axis</b>	The name of the axis (motor) that is used to extrude from	E means that the axis you use to extrude from is defined as the letter "E"	N/A	N/A
<b>Z Offset</b>	Amount to add to all Z axis coordinates.	1 would shift your entire print up 1mm. eg. You have a box and wish to print a semi sphere on top. Z OFFSET will need to be raised to the top of the box.	mm	0 to ...

## Filament

Option	Description	Example	Unit	Range
Diameter (mm)	The diameter of the filament used (generally 1.75 or 3mm)	3 means that your filament has a diameter of 3mm. It is vital to measure your filament at various points, to determine the average diameter. Eg one point is 3.00mm, another measures 2.8mm = $5.8 / 2 = 2.9$ mm filament diameter.	mm	1.75 or 3mm
Extrusion Multiplier	It determines the rotation speed of the extruder, which in turn	0.5 pushed $\frac{1}{2}$ the amount of PLA than 1. 2 increases the extruder push speed by double than 1. A small layer height eg 0.1mm and a fine object needs 0.25.	%	1= default. 2=Double speed.
Temperature (°C)	The hot-end temperature to print with (generally 200'-220°C for PLA and 250°C for ABS)	205' would mean that your hot end will be heated to 205°C before and during printing	(°C)	180' to 250' Safe range 180'-to 225'
First layer temperature (°C)	This command sets the temperature of the first print layer.	Use a higher temperature than the print temperature (5 - 15' more), to make the print stick better to your print plate.	(°C)	180' to 250' Safe range 180'-to 225'
Bed temperature (°C)	Not used by us.	Our print method does not require a hot bed. Use paint masking tape is your prints need to stick tighter to the plate.	(°C)	N/A
First layer bed temperature (°C)	N/A		(°C)	N/A

## Print Speed

Option	Description	Example	Unit	Range
<p style="text-align: center;"><b>If in doubt- set all to the same value. These settings are also tied to the extrusion multiplier.</b></p>				
<p style="text-align: center;"><b>eg. Set the <u>Perimeter</u> speed to 20mm per second and the <u>Infill</u> speed to 25-35mm max (</b></p>				<p style="text-align: center;"><b>Up to 35mm/s is Small objects</b></p>
<p style="text-align: center;"><b>eg. the nozzle extrudes 40mm of plastic per second, than the print speed should not</b></p>				
Perimeters (mm/s)	The speed at which the outer shell of the object will be printed.	30 means that the skin of an object will be printed at 30mm per second.	mm/s	1 to...

<b>Small Perimeters (mm/s)</b>	<b>The speed at which the outer shell of the object will be printed.</b>	<b>Set lower than Perimeter value. Small objects require a slower print speed. You also need to make the extrusion multiplier setting smaller, to reduce the amount of PLA flow from the print nozzle.</b>	<b>mm/s</b>	<b>1 to...</b>
<b>Infill (mm/s)</b>	<b>The speed for filling objects.</b>	<b>60 means that infill will be printed at 60mm per second.</b>	<b>mm/s</b>	<b>1 to...</b>
<b>Solid Infill (mm/s)</b>	<b>The speed for infill when solid objects are printed.</b>	<b>60 means that solid infill will be printed at 60mm per second.</b>	<b>mm/s</b>	<b>1 to...</b>
<b>Bridges (mm/s)</b>	<b>It determined the speed of the print head while traversing a gap.</b>	<b>60 means that bridges will be printed at 60mm per second.</b>	<b>mm/s</b>	<b>1 to...</b>
<b>Bridges are related to retraction.</b>	<b>Retraction pulls back the print filament before a gap to stop the flow of extruded</b>			

### Other Speed Settings

<b>Option</b>	<b>Description</b>	<b>Example</b>	<b>Unit</b>	<b>Range</b>
<b>Travel (mm/s)</b>	<b>The speed at which the axes move when they are not laying down plastic.</b>	<b>130 means that the axes will move at 130mm per second when the printer is travelling.</b>	<b>mm/s</b>	<b>1 to... Safe speed under 130mm/s</b>
<b>Bottom Layer Speed Ratio</b>	<b>The ratio at which the first layer will be printed.</b>	<b>0.3 means that the first layer will be printed at 30% in comparison to the other layers. eg. When you print a large object, you need good adhesion the the print plate. Use paint masking tape and eg 30% Bottom Layer Speed- to bond the PLA well to the print plate.</b>	<b>%</b>	<b>0.3 to 1 Best around 0.5.</b>