Settings and replacement patterns

Application-defined patterns

{time}	Time when slicing started
{date}	Date when slicing started
{day}	Day when slicing started
{initial_extruder_nr}	The first extruder train used for the print
{print_temperature}	Alias for material_print_temperature (deprecated, do not use)
{print_bed_temperature}	Alias for material_bed_temperature (deprecated, do not use)

Settings

The following settings are defined in fdmprinter.def.json.

	Machine	
{machine_name}	Machine Type	The name of your 3D printer model.
{machine_show_variants}	Show Machine Variants	Whether to show the different variants of this machine, which are described in separate json files.
{machine_start_gcode}	Start G-code	G-code commands to be executed at the very start - separated by \n.
{machine_end_gcode}	End G-code	G-code commands to be executed at the very end - separated by \n.
{material_guid}	Material GUID	GUID of the material. This is set automatically.
{material_diameter}	Diameter	Adjusts the diameter of the filament used. Match this value with the diameter of the used filament.
{material_bed_temp_wait}	Wait for Build Plate Heatup	Whether to insert a command to wait until the build plate temperature is reached at the start.
{material_print_temp_wait}	Wait for Nozzle Heatup	Whether to wait until the nozzle temperature is reached at the start.
{material_print_temp_prepend}	Include Material Temperatures	Whether to include nozzle temperature commands at the start of the gcode. When the start_gcode already contains nozzle temperature commands Cura frontend will automatically disable this setting.
{material_bed_temp_prepend}	Include Build	Whether to include build plate temperature commands at the start of the gcode. When the start_gcode already contains build plate

	Plate Temperature	temperature commands Cura frontend will automatically disable this setting.
{machine_width}	Machine Width	The width (X-direction) of the printable area.
{machine_depth}	Machine Depth	The depth (Y-direction) of the printable area.
{machine_shape}	Build Plate Shape	The shape of the build plate without taking unprintable areas into account.
{machine_buildplate_type}	Build Plate Material	The material of the build plate installed on the printer.
{machine_height}	Machine Height	The height (Z-direction) of the printable area.
{machine_heated_bed}	Has Heated Build Plate	Whether the machine has a heated build plate present.
{machine_heated_build_volume}	Has Build Volume Temperature Stabilization	Whether the machine is able to stabilize the build volume temperature.
{machine_always_write_active_tool}	Always Write Active Tool	Write active tool after sending temp commands to inactive tool. Required for Dual Extruder printing with Smoothie or other firmware with modal tool commands.
{machine_center_is_zero}	Is Center Origin	Whether the X/Y coordinates of the zero position of the printer is at the center of the printable area.
{machine_extruder_count}	Number of Extruders	Number of extruder trains. An extruder train is the combination of a feeder, bowden tube, and nozzle.
{extruders_enabled_count}	Number of Extruders That Are Enabled	Number of extruder trains that are enabled; automatically set in software
{machine_nozzle_tip_outer_diameter}	Outer Nozzle Diameter	The outer diameter of the tip of the nozzle.
{machine_nozzle_head_distance}	Nozzle Length	The height difference between the tip of the nozzle and the lowest part of the print head.
{machine_nozzle_expansion_angle}	Nozzle Angle	The angle between the horizontal plane and the conical part right above the tip of the nozzle.
{machine_heat_zone_length}	Heat Zone Length	The distance from the tip of the nozzle in which heat from the nozzle is transferred to the filament.
{machine_nozzle_temp_enabled}	Enable Nozzle Temperature	Whether to control temperature from Cura. Turn this off to control nozzle temperature from outside of

	Control	Cura.
{machine_nozzle_heat_up_speed}	Heat Up Speed	The speed (°C/s) by which the nozzle heats up averaged over the window of normal printing temperatures and the standby temperature.
{machine_nozzle_cool_down_speed}	Cool Down Speed	The speed (°C/s) by which the nozzle cools down averaged over the window of normal printing temperatures and the standby temperature.
{machine_min_cool_heat_time_window}	Minimal Time Standby Temperature	The minimal time an extruder has to be inactive before the nozzle is cooled. Only when an extruder is not used for longer than this time will it be allowed to cool down to the standby temperature.
{machine_gcode_flavor}	G-code Flavor	The type of g-code to be generated.
{machine_firmware_retract}	Firmware Retraction	Whether to use firmware retract commands (G10/G11) instead of using the E property in G1 commands to retract the material.
{machine_extruders_share_heater}	Extruders Share Heater	Whether the extruders share a single heater rather than each extruder having its own heater.
{machine_disallowed_areas}	Disallowed Areas	A list of polygons with areas the print head is not allowed to enter.
{nozzle_disallowed_areas}	Nozzle Disallowed Areas	A list of polygons with areas the nozzle is not allowed to enter.
{machine_head_with_fans_polygon}	Machine Head & Fan Polygon	A 2D silhouette of the print head (fan caps included).
{gantry_height}	Gantry Height	The height difference between the tip of the nozzle and the gantry system (X and Y axes).
{machine_nozzle_id}	Nozzle ID	The nozzle ID for an extruder train, such as "AA 0.4" and "BB 0.8".
{machine_nozzle_size}	Nozzle Diameter	The inner diameter of the nozzle. Change this setting when using a non-standard nozzle size.
{machine_use_extruder_offset_to_offset_coords}	Offset with Extruder	Apply the extruder offset to the coordinate system.
{extruder_prime_pos_z}	Extruder Prime Z Position	The Z coordinate of the position where the nozzle primes at the start of printing.
{extruder_prime_pos_abs}	Absolute Extruder Prime Position	Make the extruder prime position absolute rather than relative to the

		last-known location of the head.
{machine_max_feedrate_x}	Maximum Speed X	The maximum speed for the motor of the X-direction.
{machine_max_feedrate_y}	Maximum Speed Y	The maximum speed for the motor of the Y-direction.
{machine_max_feedrate_z}	Maximum Speed Z	The maximum speed for the motor of the Z-direction.
{machine_max_feedrate_e}	Maximum Feedrate	The maximum speed of the filament.
{machine_max_acceleration_x}	Maximum Acceleration X	Maximum acceleration for the motor of the X-direction
{machine_max_acceleration_y}	Maximum Acceleration Y	Maximum acceleration for the motor of the Y-direction.
{machine_max_acceleration_z}	Maximum Acceleration Z	Maximum acceleration for the motor of the Z-direction.
{machine_max_acceleration_e}	Maximum Filament Acceleration	Maximum acceleration for the motor of the filament.
{machine_acceleration}	Default Acceleration	The default acceleration of print head movement.
{machine_max_jerk_xy}	Default X-Y Jerk	Default jerk for movement in the horizontal plane.
{machine_max_jerk_z}	Default Z Jerk	Default jerk for the motor of the Z-direction.
{machine_max_jerk_e}	Default Filament Jerk	Default jerk for the motor of the filament.
{machine_steps_per_mm_x}	Steps per Millimeter (X)	How many steps of the stepper motor will result in one millimeter of movement in the X direction.
{machine_steps_per_mm_y}	Steps per Millimeter (Y)	How many steps of the stepper motor will result in one millimeter of movement in the Y direction.
{machine_steps_per_mm_z}	Steps per Millimeter (Z)	How many steps of the stepper motor will result in one millimeter of movement in the Z direction.
{machine_steps_per_mm_e}	Steps per Millimeter (E)	How many steps of the stepper motors will result in one millimeter of extrusion.
{machine_endstop_positive_direction_x}	X Endstop in Positive Direction	Whether the endstop of the X axis is in the positive direction (high X coordinate) or negative (low X coordinate).
{machine_endstop_positive_direction_y}	Y Endstop in Positive Direction	Whether the endstop of the Y axis is in the positive direction (high Y coordinate) or negative (low Y coordinate).

{machine_endstop_positive_direction_z}	Z Endstop in Positive Direction	Whether the endstop of the Z axis is in the positive direction (high Z coordinate) or negative (low Z coordinate).
{machine_minimum_feedrate}	Minimum Feedrate	The minimal movement speed of the print head.
{machine_feeder_wheel_diameter}	Feeder Wheel Diameter	The diameter of the wheel that drives the material in the feeder.
	Quality	
{layer_height}	Layer Height	The height of each layer in mm. Higher values produce faster prints in lower resolution, lower values produce slower prints in higher resolution.
{layer_height_0}	Initial Layer Height	The height of the initial layer in mm. A thicker initial layer makes adhesion to the build plate easier.
{line_width}	Line Width	Width of a single line. Generally, the width of each line should correspond to the width of the nozzle. However, slightly reducing this value could produce better prints.
{wall_line_width}	Wall Line Width	Width of a single wall line.
{wall_line_width_0}	Outer Wall Line Width	Width of the outermost wall line. By lowering this value, higher levels of detail can be printed.
{wall_line_width_x}	Inner Wall(s) Line Width	Width of a single wall line for all wall lines except the outermost one.
{skin_line_width}	Top/Bottom Line Width	Width of a single top/bottom line.
{infill_line_width}	Infill Line Width	Width of a single infill line.
{skirt_brim_line_width}	Skirt/Brim Line Width	Width of a single skirt or brim line.
{support_line_width}	Support Line Width	Width of a single support structure line.
{support_interface_line_width}	Support Interface Line Width	Width of a single line of support roof or floor.
{support_roof_line_width}	Support Roof Line Width	Width of a single support roof line.
{support_bottom_line_width}	Support Floor Line Width	Width of a single support floor line.
{prime_tower_line_width}	Prime Tower Line Width	Width of a single prime tower line.
{initial_layer_line_width_factor}	Initial Layer Line	Multiplier of the line width on the first layer. Increasing this could

	Width	improve bed adhesion.
	Shell	
{wall_extruder_nr}	Wall Extruder	The extruder train used for printing the walls. This is used in multi-extrusion.
{wall_0_extruder_nr}	Outer Wall Extruder	The extruder train used for printing the outer wall. This is used in multi-extrusion.
{wall_x_extruder_nr}	Inner Wall Extruder	The extruder train used for printing the inner walls. This is used in multi-extrusion.
{wall_thickness}	Wall Thickness	The thickness of the walls in the horizontal direction. This value divided by the wall line width defines the number of walls.
{wall_line_count}	Wall Line Count	The number of walls. When calculated by the wall thickness, this value is rounded to a whole number.
{wall_0_wipe_dist}	Outer Wall Wipe Distance	Distance of a travel move inserted after the outer wall, to hide the Z seam better.
{roofing_extruder_nr}	Top Surface Skin Extruder	The extruder train used for printing the top most skin. This is used in multi-extrusion.
{roofing_layer_count}	Top Surface Skin Layers	The number of top most skin layers. Usually only one top most layer is sufficient to generate higher quality top surfaces.
{top_bottom_extruder_nr}	Top/Bottom Extruder	The extruder train used for printing the top and bottom skin. This is used in multi-extrusion.
{top_bottom_thickness}	Top/Bottom Thickness	The thickness of the top/bottom layers in the print. This value divided by the layer height defines the number of top/bottom layers.
{top_thickness}	Top Thickness	The thickness of the top layers in the print. This value divided by the layer height defines the number of top layers.
{top_layers}	Top Layers	The number of top layers. When calculated by the top thickness, this value is rounded to a whole number.
{bottom_thickness}	Bottom Thickness	The thickness of the bottom layers in the print. This value divided by the layer height defines the number

		of bottom layers.
{bottom_layers}	Bottom Layers	The number of bottom layers. When calculated by the bottom thickness, this value is rounded to a whole number.
{initial_bottom_layers}	Initial Bottom Layers	The number of initial bottom layers, from the build-plate upwards. When calculated by the bottom thickness, this value is rounded to a whole number.
{top_bottom_pattern}	Top/Bottom Pattern	The pattern of the top/bottom layers.
{top_bottom_pattern_0}	Bottom Pattern Initial Layer	The pattern on the bottom of the print on the first layer.
{connect_skin_polygons}	Connect Top/Bottom Polygons	Connect top/bottom skin paths where they run next to each other. For the concentric pattern enabling this setting greatly reduces the travel time, but because the connections can happen midway over infill this feature can reduce the top surface quality.
{skin_angles}	Top/Bottom Line Directions	A list of integer line directions to use when the top/bottom layers use the lines or zig zag pattern. Elements from the list are used sequentially as the layers progress and when the end of the list is reached, it starts at the beginning again. The list items are separated by commas and the whole list is contained in square brackets. Default is an empty list which means use the traditional default angles (45 and 135 degrees).
{wall_0_inset}	Outer Wall Inset	Inset applied to the path of the outer wall. If the outer wall is smaller than the nozzle, and printed after the inner walls, use this offset to get the hole in the nozzle to overlap with the inner walls instead of the outside of the model.
{optimize_wall_printing_order}	Optimize Wall Printing Order	Optimize the order in which walls are printed so as to reduce the number of retractions and the distance travelled. Most parts will benefit from this being enabled but some may actually take longer so please compare the print time

		estimates with and without optimization. First layer is not optimized when choosing brim as build plate adhesion type.
{outer_inset_first}	Outer Before Inner Walls	Prints walls in order of outside to inside when enabled. This can help improve dimensional accuracy in X and Y when using a high viscosity plastic like ABS; however it can decrease outer surface print quality, especially on overhangs.
{alternate_extra_perimeter}	Alternate Extra Wall	Prints an extra wall at every other layer. This way infill gets caught between these extra walls, resulting in stronger prints.
{travel_compensate_overlapping_walls_enabled}	Compensate Wall Overlaps	Compensate the flow for parts of a wall being printed where there is already a wall in place.
{travel_compensate_overlapping_walls_0_enabled}	Compensate Outer Wall Overlaps	Compensate the flow for parts of an outer wall being printed where there is already a wall in place.
{travel_compensate_overlapping_walls_x_enabled}	Compensate Inner Wall Overlaps	Compensate the flow for parts of an inner wall being printed where there is already a wall in place.
{wall_min_flow}	Minimum Wall Flow	Minimum allowed percentage flow for a wall line. The wall overlap compensation reduces a wall's flow when it lies close to an existing wall. Walls whose flow is less than this value will be replaced with a travel move. When using this setting, you must enable the wall overlap compensation and print the outer wall before inner walls.
{wall_min_flow_retract}	Prefer Retract	If enabled, retraction is used rather than combing for travel moves that replace walls whose flow is below the minimum flow threshold.
{fill_perimeter_gaps}	Fill Gaps Between Walls	Fills the gaps between walls where no walls fit.
{filter_out_tiny_gaps}	Filter Out Tiny Gaps	Filter out tiny gaps to reduce blobs on outside of model.
{fill_outline_gaps}	Print Thin Walls	Print pieces of the model which are horizontally thinner than the nozzle size.
{xy_offset}	Horizontal	Amount of offset applied to all polygons in each layer. Positive values can compensate for too big

	Expansion	holes; negative values can compensate for too small holes.
{xy_offset_layer_0}	Initial Layer Horizontal Expansion	Amount of offset applied to all polygons in the first layer. A negative value can compensate for squishing of the first layer known as "elephant's foot".
{hole_xy_offset}	Hole Horizontal Expansion	Amount of offset applied to all holes in each layer. Positive values increase the size of the holes, negative values reduce the size of the holes.
{z_seam_type}	Z Seam Alignment	Starting point of each path in a layer. When paths in consecutive layers start at the same point a vertical seam may show on the print. When aligning these near a user specified location, the seam is easiest to remove. When placed randomly the inaccuracies at the paths' start will be less noticeable. When taking the shortest path the print will be quicker.
{z_seam_position}	Z Seam Position	The position near where to start printing each part in a layer.
{z_seam_x}	Z Seam X	The X coordinate of the position near where to start printing each part in a layer.
{z_seam_y}	Z Seam Y	The Y coordinate of the position near where to start printing each part in a layer.
{z_seam_corner}	Seam Corner Preference	Control whether corners on the model outline influence the position of the seam. None means that corners have no influence on the seam position. Hide Seam makes the seam more likely to occur on an inside corner. Expose Seam makes the seam more likely to occur on an outside corner. Hide or Expose Seam makes the seam more likely to occur at an inside or outside corner. Smart Hiding allows both inside and outside corners, but chooses inside corners more frequently, if appropriate.
		When enabled, the z seam coordinates are relative to each

{z_seam_relative}	Z Seam Relative	part's centre. When disabled, the coordinates define an absolute position on the build plate.
{skin_no_small_gaps_heuristic}	No Skin in Z Gaps	When the model has small vertical gaps of only a few layers, there should normally be skin around those layers in the narrow space. Enable this setting to not generate skin if the vertical gap is very small. This improves printing time and slicing time, but technically leaves infill exposed to the air.
{skin_outline_count}	Extra Skin Wall Count	Replaces the outermost part of the top/bottom pattern with a number of concentric lines. Using one or two lines improves roofs that start on infill material.
{ironing_enabled}	Enable Ironing	Go over the top surface one additional time, but this time extruding very little material. This is meant to melt the plastic on top further, creating a smoother surface. The pressure in the nozzle chamber is kept high so that the creases in the surface are filled with material.
{ironing_only_highest_layer}	Iron Only Highest Layer	Only perform ironing on the very last layer of the mesh. This saves time if the lower layers don't need a smooth surface finish.
{ironing_pattern}	Ironing Pattern	The pattern to use for ironing top surfaces.
{ironing_line_spacing}	Ironing Line Spacing	The distance between the lines of ironing.
{ironing_flow}	Ironing Flow	The amount of material, relative to a normal skin line, to extrude during ironing. Keeping the nozzle filled helps filling some of the crevices of the top surface, but too much results in overextrusion and blips on the side of the surface.
{ironing_inset}	Ironing Inset	A distance to keep from the edges of the model. Ironing all the way to the edge of the mesh may result in a jagged edge on your print.
{speed_ironing}	Ironing Speed	The speed at which to pass over the top surface.
{acceleration_ironing}	Ironing Acceleration	The acceleration with which ironing is performed.

{jerk_ironing}	Ironing Jerk	The maximum instantaneous velocity change while performing ironing.
{skin_overlap}	Skin Overlap Percentage	Adjust the amount of overlap between the walls and (the endpoints of) the skin-centerlines, as a percentage of the line widths of the skin lines and the innermost wall. A slight overlap allows the walls to connect firmly to the skin. Note that, given an equal skin and wall line-width, any percentage over 50% may already cause any skin to go past the wall, because at that point the position of the nozzle of the skin-extruder may already reach past the middle of the wall.
{skin_overlap_mm}	Skin Overlap	Adjust the amount of overlap between the walls and (the endpoints of) the skin-centerlines. A slight overlap allows the walls to connect firmly to the skin. Note that, given an equal skin and wall line-width, any value over half the width of the wall may already cause any skin to go past the wall, because at that point the position of the nozzle of the skin-extruder may already reach past the middle of the wall.
	Infill	
{infill_extruder_nr}	Infill Extruder	The extruder train used for printing infill. This is used in multi-extrusion.
{infill_sparse_density}	Infill Density	Adjusts the density of infill of the print.
{infill_line_distance}	Infill Line Distance	Distance between the printed infill lines. This setting is calculated by the infill density and the infill line width.
{infill_pattern}	Infill Pattern	The pattern of the infill material of the print. The line and zig zag infill swap direction on alternate layers, reducing material cost. The grid, triangle, tri-hexagon, cubic, octet, quarter cubic, cross and concentric patterns are fully printed every layer. Gyroid, cubic, quarter cubic and octet infill change with every

		layer to provide a more equal distribution of strength over each direction.
{zig_zaggify_infill}	Connect Infill Lines	Connect the ends where the infill pattern meets the inner wall using a line which follows the shape of the inner wall. Enabling this setting can make the infill adhere to the walls better and reduce the effects of infill on the quality of vertical surfaces. Disabling this setting reduces the amount of material used.
{connect_infill_polygons}	Connect Infill Polygons	Connect infill paths where they run next to each other. For infill patterns which consist of several closed polygons, enabling this setting greatly reduces the travel time.
{infill_angles}	Infill Line Directions	A list of integer line directions to use. Elements from the list are used sequentially as the layers progress and when the end of the list is reached, it starts at the beginning again. The list items are separated by commas and the whole list is contained in square brackets. Default is an empty list which means use the traditional default angles (45 and 135 degrees for the lines and zig zag patterns and 45 degrees for all other patterns).
{infill_offset_x}	Infill X Offset	The infill pattern is moved this distance along the X axis.
{infill_offset_y}	Infill Y Offset	The infill pattern is moved this distance along the Y axis.
{infill_randomize_start_location}	Randomize Infill Start	Randomize which infill line is printed first. This prevents one segment becoming the strongest, but it does so at the cost of an additional travel move.
{infill_multiplier}	Infill Line Multiplier	Convert each infill line to this many lines. The extra lines do not cross over each other, but avoid each other. This makes the infill stiffer, but increases print time and material usage.
		Add extra walls around the infill area. Such walls can make top/bottom skin lines sag down less

{infill_wall_line_count}	Extra Infill Wall Count	which means you need less top/bottom skin layers for the same quality at the cost of some extra material. This feature can combine with the Connect Infill Polygons to connect all the infill into a single extrusion path without the need for travels or retractions if configured right.
{sub_div_rad_add}	Cubic Subdivision Shell	An addition to the radius from the center of each cube to check for the boundary of the model, as to decide whether this cube should be subdivided. Larger values lead to a thicker shell of small cubes near the boundary of the model.
{infill_overlap}	Infill Overlap Percentage	The amount of overlap between the infill and the walls as a percentage of the infill line width. A slight overlap allows the walls to connect firmly to the infill.
{infill_overlap_mm}	Infill Overlap	The amount of overlap between the infill and the walls. A slight overlap allows the walls to connect firmly to the infill.
{infill_wipe_dist}	Infill Wipe Distance	Distance of a travel move inserted after every infill line, to make the infill stick to the walls better. This option is similar to infill overlap, but without extrusion and only on one end of the infill line.
{infill_sparse_thickness}	Infill Layer Thickness	The thickness per layer of infill material. This value should always be a multiple of the layer height and is otherwise rounded.
{gradual_infill_steps}	Gradual Infill Steps	Number of times to reduce the infill density by half when getting further below top surfaces. Areas which are closer to top surfaces get a higher density, up to the Infill Density.
{gradual_infill_step_height}	Gradual Infill Step Height	The height of infill of a given density before switching to half the density.
{infill_before_walls}	Infill Before Walls	Print the infill before printing the walls. Printing the walls first may lead to more accurate walls, but overhangs print worse. Printing the infill first leads to sturdier walls, but the infill pattern might sometimes

		show through the surface.
{min_infill_area}	Minimum Infill Area	Don't generate areas of infill smaller than this (use skin instead).
{infill_support_enabled}	Infill Support	Print infill structures only where tops of the model should be supported. Enabling this reduces print time and material usage, but leads to ununiform object strength.
{infill_support_angle}	Infill Overhang Angle	The minimum angle of internal overhangs for which infill is added. At a value of 0° objects are totally filled with infill, 90° will not provide any infill.
{skin_preshrink}	Skin Removal Width	The largest width of skin areas which are to be removed. Every skin area smaller than this value will disappear. This can help in limiting the amount of time and material spent on printing top/bottom skin at slanted surfaces in the model.
{top_skin_preshrink}	Top Skin Removal Width	The largest width of top skin areas which are to be removed. Every skin area smaller than this value will disappear. This can help in limiting the amount of time and material spent on printing top skin at slanted surfaces in the model.
{bottom_skin_preshrink}	Bottom Skin Removal Width	The largest width of bottom skin areas which are to be removed. Every skin area smaller than this value will disappear. This can help in limiting the amount of time and material spent on printing bottom skin at slanted surfaces in the model.
{expand_skins_expand_distance}	Skin Expand Distance	The distance the skins are expanded into the infill. Higher values makes the skin attach better to the infill pattern and makes the walls on neighboring layers adhere better to the skin. Lower values save amount of material used.
{top_skin_expand_distance}	Top Skin Expand Distance	The distance the top skins are expanded into the infill. Higher values makes the skin attach better to the infill pattern and makes the walls on the layer above adhere

		better to the skin. Lower values save amount of material used.
{bottom_skin_expand_distance}	Bottom Skin Expand Distance	The distance the bottom skins are expanded into the infill. Higher values makes the skin attach better to the infill pattern and makes the skin adhere better to the walls on the layer below. Lower values save amount of material used.
{max_skin_angle_for_expansion}	Maximum Skin Angle for Expansion	Top and/or bottom surfaces of your object with an angle larger than this setting, won't have their top/bottom skin expanded. This avoids expanding the narrow skin areas that are created when the model surface has a near vertical slope. An angle of 0° is horizontal and will cause no skin to be expanded, while an angle of 90° is vertical and will cause all skin to be expanded.
{min_skin_width_for_expansion}	Minimum Skin Width for Expansion	Skin areas narrower than this are not expanded. This avoids expanding the narrow skin areas that are created when the model surface has a slope close to the vertical.
{skin_edge_support_thickness}	Skin Edge Support Thickness	The thickness of the extra infill that supports skin edges.
{skin_edge_support_layers}	Skin Edge Support Layers	The number of infill layers that supports skin edges.
	Material	
{default_material_print_temperature}	Default Printing Temperature	The default temperature used for printing. This should be the "base" temperature of a material. All other print temperatures should use offsets based on this value
{build_volume_temperature}	Build Volume Temperature	The temperature of the environment to print in. If this is 0, the build volume temperature will not be adjusted.
{material_print_temperature}	Printing Temperature	The temperature used for printing.
{material_print_temperature_layer_0}	Printing Temperature Initial Layer	The temperature used for printing the first layer. Set at 0 to disable special handling of the initial layer.
{material_initial_print_temperature}	Initial Printing Temperature	The minimal temperature while heating up to the Printing Temperature at which printing can

		already start.
{material_final_print_temperature}	Final Printing Temperature	The temperature to which to already start cooling down just before the end of printing.
{material_extrusion_cool_down_speed}	Extrusion Cool Down Speed Modifier	The extra speed by which the nozzle cools while extruding. The same value is used to signify the heat up speed lost when heating up while extruding.
{default_material_bed_temperature}	Default Build Plate Temperature	The default temperature used for the heated build plate. This should be the "base" temperature of a build plate. All other print temperatures should use offsets based on this value
{material_bed_temperature}	Build Plate Temperature	The temperature used for the heated build plate. If this is 0, the build plate is left unheated.
{material_bed_temperature_layer_0}	Build Plate Temperature Initial Layer	The temperature used for the heated build plate at the first layer. If this is 0, the build plate is left unheated during the first layer.
{material_adhesion_tendency}	Adhesion Tendency	Surface adhesion tendency.
{material_surface_energy}	Surface Energy	Surface energy.
{material_shrinkage_percentage}	Scaling Factor Shrinkage Compensation	To compensate for the shrinkage of the material as it cools down, the model will be scaled with this factor.
{material_crystallinity}	Crystalline Material	Is this material the type that breaks off cleanly when heated (crystalline), or is it the type that produces long intertwined polymer chains (non-crystalline)?
{material_anti_ooze_retracted_position}	Anti-ooze Retracted Position	How far the material needs to be retracted before it stops oozing.
{material_anti_ooze_retraction_speed}	Anti-ooze Retraction Speed	How fast the material needs to be retracted during a filament switch to prevent oozing.
{material_break_preparation_retracted_position}	Break Preparation Retracted Position	How far the filament can be stretched before it breaks, while heated.
{material_break_preparation_speed}	Break Preparation Retraction Speed	How fast the filament needs to be retracted just before breaking it off in a retraction.
	Break Preparation	The temperature used to purge material, should be roughly equal to

{material_break_preparation_temperature}	Temperature	the highest possible printing temperature.
{material_break_retracted_position}	Break Retracted Position	How far to retract the filament in order to break it cleanly.
{material_break_speed}	Break Retraction Speed	The speed at which to retract the filament in order to break it cleanly.
{material_break_temperature}	Break Temperature	The temperature at which the filament is broken for a clean break.
{material_flush_purge_speed}	Flush Purge Speed	How fast to prime the material after switching to a different material.
{material_flush_purge_length}	Flush Purge Length	How much material to use to purge the previous material out of the nozzle (in length of filament) when switching to a different material.
{material_end_of_filament_purge_speed}	End of Filament Purge Speed	How fast to prime the material after replacing an empty spool with a fresh spool of the same material.
{material_end_of_filament_purge_length}	End of Filament Purge Length	How much material to use to purge the previous material out of the nozzle (in length of filament) when replacing an empty spool with a fresh spool of the same material.
{material_maximum_park_duration}	Maximum Park Duration	How long the material can be kept out of dry storage safely.
{material_no_load_move_factor}	No Load Move Factor	A factor indicating how much the filament gets compressed between the feeder and the nozzle chamber, used to determine how far to move the material for a filament switch.
{material_flow}	Flow	Flow compensation: the amount of material extruded is multiplied by this value.
{wall_material_flow}	Wall Flow	Flow compensation on wall lines.
{wall_0_material_flow}	Outer Wall Flow	Flow compensation on the outermost wall line.
{wall_x_material_flow}	Inner Wall(s) Flow	Flow compensation on wall lines for all wall lines except the outermost one.
{skin_material_flow}	Top/Bottom Flow	Flow compensation on top/bottom lines.
{roofing_material_flow}	Top Surface Skin Flow	Flow compensation on lines of the areas at the top of the print.
{infill_material_flow}	Infill Flow	Flow compensation on infill lines.
{skirt_brim_material_flow}	Skirt/Brim Flow	Flow compensation on skirt or brim lines.

{support_material_flow}	Support Flow	Flow compensation on support structure lines.
{support_interface_material_flow}	Support Interface Flow	Flow compensation on lines of support roof or floor.
{support_roof_material_flow}	Support Roof Flow	Flow compensation on support roof lines.
{support_bottom_material_flow}	Support Floor Flow	Flow compensation on support floor lines.
{prime_tower_flow}	Prime Tower Flow	Flow compensation on prime tower lines.
{material_flow_layer_0}	Initial Layer Flow	Flow compensation for the first layer: the amount of material extruded on the initial layer is multiplied by this value.
{material_standby_temperature}	Standby Temperature	The temperature of the nozzle when another nozzle is currently used for printing.
	Speed	
{speed_print}	Print Speed	The speed at which printing happens.
{speed_infill}	Infill Speed	The speed at which infill is printed.
{speed_wall}	Wall Speed	The speed at which the walls are printed.
{speed_wall_0}	Outer Wall Speed	The speed at which the outermost walls are printed. Printing the outer wall at a lower speed improves the final skin quality. However, having a large difference between the inner wall speed and the outer wall speed will affect quality in a negative way.
{speed_wall_x}	Inner Wall Speed	The speed at which all inner walls are printed. Printing the inner wall faster than the outer wall will reduce printing time. It works well to set this in between the outer wall speed and the infill speed.
{speed_roofing}	Top Surface Skin Speed	The speed at which top surface skin layers are printed.
{speed_topbottom}	Top/Bottom Speed	The speed at which top/bottom layers are printed.
{speed_support}	Support Speed	The speed at which the support structure is printed. Printing support at higher speeds can greatly reduce printing time. The surface quality of the support structure is not important since it is removed after printing.

{speed_support_infill}	Support Infill Speed	The speed at which the infill of support is printed. Printing the infill at lower speeds improves stability.
{speed_support_interface}	Support Interface Speed	The speed at which the roofs and floors of support are printed. Printing them at lower speeds can improve overhang quality.
{speed_support_roof}	Support Roof Speed	The speed at which the roofs of support are printed. Printing them at lower speeds can improve overhang quality.
{speed_support_bottom}	Support Floor Speed	The speed at which the floor of support is printed. Printing it at lower speed can improve adhesion of support on top of your model.
{speed_prime_tower}	Prime Tower Speed	The speed at which the prime tower is printed. Printing the prime tower slower can make it more stable when the adhesion between the different filaments is suboptimal.
{speed_travel}	Travel Speed	The speed at which travel moves are made.
{speed_layer_0}	Initial Layer Speed	The speed for the initial layer. A lower value is advised to improve adhesion to the build plate.
{speed_print_layer_0}	Initial Layer Print Speed	The speed of printing for the initial layer. A lower value is advised to improve adhesion to the build plate.
{speed_travel_layer_0}	Initial Layer Travel Speed	The speed of travel moves in the initial layer. A lower value is advised to prevent pulling previously printed parts away from the build plate. The value of this setting can automatically be calculated from the ratio between the Travel Speed and the Print Speed.
{skirt_brim_speed}	Skirt/Brim Speed	The speed at which the skirt and brim are printed. Normally this is done at the initial layer speed, but sometimes you might want to print the skirt or brim at a different speed.
{speed_z_hop}	Z Hop Speed	The speed at which the vertical Z movement is made for Z Hops. This is typically lower than the print speed since the build plate or machine's gantry is harder to move.

{speed_slowdown_layers}	Number of Slower Layers	The first few layers are printed slower than the rest of the model, to get better adhesion to the build plate and improve the overall success rate of prints. The speed is gradually increased over these layers.
{speed_equalize_flow_enabled}	Equalize Filament Flow	Print thinner than normal lines faster so that the amount of material extruded per second remains the same. Thin pieces in your model might require lines printed with smaller line width than provided in the settings. This setting controls the speed changes for such lines.
{speed_equalize_flow_max}	Maximum Speed for Flow Equalization	Maximum print speed when adjusting the print speed in order to equalize flow.
{acceleration_enabled}	Enable Acceleration Control	Enables adjusting the print head acceleration. Increasing the accelerations can reduce printing time at the cost of print quality.
{acceleration_print}	Print Acceleration	The acceleration with which printing happens.
{acceleration_infill}	Infill Acceleration	The acceleration with which infill is printed.
{acceleration_wall}	Wall Acceleration	The acceleration with which the walls are printed.
{acceleration_wall_0}	Outer Wall Acceleration	The acceleration with which the outermost walls are printed.
{acceleration_wall_x}	Inner Wall Acceleration	The acceleration with which all inner walls are printed.
{acceleration_roofing}	Top Surface Skin Acceleration	The acceleration with which top surface skin layers are printed.
{acceleration_topbottom}	Top/Bottom Acceleration	The acceleration with which top/bottom layers are printed.
{acceleration_support}	Support Acceleration	The acceleration with which the support structure is printed.
{acceleration_support_infill}	Support Infill Acceleration	The acceleration with which the infill of support is printed.
{acceleration_support_interface}	Support Interface Acceleration	The acceleration with which the roofs and floors of support are printed. Printing them at lower acceleration can improve overhang quality.
{acceleration_support_roof}	Support Roof Acceleration	The acceleration with which the roofs of support are printed. Printing them at lower acceleration

		can improve overhang quality.
{acceleration_support_bottom}	Support Floor Acceleration	The acceleration with which the floors of support are printed. Printing them at lower acceleration can improve adhesion of support on top of your model.
{acceleration_prime_tower}	Prime Tower Acceleration	The acceleration with which the prime tower is printed.
{acceleration_travel}	Travel Acceleration	The acceleration with which travel moves are made.
{acceleration_layer_0}	Initial Layer Acceleration	The acceleration for the initial layer.
{acceleration_print_layer_0}	Initial Layer Print Acceleration	The acceleration during the printing of the initial layer.
{acceleration_travel_layer_0}	Initial Layer Travel Acceleration	The acceleration for travel moves in the initial layer.
{acceleration_skirt_brim}	Skirt/Brim Acceleration	The acceleration with which the skirt and brim are printed. Normally this is done with the initial layer acceleration, but sometimes you might want to print the skirt or brim at a different acceleration.
{jerk_enabled}	Enable Jerk Control	Enables adjusting the jerk of print head when the velocity in the X or Y axis changes. Increasing the jerk can reduce printing time at the cost of print quality.
{jerk_print}	Print Jerk	The maximum instantaneous velocity change of the print head.
{jerk_infill}	Infill Jerk	The maximum instantaneous velocity change with which infill is printed.
{jerk_wall}	Wall Jerk	The maximum instantaneous velocity change with which the walls are printed.
{jerk_wall_0}	Outer Wall Jerk	The maximum instantaneous velocity change with which the outermost walls are printed.
{jerk_wall_x}	Inner Wall Jerk	The maximum instantaneous velocity change with which all inner walls are printed.
{jerk_roofing}	Top Surface Skin Jerk	The maximum instantaneous velocity change with which top surface skin layers are printed.
{jerk_topbottom}	Top/Bottom Jerk	The maximum instantaneous velocity change with which

		top/bottom layers are printed.
{jerk_support}	Support Jerk	The maximum instantaneous velocity change with which the support structure is printed.
{jerk_support_infill}	Support Infill Jerk	The maximum instantaneous velocity change with which the infill of support is printed.
{jerk_support_interface}	Support Interface Jerk	The maximum instantaneous velocity change with which the roofs and floors of support are printed.
{jerk_support_roof}	Support Roof Jerk	The maximum instantaneous velocity change with which the roofs of support are printed.
{jerk_support_bottom}	Support Floor Jerk	The maximum instantaneous velocity change with which the floors of support are printed.
{jerk_prime_tower}	Prime Tower Jerk	The maximum instantaneous velocity change with which the prime tower is printed.
{jerk_travel}	Travel Jerk	The maximum instantaneous velocity change with which travel moves are made.
{jerk_layer_0}	Initial Layer Jerk	The print maximum instantaneous velocity change for the initial layer.
{jerk_print_layer_0}	Initial Layer Print Jerk	The maximum instantaneous velocity change during the printing of the initial layer.
{jerk_travel_layer_0}	Initial Layer Travel Jerk	The acceleration for travel moves in the initial layer.
{jerk_skirt_brim}	Skirt/Brim Jerk	The maximum instantaneous velocity change with which the skirt and brim are printed.
	Travel	
{retraction_enable}	Enable Retraction	Retract the filament when the nozzle is moving over a non-printed area.
{retract_at_layer_change}	Retract at Layer Change	Retract the filament when the nozzle is moving to the next layer.
{retraction_amount}	Retraction Distance	The length of material retracted during a retraction move.
{retraction_speed}	Retraction Speed	The speed at which the filament is retracted and primed during a retraction move.
{retraction_retract_speed}	Retraction Retract Speed	The speed at which the filament is retracted during a retraction move.

{retraction_prime_speed}	Retraction Prime Speed	The speed at which the filament is primed during a retraction move.
{retraction_extra_prime_amount}	Retraction Extra Prime Amount	Some material can ooze away during a travel move, which can be compensated for here.
{retraction_min_travel}	Retraction Minimum Travel	The minimum distance of travel needed for a retraction to happen at all. This helps to get fewer retractions in a small area.
{retraction_count_max}	Maximum Retraction Count	This setting limits the number of retractions occurring within the minimum extrusion distance window. Further retractions within this window will be ignored. This avoids retracting repeatedly on the same piece of filament, as that can flatten the filament and cause grinding issues.
{retraction_extrusion_window}	Minimum Extrusion Distance Window	The window in which the maximum retraction count is enforced. This value should be approximately the same as the retraction distance, so that effectively the number of times a retraction passes the same patch of material is limited.
{limit_support_retractions}	Limit Support Retractions	Omit retraction when moving from support to support in a straight line. Enabling this setting saves print time, but can lead to excessive stringing within the support structure.
{retraction_combing}	Combing Mode	Combing keeps the nozzle within already printed areas when traveling. This results in slightly longer travel moves but reduces the need for retractions. If combing is off, the material will retract and the nozzle moves in a straight line to the next point. It is also possible to avoid combing over top/bottom skin areas or to only comb within the infill.
{retraction_combing_max_distance}	Max Comb Distance With No Retract	When non-zero, combing travel moves that are longer than this distance will use retraction.
{travel_retract_before_outer_wall}	Retract Before Outer Wall	Always retract when moving to start an outer wall.
	Avoid Printed	The nozzle avoids already printed

{travel_avoid_other_parts}	Parts When Traveling	parts when traveling. This option is only available when combing is enabled.
{travel_avoid_supports}	Avoid Supports When Traveling	The nozzle avoids already printed supports when traveling. This option is only available when combing is enabled.
{travel_avoid_distance}	Travel Avoid Distance	The distance between the nozzle and already printed parts when avoiding during travel moves.
{layer_start_x}	Layer Start X	The X coordinate of the position near where to find the part to start printing each layer.
{layer_start_y}	Layer Start Y	The Y coordinate of the position near where to find the part to start printing each layer.
{retraction_hop_enabled}	Z Hop When Retracted	Whenever a retraction is done, the build plate is lowered to create clearance between the nozzle and the print. It prevents the nozzle from hitting the print during travel moves, reducing the chance to knock the print from the build plate.
{retraction_hop_only_when_collides}	Z Hop Only Over Printed Parts	Only perform a Z Hop when moving over printed parts which cannot be avoided by horizontal motion by Avoid Printed Parts when Traveling.
{retraction_hop}	Z Hop Height	The height difference when performing a Z Hop.
{retraction_hop_after_extruder_switch}	Z Hop After Extruder Switch	After the machine switched from one extruder to the other, the build plate is lowered to create clearance between the nozzle and the print. This prevents the nozzle from leaving oozed material on the outside of a print.
{retraction_hop_after_extruder_switch_height}	Z Hop After Extruder Switch Height	The height difference when performing a Z Hop after extruder switch.
	Cooling	
{cool_fan_enabled}	Enable Print Cooling	Enables the print cooling fans while printing. The fans improve print quality on layers with short layer times and bridging / overhangs.
{cool_fan_speed}	Fan Speed	The speed at which the print cooling fans spin.
		The speed at which the fans spin

{cool_fan_speed_min}	Regular Fan Speed	before hitting the threshold. When a layer prints faster than the threshold, the fan speed gradually inclines towards the maximum fan speed.
{cool_fan_speed_max}	Maximum Fan Speed	The speed at which the fans spin on the minimum layer time. The fan speed gradually increases between the regular fan speed and maximum fan speed when the threshold is hit.
{cool_min_layer_time_fan_speed_max}	Regular/Maximum Fan Speed Threshold	The layer time which sets the threshold between regular fan speed and maximum fan speed. Layers that print slower than this time use regular fan speed. For faster layers the fan speed gradually increases towards the maximum fan speed.
{cool_fan_speed_0}	Initial Fan Speed	The speed at which the fans spin at the start of the print. In subsequent layers the fan speed is gradually increased up to the layer corresponding to Regular Fan Speed at Height.
{cool_fan_full_at_height}	Regular Fan Speed at Height	The height at which the fans spin on regular fan speed. At the layers below the fan speed gradually increases from Initial Fan Speed to Regular Fan Speed.
{cool_fan_full_layer}	Regular Fan Speed at Layer	The layer at which the fans spin on regular fan speed. If regular fan speed at height is set, this value is calculated and rounded to a whole number.
{cool_min_layer_time}	Minimum Layer Time	The minimum time spent in a layer. This forces the printer to slow down, to at least spend the time set here in one layer. This allows the printed material to cool down properly before printing the next layer. Layers may still take shorter than the minimal layer time if Lift Head is disabled and if the Minimum Speed would otherwise be violated.
{cool_min_speed}	Minimum Speed	The minimum print speed, despite slowing down due to the minimum layer time. When the printer would slow down too much, the pressure in the nozzle would be too low and

		result in bad print quality.
{cool_lift_head}	Lift Head	When the minimum speed is hit because of minimum layer time, lift the head away from the print and wait the extra time until the minimum layer time is reached.
	Support	
{support_enable}	Generate Support	Generate structures to support parts of the model which have overhangs. Without these structures, such parts would collapse during printing.
{support_extruder_nr}	Support Extruder	The extruder train to use for printing the support. This is used in multi-extrusion.
{support_infill_extruder_nr}	Support Infill Extruder	The extruder train to use for printing the infill of the support. This is used in multi-extrusion.
{support_extruder_nr_layer_0}	First Layer Support Extruder	The extruder train to use for printing the first layer of support infill. This is used in multi-extrusion.
{support_interface_extruder_nr}	Support Interface Extruder	The extruder train to use for printing the roofs and floors of the support. This is used in multi-extrusion.
{support_roof_extruder_nr}	Support Roof Extruder	The extruder train to use for printing the roofs of the support. This is used in multi-extrusion.
{support_bottom_extruder_nr}	Support Floor Extruder	The extruder train to use for printing the floors of the support. This is used in multi-extrusion.
{support_structure}	Support Structure	Chooses between the techniques available to generate support. "Normal" support creates a support structure directly below the overhanging parts and drops those areas straight down. "Tree" support creates branches towards the overhanging areas that support the model on the tips of those branches, and allows the branches to crawl around the model to support it from the build plate as much as possible.
{support_tree_angle}	Tree Support Branch Angle	The angle of the branches. Use a lower angle to make them more vertical and more stable. Use a higher angle to be able to have more reach.

{support_tree_branch_distance}	Tree Support Branch Distance	How far apart the branches need to be when they touch the model. Making this distance small will cause the tree support to touch the model at more points, causing better overhang but making support harder to remove.
{support_tree_branch_diameter}	Tree Support Branch Diameter	The diameter of the thinnest branches of tree support. Thicker branches are more sturdy. Branches towards the base will be thicker than this.
{support_tree_branch_diameter_angle}	Tree Support Branch Diameter Angle	The angle of the branches' diameter as they gradually become thicker towards the bottom. An angle of 0 will cause the branches to have uniform thickness over their length. A bit of an angle can increase stability of the tree support.
{support_tree_collision_resolution}	Tree Support Collision Resolution	Resolution to compute collisions with to avoid hitting the model. Setting this lower will produce more accurate trees that fail less often, but increases slicing time dramatically.
{support_type}	Support Placement	Adjusts the placement of the support structures. The placement can be set to touching build plate or everywhere. When set to everywhere the support structures will also be printed on the model.
{support_angle}	Support Overhang Angle	The minimum angle of overhangs for which support is added. At a value of 0° all overhangs are supported, 90° will not provide any support.
{support_pattern}	Support Pattern	The pattern of the support structures of the print. The different options available result in sturdy or easy to remove support.
{support_wall_count}	Support Wall Line Count	The number of walls with which to surround support infill. Adding a wall can make support print more reliably and can support overhangs better, but increases print time and material used.
		Connect the ends of the support lines together. Enabling this setting

{zig_zaggify_support}	Connect Support Lines	can make your support more sturdy and reduce underextrusion, but it will cost more material.
{support_connect_zigzags}	Connect Support ZigZags	Connect the ZigZags. This will increase the strength of the zig zag support structure.
{support_infill_rate}	Support Density	Adjusts the density of the support structure. A higher value results in better overhangs, but the supports are harder to remove.
{support_line_distance}	Support Line Distance	Distance between the printed support structure lines. This setting is calculated by the support density.
{support_initial_layer_line_distance}	Initial Layer Support Line Distance	Distance between the printed initial layer support structure lines. This setting is calculated by the support density.
{support_infill_angles}	Support Infill Line Directions	A list of integer line directions to use. Elements from the list are used sequentially as the layers progress and when the end of the list is reached, it starts at the beginning again. The list items are separated by commas and the whole list is contained in square brackets. Default is an empty list which means use the default angle 0 degrees.
{support_brim_enable}	Enable Support Brim	Generate a brim within the support infill regions of the first layer. This brim is printed underneath the support, not around it. Enabling this setting increases the adhesion of support to the build plate.
{support_brim_width}	Support Brim Width	The width of the brim to print underneath the support. A larger brim enhances adhesion to the build plate, at the cost of some extra material.
{support_brim_line_count}	Support Brim Line Count	The number of lines used for the support brim. More brim lines enhance adhesion to the build plate, at the cost of some extra material.
{support_z_distance}	Support Z Distance	Distance from the top/bottom of the support structure to the print. This gap provides clearance to remove the supports after the model is printed. This value is rounded up to

		a multiple of the layer height.
{support_top_distance}	Support Top Distance	Distance from the top of the support to the print.
{support_bottom_distance}	Support Bottom Distance	Distance from the print to the bottom of the support.
{support_xy_distance}	Support X/Y Distance	Distance of the support structure from the print in the X/Y directions.
{support_xy_overrides_z}	Support Distance Priority	Whether the Support X/Y Distance overrides the Support Z Distance or vice versa. When X/Y overrides Z the X/Y distance can push away the support from the model, influencing the actual Z distance to the overhang. We can disable this by not applying the X/Y distance around overhangs.
{support_xy_distance_overhang}	Minimum Support X/Y Distance	Distance of the support structure from the overhang in the X/Y directions.
{support_bottom_stair_step_height}	Support Stair Step Height	The height of the steps of the stair-like bottom of support resting on the model. A low value makes the support harder to remove, but too high values can lead to unstable support structures. Set to zero to turn off the stair-like behaviour.
{support_bottom_stair_step_width}	Support Stair Step Maximum Width	The maximum width of the steps of the stair-like bottom of support resting on the model. A low value makes the support harder to remove, but too high values can lead to unstable support structures.
{support_bottom_stair_step_min_slope}	Support Stair Step Minimum Slope Angle	The minimum slope of the area for stair-stepping to take effect. Low values should make support easier to remove on shallower slopes, but really low values may result in some very counter-intuitive results on other parts of the model.
{support_join_distance}	Support Join Distance	The maximum distance between support structures in the X/Y directions. When separate structures are closer together than this value, the structures merge into one.
{support_offset}	Support Horizontal Expansion	Amount of offset applied to all support polygons in each layer. Positive values can smooth out the support areas and result in more

		sturdy support.
{support_infill_sparse_thickness}	Support Infill Layer Thickness	The thickness per layer of support infill material. This value should always be a multiple of the layer height and is otherwise rounded.
{gradual_support_infill_steps}	Gradual Support Infill Steps	Number of times to reduce the support infill density by half when getting further below top surfaces. Areas which are closer to top surfaces get a higher density, up to the Support Infill Density.
{gradual_support_infill_step_height}	Gradual Support Infill Step Height	The height of support infill of a given density before switching to half the density.
{minimum_support_area}	Minimum Support Area	Minimum area size for support polygons. Polygons which have an area smaller than this value will not be generated.
{support_interface_enable}	Enable Support Interface	Generate a dense interface between the model and the support. This will create a skin at the top of the support on which the model is printed and at the bottom of the support, where it rests on the model.
{support_roof_enable}	Enable Support Roof	Generate a dense slab of material between the top of support and the model. This will create a skin between the model and support.
{support_bottom_enable}	Enable Support Floor	Generate a dense slab of material between the bottom of the support and the model. This will create a skin between the model and support.
{support_interface_height}	Support Interface Thickness	The thickness of the interface of the support where it touches with the model on the bottom or the top.
{support_roof_height}	Support Roof Thickness	The thickness of the support roofs. This controls the amount of dense layers at the top of the support on which the model rests.
{support_bottom_height}	Support Floor Thickness	The thickness of the support floors. This controls the number of dense layers that are printed on top of places of a model on which support rests.
		When checking where there's model above and below the support, take steps of the given height. Lower

{support_interface_skip_height}	Support Interface Resolution	values will slice slower, while higher values may cause normal support to be printed in some places where there should have been support interface.
{support_interface_density}	Support Interface Density	Adjusts the density of the roofs and floors of the support structure. A higher value results in better overhangs, but the supports are harder to remove.
{support_roof_density}	Support Roof Density	The density of the roofs of the support structure. A higher value results in better overhangs, but the supports are harder to remove.
{support_roof_line_distance}	Support Roof Line Distance	Distance between the printed support roof lines. This setting is calculated by the Support Roof Density, but can be adjusted separately.
{support_bottom_density}	Support Floor Density	The density of the floors of the support structure. A higher value results in better adhesion of the support on top of the model.
{support_bottom_line_distance}	Support Floor Line Distance	Distance between the printed support floor lines. This setting is calculated by the Support Floor Density, but can be adjusted separately.
{support_interface_pattern}	Support Interface Pattern	The pattern with which the interface of the support with the model is printed.
{support_roof_pattern}	Support Roof Pattern	The pattern with which the roofs of the support are printed.
{support_bottom_pattern}	Support Floor Pattern	The pattern with which the floors of the support are printed.
{minimum_interface_area}	Minimum Support Interface Area	Minimum area size for support interface polygons. Polygons which have an area smaller than this value will be printed as normal support.
{minimum_roof_area}	Minimum Support Roof Area	Minimum area size for the roofs of the support. Polygons which have an area smaller than this value will be printed as normal support.
{minimum_bottom_area}	Minimum Support Floor Area	Minimum area size for the floors of the support. Polygons which have an area smaller than this value will be printed as normal support.

{support_interface_offset}	Support Interface Horizontal Expansion	Amount of offset applied to the support interface polygons.
{support_roof_offset}	Support Roof Horizontal Expansion	Amount of offset applied to the roofs of the support.
{support_bottom_offset}	Support Floor Horizontal Expansion	Amount of offset applied to the floors of the support.
{support_interface_angles}	Support Interface Line Directions	A list of integer line directions to use. Elements from the list are used sequentially as the layers progress and when the end of the list is reached, it starts at the beginning again. The list items are separated by commas and the whole list is contained in square brackets. Default is an empty list which means use the default angles (alternates between 45 and 135 degrees if interfaces are quite thick or 90 degrees).
{support_roof_angles}	Support Roof Line Directions	A list of integer line directions to use. Elements from the list are used sequentially as the layers progress and when the end of the list is reached, it starts at the beginning again. The list items are separated by commas and the whole list is contained in square brackets. Default is an empty list which means use the default angles (alternates between 45 and 135 degrees if interfaces are quite thick or 90 degrees).
{support_bottom_angles}	Support Floor Line Directions	A list of integer line directions to use. Elements from the list are used sequentially as the layers progress and when the end of the list is reached, it starts at the beginning again. The list items are separated by commas and the whole list is contained in square brackets. Default is an empty list which means use the default angles (alternates between 45 and 135 degrees if interfaces are quite thick or 90 degrees).

{support_fan_enable}	Fan Speed Override	When enabled, the print cooling fan speed is altered for the skin regions immediately above the support.
{support_supported_skin_fan_speed}	Supported Skin Fan Speed	Percentage fan speed to use when printing the skin regions immediately above the support. Using a high fan speed can make the support easier to remove.
{support_use_towers}	Use Towers	Use specialized towers to support tiny overhang areas. These towers have a larger diameter than the region they support. Near the overhang the towers' diameter decreases, forming a roof.
{support_tower_diameter}	Tower Diameter	The diameter of a special tower.
{support_tower_maximum_supported_diameter}	Maximum Tower-Supported Diameter	Maximum diameter in the X/Y directions of a small area which is to be supported by a specialized support tower.
{support_tower_roof_angle}	Tower Roof Angle	The angle of a rooftop of a tower. A higher value results in pointed tower roofs, a lower value results in flattened tower roofs.
{support_mesh_drop_down}	Drop Down Support Mesh	Make support everywhere below the support mesh, so that there's no overhang in the support mesh.
{support_meshes_present}	Scene Has Support Meshes	There are support meshes present in the scene. This setting is controlled by Cura.
Build Plate Adhesion		
{prime_blob_enable}	Enable Prime Blob	Whether to prime the filament with a blob before printing. Turning this setting on will ensure that the extruder will have material ready at the nozzle before printing. Printing Brim or Skirt can act like priming too, in which case turning this setting off saves some time.
{extruder_prime_pos_x}	Extruder Prime X Position	The X coordinate of the position where the nozzle primes at the start of printing.
{extruder_prime_pos_y}	Extruder Prime Y Position	The Y coordinate of the position where the nozzle primes at the start of printing.
		Different options that help to improve both priming your extrusion and adhesion to the build plate. Brim adds a single layer flat

{adhesion_type}	Build Plate Adhesion Type	area around the base of your model to prevent warping. Raft adds a thick grid with a roof below the model. Skirt is a line printed around the model, but not connected to the model.
{adhesion_extruder_nr}	Build Plate Adhesion Extruder	The extruder train to use for printing the skirt/brim/raft. This is used in multi-extrusion.
{skirt_line_count}	Skirt Line Count	Multiple skirt lines help to prime your extrusion better for small models. Setting this to 0 will disable the skirt.
{skirt_gap}	Skirt Distance	The horizontal distance between the skirt and the first layer of the print. This is the minimum distance. Multiple skirt lines will extend outwards from this distance.
{skirt_brim_minimal_length}	Skirt/Brim Minimum Length	The minimum length of the skirt or brim. If this length is not reached by all skirt or brim lines together, more skirt or brim lines will be added until the minimum length is reached. Note: If the line count is set to 0 this is ignored.
{brim_width}	Brim Width	The distance from the model to the outermost brim line. A larger brim enhances adhesion to the build plate, but also reduces the effective print area.
{brim_line_count}	Brim Line Count	The number of lines used for a brim. More brim lines enhance adhesion to the build plate, but also reduces the effective print area.
{brim_gap}	Brim Distance	The horizontal distance between the first brim line and the outline of the first layer of the print. A small gap can make the brim easier to remove while still providing the thermal benefits.
{brim_replaces_support}	Brim Replaces Support	Enforce brim to be printed around the model even if that space would otherwise be occupied by support. This replaces some regions of the first layer of support by brim regions.
	Brim Only on	Only print the brim on the outside of the model. This reduces the

{brim_outside_only}	Outside	amount of brim you need to remove afterwards, while it doesn't reduce the bed adhesion that much.
{raft_margin}	Raft Extra Margin	If the raft is enabled, this is the extra raft area around the model which is also given a raft. Increasing this margin will create a stronger raft while using more material and leaving less area for your print.
{raft_smoothing}	Raft Smoothing	This setting controls how much inner corners in the raft outline are rounded. Inward corners are rounded to a semi circle with a radius equal to the value given here. This setting also removes holes in the raft outline which are smaller than such a circle.
{raft_airgap}	Raft Air Gap	The gap between the final raft layer and the first layer of the model. Only the first layer is raised by this amount to lower the bonding between the raft layer and the model. Makes it easier to peel off the raft.
{layer_0_z_overlap}	Initial Layer Z Overlap	Make the first and second layer of the model overlap in the Z direction to compensate for the filament lost in the airgap. All models above the first model layer will be shifted down by this amount.
{raft_surface_layers}	Raft Top Layers	The number of top layers on top of the 2nd raft layer. These are fully filled layers that the model sits on. 2 layers result in a smoother top surface than 1.
{raft_surface_thickness}	Raft Top Layer Thickness	Layer thickness of the top raft layers.
{raft_surface_line_width}	Raft Top Line Width	Width of the lines in the top surface of the raft. These can be thin lines so that the top of the raft becomes smooth.
{raft_surface_line_spacing}	Raft Top Spacing	The distance between the raft lines for the top raft layers. The spacing should be equal to the line width, so that the surface is solid.
{raft_interface_thickness}	Raft Middle Thickness	Layer thickness of the middle raft layer.

{raft_interface_line_width}	Raft Middle Line Width	Width of the lines in the middle raft layer. Making the second layer extrude more causes the lines to stick to the build plate.
{raft_interface_line_spacing}	Raft Middle Spacing	The distance between the raft lines for the middle raft layer. The spacing of the middle should be quite wide, while being dense enough to support the top raft layers.
{raft_base_thickness}	Raft Base Thickness	Layer thickness of the base raft layer. This should be a thick layer which sticks firmly to the printer build plate.
{raft_base_line_width}	Raft Base Line Width	Width of the lines in the base raft layer. These should be thick lines to assist in build plate adhesion.
{raft_base_line_spacing}	Raft Base Line Spacing	The distance between the raft lines for the base raft layer. Wide spacing makes for easy removal of the raft from the build plate.
{raft_speed}	Raft Print Speed	The speed at which the raft is printed.
{raft_surface_speed}	Raft Top Print Speed	The speed at which the top raft layers are printed. These should be printed a bit slower, so that the nozzle can slowly smooth out adjacent surface lines.
{raft_interface_speed}	Raft Middle Print Speed	The speed at which the middle raft layer is printed. This should be printed quite slowly, as the volume of material coming out of the nozzle is quite high.
{raft_base_speed}	Raft Base Print Speed	The speed at which the base raft layer is printed. This should be printed quite slowly, as the volume of material coming out of the nozzle is quite high.
{raft_acceleration}	Raft Print Acceleration	The acceleration with which the raft is printed.
{raft_surface_acceleration}	Raft Top Print Acceleration	The acceleration with which the top raft layers are printed.
{raft_interface_acceleration}	Raft Middle Print Acceleration	The acceleration with which the middle raft layer is printed.
{raft_base_acceleration}	Raft Base Print Acceleration	The acceleration with which the base raft layer is printed.
{raft_jerk}	Raft Print Jerk	The jerk with which the raft is

		printed.
{raft_surface_jerk}	Raft Top Print Jerk	The jerk with which the top raft layers are printed.
{raft_interface_jerk}	Raft Middle Print Jerk	The jerk with which the middle raft layer is printed.
{raft_base_jerk}	Raft Base Print Jerk	The jerk with which the base raft layer is printed.
{raft_fan_speed}	Raft Fan Speed	The fan speed for the raft.
{raft_surface_fan_speed}	Raft Top Fan Speed	The fan speed for the top raft layers.
{raft_interface_fan_speed}	Raft Middle Fan Speed	The fan speed for the middle raft layer.
{raft_base_fan_speed}	Raft Base Fan Speed	The fan speed for the base raft layer.
Dual Extrusion		
{prime_tower_enable}	Enable Prime Tower	Print a tower next to the print which serves to prime the material after each nozzle switch.
{prime_tower_size}	Prime Tower Size	The width of the prime tower.
{prime_tower_min_volume}	Prime Tower Minimum Volume	The minimum volume for each layer of the prime tower in order to purge enough material.
{prime_tower_position_x}	Prime Tower X Position	The x coordinate of the position of the prime tower.
{prime_tower_position_y}	Prime Tower Y Position	The y coordinate of the position of the prime tower.
{prime_tower_wipe_enabled}	Wipe Inactive Nozzle on Prime Tower	After printing the prime tower with one nozzle, wipe the oozed material from the other nozzle off on the prime tower.
{prime_tower_brim_enable}	Prime Tower Brim	Prime-towers might need the extra adhesion afforded by a brim even if the model doesn't. Presently can't be used with the 'Raft' adhesion-type.
{ooze_shield_enabled}	Enable Ooze Shield	Enable exterior ooze shield. This will create a shell around the model which is likely to wipe a second nozzle if it's at the same height as the first nozzle.
{ooze_shield_angle}	Ooze Shield Angle	The maximum angle a part in the ooze shield will have. With 0 degrees being vertical, and 90 degrees being horizontal. A smaller angle leads to less failed ooze shields, but more material.
{ooze_shield_dist}	Ooze Shield	Distance of the ooze shield from the

	Distance	print, in the X/Y directions.
{switch_extruder_retraction_amount}	Nozzle Switch Retraction Distance	The amount of retraction when switching extruders. Set to 0 for no retraction at all. This should generally be the same as the length of the heat zone.
{switch_extruder_retraction_speeds}	Nozzle Switch Retraction Speed	The speed at which the filament is retracted. A higher retraction speed works better, but a very high retraction speed can lead to filament grinding.
{switch_extruder_retraction_speed}	Nozzle Switch Retract Speed	The speed at which the filament is retracted during a nozzle switch retract.
{switch_extruder_prime_speed}	Nozzle Switch Prime Speed	The speed at which the filament is pushed back after a nozzle switch retraction.
{switch_extruder_extra_prime_amount}	Nozzle Switch Extra Prime Amount	Extra material to prime after nozzle switching.
	Mesh Fixes	
{meshfix_union_all}	Union Overlapping Volumes	Ignore the internal geometry arising from overlapping volumes within a mesh and print the volumes as one. This may cause unintended internal cavities to disappear.
{meshfix_union_all_remove_holes}	Remove All Holes	Remove the holes in each layer and keep only the outside shape. This will ignore any invisible internal geometry. However, it also ignores layer holes which can be viewed from above or below.
{meshfix_extensive_stitching}	Extensive Stitching	Extensive stitching tries to stitch up open holes in the mesh by closing the hole with touching polygons. This option can introduce a lot of processing time.
{meshfix_keep_open_polygons}	Keep Disconnected Faces	Normally Cura tries to stitch up small holes in the mesh and remove parts of a layer with big holes. Enabling this option keeps those parts which cannot be stitched. This option should be used as a last resort option when everything else fails to produce proper g-code.
{multiple_mesh_overlap}	Merged Meshes Overlap	Make meshes which are touching each other overlap a bit. This makes them bond together better.

{carve_multiple_volumes}	Remove Mesh Intersection	Remove areas where multiple meshes are overlapping with each other. This may be used if merged dual material objects overlap with each other.
{alternate_carve_order}	Alternate Mesh Removal	Switch to which mesh intersecting volumes will belong with every layer, so that the overlapping meshes become interwoven. Turning this setting off will cause one of the meshes to obtain all of the volume in the overlap, while it is removed from the other meshes.
{remove_empty_first_layers}	Remove Empty First Layers	Remove empty layers beneath the first printed layer if they are present. Disabling this setting can cause empty first layers if the Slicing Tolerance setting is set to Exclusive or Middle.
{meshfix_maximum_resolution}	Maximum Resolution	The minimum size of a line segment after slicing. If you increase this, the mesh will have a lower resolution. This may allow the printer to keep up with the speed it has to process g-code and will increase slice speed by removing details of the mesh that it can't process anyway.
{meshfix_maximum_travel_resolution}	Maximum Travel Resolution	The minimum size of a travel line segment after slicing. If you increase this, the travel moves will have less smooth corners. This may allow the printer to keep up with the speed it has to process g-code, but it may cause model avoidance to become less accurate.
{meshfix_maximum_deviation}	Maximum Deviation	The maximum deviation allowed when reducing the resolution for the Maximum Resolution setting. If you increase this, the print will be less accurate, but the g-code will be smaller. Maximum Deviation is a limit for Maximum Resolution, so if the two conflict the Maximum Deviation will always be held true.
Special Modes		
		Whether to print all models one layer at a time or to wait for one

{print_sequence}	Print Sequence	model to finish, before moving on to the next. One at a time mode is possible if a) only one extruder is enabled and b) all models are separated in such a way that the whole print head can move in between and all models are lower than the distance between the nozzle and the X/Y axes.
{infill_mesh}	Infill Mesh	Use this mesh to modify the infill of other meshes with which it overlaps. Replaces infill regions of other meshes with regions for this mesh. It's suggested to only print one Wall and no Top/Bottom Skin for this mesh.
{infill_mesh_order}	Mesh Processing Rank	Determines the priority of this mesh when considering multiple overlapping infill meshes. Areas where multiple infill meshes overlap will take on the settings of the mesh with the highest rank. An infill mesh with a higher rank will modify the infill of infill meshes with lower rank and normal meshes.
{cutting_mesh}	Cutting Mesh	Limit the volume of this mesh to within other meshes. You can use this to make certain areas of one mesh print with different settings and with a whole different extruder.
{mold_enabled}	Mold	Print models as a mold, which can be cast in order to get a model which resembles the models on the build plate.
{mold_width}	Minimal Mold Width	The minimal distance between the ouside of the mold and the outside of the model.
{mold_roof_height}	Mold Roof Height	The height above horizontal parts in your model which to print mold.
{mold_angle}	Mold Angle	The angle of overhang of the outer walls created for the mold. 0° will make the outer shell of the mold vertical, while 90° will make the outside of the model follow the contour of the model.
{support_mesh}	Support Mesh	Use this mesh to specify support areas. This can be used to generate support structure.

{anti_overhang_mesh}	Anti Overhang Mesh	Use this mesh to specify where no part of the model should be detected as overhang. This can be used to remove unwanted support structure.
{magic_mesh_surface_mode}	Surface Mode	Treat the model as a surface only, a volume, or volumes with loose surfaces. The normal print mode only prints enclosed volumes. "Surface" prints a single wall tracing the mesh surface with no infill and no top/bottom skin. "Both" prints enclosed volumes like normal and any remaining polygons as surfaces.
{magic_spiralize}	Spiralize Outer Contour	Spiralize smooths out the Z move of the outer edge. This will create a steady Z increase over the whole print. This feature turns a solid model into a single walled print with a solid bottom. This feature should only be enabled when each layer only contains a single part.
{smooth_spiralized_contours}	Smooth Spiralized Contours	Smooth the spiralized contours to reduce the visibility of the Z seam (the Z seam should be barely visible on the print but will still be visible in the layer view). Note that smoothing will tend to blur fine surface details.
{relative_extrusion}	Relative Extrusion	Use relative extrusion rather than absolute extrusion. Using relative E-steps makes for easier post-processing of the g-code. However, it's not supported by all printers and it may produce very slight deviations in the amount of deposited material compared to absolute E-steps. Irrespective of this setting, the extrusion mode will always be set to absolute before any g-code script is output.
	Experimental	
		Vertical tolerance in the sliced layers. The contours of a layer are normally generated by taking cross sections through the middle of each layer's thickness (Middle). Alternatively each layer can have the areas which fall inside of the

{slicing_tolerance}	Slicing Tolerance	volume throughout the entire thickness of the layer (Exclusive) or a layer has the areas which fall inside anywhere within the layer (Inclusive). Inclusive retains the most details, Exclusive makes for the best fit and Middle stays closest to the original surface.
{roofing_line_width}	Top Surface Skin Line Width	Width of a single line of the areas at the top of the print.
{roofing_pattern}	Top Surface Skin Pattern	The pattern of the top most layers.
{roofing_angles}	Top Surface Skin Line Directions	A list of integer line directions to use when the top surface skin layers use the lines or zig zag pattern. Elements from the list are used sequentially as the layers progress and when the end of the list is reached, it starts at the beginning again. The list items are separated by commas and the whole list is contained in square brackets. Default is an empty list which means use the traditional default angles (45 and 135 degrees).
{infill_enable_travel_optimization}	Infill Travel Optimization	When enabled, the order in which the infill lines are printed is optimized to reduce the distance travelled. The reduction in travel time achieved very much depends on the model being sliced, infill pattern, density, etc. Note that, for some models that have many small areas of infill, the time to slice the model may be greatly increased.
{material_flow_dependent_temperature}	Auto Temperature	Change the temperature for each layer automatically with the average flow speed of that layer.
{material_flow_temp_graph}	Flow Temperature Graph	Data linking material flow (in mm3 per second) to temperature (degrees Celsius).
{minimum_polygon_circumference}	Minimum Polygon Circumference	Polygons in sliced layers that have a circumference smaller than this amount will be filtered out. Lower values lead to higher resolution mesh at the cost of slicing time. It is meant mostly for high resolution SLA printers and very tiny 3D models with a lot of details.

{support_skip_some_zags}	Break Up Support In Chunks	Skip some support line connections to make the support structure easier to break away. This setting is applicable to the Zig Zag support infill pattern.
{support_skip_zag_per_mm}	Support Chunk Size	Leave out a connection between support lines once every N millimeter to make the support structure easier to break away.
{support_zag_skip_count}	Support Chunk Line Count	Skip one in every N connection lines to make the support structure easier to break away.
{draft_shield_enabled}	Enable Draft Shield	This will create a wall around the model, which traps (hot) air and shields against exterior airflow. Especially useful for materials which warp easily.
{draft_shield_dist}	Draft Shield X/Y Distance	Distance of the draft shield from the print, in the X/Y directions.
{draft_shield_height_limitation}	Draft Shield Limitation	Set the height of the draft shield. Choose to print the draft shield at the full height of the model or at a limited height.
{draft_shield_height}	Draft Shield Height	Height limitation of the draft shield. Above this height no draft shield will be printed.
{conical_overhang_enabled}	Make Overhang Printable	Change the geometry of the printed model such that minimal support is required. Steep overhangs will become shallow overhangs. Overhanging areas will drop down to become more vertical.
{conical_overhang_angle}	Maximum Model Angle	The maximum angle of overhangs after the they have been made printable. At a value of 0° all overhangs are replaced by a piece of model connected to the build plate, 90° will not change the model in any way.
{coasting_enable}	Enable Coasting	Coasting replaces the last part of an extrusion path with a travel path. The oozed material is used to print the last piece of the extrusion path in order to reduce stringing.
{coasting_volume}	Coasting Volume	The volume otherwise oozed. This value should generally be close to the nozzle diameter cubed.

{coasting_min_volume}	Minimum Volume Before Coasting	The smallest volume an extrusion path should have before allowing coasting. For smaller extrusion paths, less pressure has been built up in the bowden tube and so the coasted volume is scaled linearly. This value should always be larger than the Coasting Volume.
{coasting_speed}	Coasting Speed	The speed by which to move during coasting, relative to the speed of the extrusion path. A value slightly under 100% is advised, since during the coasting move the pressure in the bowden tube drops.
{cross_infill_pocket_size}	Cross 3D Pocket Size	The size of pockets at four-way crossings in the cross 3D pattern at heights where the pattern is touching itself.
{cross_infill_density_image}	Cross Infill Density Image	The file location of an image of which the brightness values determine the minimal density at the corresponding location in the infill of the print.
{cross_support_density_image}	Cross Fill Density Image for Support	The file location of an image of which the brightness values determine the minimal density at the corresponding location in the support.
{support_conical_enabled}	Enable Conical Support	Make support areas smaller at the bottom than at the overhang.
{support_conical_angle}	Conical Support Angle	The angle of the tilt of conical support. With 0 degrees being vertical, and 90 degrees being horizontal. Smaller angles cause the support to be more sturdy, but consist of more material. Negative angles cause the base of the support to be wider than the top.
{support_conical_min_width}	Conical Support Minimum Width	Minimum width to which the base of the conical support area is reduced. Small widths can lead to unstable support structures.
{magic_fuzzy_skin_enabled}	Fuzzy Skin	Randomly jitter while printing the outer wall, so that the surface has a rough and fuzzy look.
{magic_fuzzy_skin_outside_only}	Fuzzy Skin Outside Only	Jitter only the parts' outlines and not the parts' holes.
		The width within which to jitter. It's

{magic_fuzzy_skin_thickness}	Fuzzy Skin Thickness	advised to keep this below the outer wall width, since the inner walls are unaltered.
{magic_fuzzy_skin_point_density}	Fuzzy Skin Density	The average density of points introduced on each polygon in a layer. Note that the original points of the polygon are discarded, so a low density results in a reduction of the resolution.
{magic_fuzzy_skin_point_dist}	Fuzzy Skin Point Distance	The average distance between the random points introduced on each line segment. Note that the original points of the polygon are discarded, so a high smoothness results in a reduction of the resolution. This value must be higher than half the Fuzzy Skin Thickness.
{flow_rate_max_extrusion_offset}	Flow Rate Compensation Max Extrusion Offset	The maximum distance in mm to move the filament to compensate for changes in flow rate.
{flow_rate_extrusion_offset_factor}	Flow Rate Compensation Factor	How far to move the filament in order to compensate for changes in flow rate, as a percentage of how far the filament would move in one second of extrusion.
{wireframe_enabled}	Wire Printing	Print only the outside surface with a sparse webbed structure, printing 'in thin air'. This is realized by horizontally printing the contours of the model at given Z intervals which are connected via upward and diagonally downward lines.
{wireframe_height}	WP Connection Height	The height of the upward and diagonally downward lines between two horizontal parts. This determines the overall density of the net structure. Only applies to Wire Printing.
{wireframe_roof_inset}	WP Roof Inset Distance	The distance covered when making a connection from a roof outline inward. Only applies to Wire Printing.
{wireframe_printspeed}	WP Speed	Speed at which the nozzle moves when extruding material. Only applies to Wire Printing.
{wireframe_printspeed_bottom}	WP Bottom	Speed of printing the first layer, which is the only layer touching the

	Printing Speed	build platform. Only applies to Wire Printing.
{wireframe_printspeed_up}	WP Upward Printing Speed	Speed of printing a line upward 'in thin air'. Only applies to Wire Printing.
{wireframe_printspeed_down}	WP Downward Printing Speed	Speed of printing a line diagonally downward. Only applies to Wire Printing.
{wireframe_printspeed_flat}	WP Horizontal Printing Speed	Speed of printing the horizontal contours of the model. Only applies to Wire Printing.
{wireframe_flow}	WP Flow	Flow compensation: the amount of material extruded is multiplied by this value. Only applies to Wire Printing.
{wireframe_flow_connection}	WP Connection Flow	Flow compensation when going up or down. Only applies to Wire Printing.
{wireframe_flow_flat}	WP Flat Flow	Flow compensation when printing flat lines. Only applies to Wire Printing.
{wireframe_top_delay}	WP Top Delay	Delay time after an upward move, so that the upward line can harden. Only applies to Wire Printing.
{wireframe_bottom_delay}	WP Bottom Delay	Delay time after a downward move. Only applies to Wire Printing.
{wireframe_flat_delay}	WP Flat Delay	Delay time between two horizontal segments. Introducing such a delay can cause better adhesion to previous layers at the connection points, while too long delays cause sagging. Only applies to Wire Printing.
{wireframe_up_half_speed}	WP Ease Upward	Distance of an upward move which is extruded with half speed. This can cause better adhesion to previous layers, while not heating the material in those layers too much. Only applies to Wire Printing.
{wireframe_top_jump}	WP Knot Size	Creates a small knot at the top of an upward line, so that the consecutive horizontal layer has a better chance to connect to it. Only applies to Wire Printing.
		Distance with which the material falls down after an upward

{wireframe_fall_down}	WP Fall Down	extrusion. This distance is compensated for. Only applies to Wire Printing.
{wireframe_drag_along}	WP Drag Along	Distance with which the material of an upward extrusion is dragged along with the diagonally downward extrusion. This distance is compensated for. Only applies to Wire Printing.
{wireframe_strategy}	WP Strategy	Strategy for making sure two consecutive layers connect at each connection point. Retraction lets the upward lines harden in the right position, but may cause filament grinding. A knot can be made at the end of an upward line to heighten the chance of connecting to it and to let the line cool; however, it may require slow printing speeds. Another strategy is to compensate for the sagging of the top of an upward line; however, the lines won't always fall down as predicted.
{wireframe_straight_before_down}	WP Straighten Downward Lines	Percentage of a diagonally downward line which is covered by a horizontal line piece. This can prevent sagging of the top most point of upward lines. Only applies to Wire Printing.
{wireframe_roof_fall_down}	WP Roof Fall Down	The distance which horizontal roof lines printed 'in thin air' fall down when being printed. This distance is compensated for. Only applies to Wire Printing.
{wireframe_roof_drag_along}	WP Roof Drag Along	The distance of the end piece of an inward line which gets dragged along when going back to the outer outline of the roof. This distance is compensated for. Only applies to Wire Printing.
{wireframe_roof_outer_delay}	WP Roof Outer Delay	Time spent at the outer perimeters of hole which is to become a roof. Longer times can ensure a better connection. Only applies to Wire Printing.
		Distance between the nozzle and horizontally downward lines. Larger clearance results in

{wireframe_nozzle_clearance}	WP Nozzle Clearance	diagonally downward lines with a less steep angle, which in turn results in less upward connections with the next layer. Only applies to Wire Printing.
{adaptive_layer_height_enabled}	Use Adaptive Layers	Adaptive layers computes the layer heights depending on the shape of the model.
{adaptive_layer_height_variation}	Adaptive Layers Maximum Variation	The maximum allowed height different from the base layer height.
{adaptive_layer_height_variation_step}	Adaptive Layers Variation Step Size	The difference in height of the next layer height compared to the previous one.
{adaptive_layer_height_threshold}	Adaptive Layers Topography Size	Target horizontal distance between two adjacent layers. Reducing this setting causes thinner layers to be used to bring the edges of the layers closer together.
{wall_overhang_angle}	Overhanging Wall Angle	Walls that overhang more than this angle will be printed using overhanging wall settings. When the value is 90, no walls will be treated as overhanging. Overhang that gets supported by support will not be treated as overhang either.
{wall_overhang_speed_factor}	Overhanging Wall Speed	Overhanging walls will be printed at this percentage of their normal print speed.
{bridge_settings_enabled}	Enable Bridge Settings	Detect bridges and modify print speed, flow and fan settings while bridges are printed.
{bridge_wall_min_length}	Minimum Bridge Wall Length	Unsupported walls shorter than this will be printed using the normal wall settings. Longer unsupported walls will be printed using the bridge wall settings.
{bridge_skin_support_threshold}	Bridge Skin Support Threshold	If a skin region is supported for less than this percentage of its area, print it using the bridge settings. Otherwise it is printed using the normal skin settings.
{bridge_sparse_infill_max_density}	Bridge Sparse Infill Max Density	Maximum density of infill considered to be sparse. Skin over sparse infill is considered to be unsupported and so may be treated as a bridge skin.
		This controls the distance the

{bridge_wall_coast}	Bridge Wall Coasting	extruder should coast immediately before a bridge wall begins. Coasting before the bridge starts can reduce the pressure in the nozzle and may produce a flatter bridge.
{bridge_wall_speed}	Bridge Wall Speed	The speed at which the bridge walls are printed.
{bridge_wall_material_flow}	Bridge Wall Flow	When printing bridge walls, the amount of material extruded is multiplied by this value.
{bridge_skin_speed}	Bridge Skin Speed	The speed at which bridge skin regions are printed.
{bridge_skin_material_flow}	Bridge Skin Flow	When printing bridge skin regions, the amount of material extruded is multiplied by this value.
{bridge_skin_density}	Bridge Skin Density	The density of the bridge skin layer. Values less than 100 will increase the gaps between the skin lines.
{bridge_fan_speed}	Bridge Fan Speed	Percentage fan speed to use when printing bridge walls and skin.
{bridge_enable_more_layers}	Bridge Has Multiple Layers	If enabled, the second and third layers above the air are printed using the following settings. Otherwise, those layers are printed using the normal settings.
{bridge_skin_speed_2}	Bridge Second Skin Speed	Print speed to use when printing the second bridge skin layer.
{bridge_skin_material_flow_2}	Bridge Second Skin Flow	When printing the second bridge skin layer, the amount of material extruded is multiplied by this value.
{bridge_skin_density_2}	Bridge Second Skin Density	The density of the second bridge skin layer. Values less than 100 will increase the gaps between the skin lines.
{bridge_fan_speed_2}	Bridge Second Skin Fan Speed	Percentage fan speed to use when printing the second bridge skin layer.
{bridge_skin_speed_3}	Bridge Third Skin Speed	Print speed to use when printing the third bridge skin layer.
{bridge_skin_material_flow_3}	Bridge Third Skin Flow	When printing the third bridge skin layer, the amount of material extruded is multiplied by this value.
{bridge_skin_density_3}	Bridge Third Skin Density	The density of the third bridge skin layer. Values less than 100 will increase the gaps between the skin lines.

{bridge_fan_speed_3}	Bridge Third Skin Fan Speed	Percentage fan speed to use when printing the third bridge skin layer.
{clean_between_layers}	Wipe Nozzle Between Layers	Whether to include nozzle wipe G-Code between layers (maximum 1 per layer). Enabling this setting could influence behavior of retract at layer change. Please use Wipe Retraction settings to control retraction at layers where the wipe script will be working.
{max_extrusion_before_wipe}	Material Volume Between Wipes	Maximum material that can be extruded before another nozzle wipe is initiated. If this value is less than the volume of material required in a layer, the setting has no effect in this layer, i.e. it is limited to one wipe per layer.
{wipe_retraction_enable}	Wipe Retraction Enable	Retract the filament when the nozzle is moving over a non-printed area.
{wipe_retraction_amount}	Wipe Retraction Distance	Amount to retract the filament so it does not ooze during the wipe sequence.
{wipe_retraction_extra_prime_amount}	Wipe Retraction Extra Prime Amount	Some material can ooze away during a wipe travel moves, which can be compensated for here.
{wipe_retraction_speed}	Wipe Retraction Speed	The speed at which the filament is retracted and primed during a wipe retraction move.
{wipe_retraction_retract_speed}	Wipe Retraction Retract Speed	The speed at which the filament is retracted during a wipe retraction move.
{wipe_retraction_prime_speed}	Wipe Retraction Prime Speed	The speed at which the filament is primed during a wipe retraction move.
{wipe_pause}	Wipe Pause	Pause after the unretract.
{wipe_hop_enable}	Wipe Z Hop	When wiping, the build plate is lowered to create clearance between the nozzle and the print. It prevents the nozzle from hitting the print during travel moves, reducing the chance to knock the print from the build plate.
{wipe_hop_amount}	Wipe Z Hop Height	The height difference when performing a Z Hop.
{wipe_hop_speed}	Wipe Hop Speed	Speed to move the z-axis during the hop.

{wipe_brush_pos_x}	Wipe Brush X Position	X location where wipe script will start.
{wipe_repeat_count}	Wipe Repeat Count	Number of times to move the nozzle across the brush.
{wipe_move_distance}	Wipe Move Distance	The distance to move the head back and forth across the brush.
{small_hole_max_size}	Small Hole Max Size	Holes and part outlines with a diameter smaller than this will be printed using Small Feature Speed.
{small_feature_max_length}	Small Feature Max Length	Feature outlines that are shorter than this length will be printed using Small Feature Speed.
{small_feature_speed_factor}	Small Feature Speed	Small features will be printed at this percentage of their normal print speed. Slower printing can help with adhesion and accuracy.
{small_feature_speed_factor_0}	Small Feature Initial Layer Speed	Small features on the first layer will be printed at this percentage of their normal print speed. Slower printing can help with adhesion and accuracy.
Command Line Settings		
{center_object}	Center Object	Whether to center the object on the middle of the build platform (0,0), instead of using the coordinate system in which the object was saved.
{mesh_position_x}	Mesh Position X	Offset applied to the object in the x direction.
{mesh_position_y}	Mesh Position Y	Offset applied to the object in the y direction.
{mesh_position_z}	Mesh Position Z	Offset applied to the object in the z direction. With this you can perform what was used to be called 'Object Sink'.
{mesh_rotation_matrix}	Mesh Rotation Matrix	Transformation matrix to be applied to the model when loading it from file.

Extruder settings

The following settings are defined in fdmextruder.def.json, and are only settable per extruder

Machine		
{extruder_nr}	Extruder	The extruder train used for printing. This is used in multi-extrusion.

{machine_nozzle_id}	Nozzle ID	The nozzle ID for an extruder train, such as "AA 0.4" and "BB 0.8".
{machine_nozzle_size}	Nozzle Diameter	The inner diameter of the nozzle. Change this setting when using a non-standard nozzle size.
{machine_nozzle_offset_x}	Nozzle X Offset	The x-coordinate of the offset of the nozzle.
{machine_nozzle_offset_y}	Nozzle Y Offset	The y-coordinate of the offset of the nozzle.
{machine_extruder_start_code}	Extruder Start G-Code	Start g-code to execute when switching to this extruder.
{machine_extruder_start_pos_abs}	Extruder Start Position Absolute	Make the extruder starting position absolute rather than relative to the last-known location of the head.
{machine_extruder_start_pos_x}	Extruder Start Position X	The x-coordinate of the starting position when turning the extruder on.
{machine_extruder_start_pos_y}	Extruder Start Position Y	The y-coordinate of the starting position when turning the extruder on.
{machine_extruder_end_code}	Extruder End G-Code	End g-code to execute when switching away from this extruder.
{machine_extruder_end_pos_abs}	Extruder End Position Absolute	Make the extruder ending position absolute rather than relative to the last-known location of the head.
{machine_extruder_end_pos_x}	Extruder End Position X	The x-coordinate of the ending position when turning the extruder off.
{machine_extruder_end_pos_y}	Extruder End Position Y	The y-coordinate of the ending position when turning the extruder off.
{extruder_prime_pos_z}	Extruder Prime Z Position	The Z coordinate of the position where the nozzle primes at the start of printing.
{machine_extruder_cooling_fan_number}	Extruder Print Cooling Fan	The number of the print cooling fan associated with this extruder. Only change this from the default value of 0 when you have a different print cooling fan for each extruder.
	Build Plate Adhesion	
	Extruder	

{extruder_prime_pos_x}	Prime X Position	The X coordinate of the position where the nozzle primes at the start of printing.
{extruder_prime_pos_y}	Extruder Prime Y Position	The Y coordinate of the position where the nozzle primes at the start of printing.

Material

{material_diameter}	Diameter	Adjusts the diameter of the filament used. Match this value with the diameter of the used filament.