ANISOPRINT AURA USER MANUAL

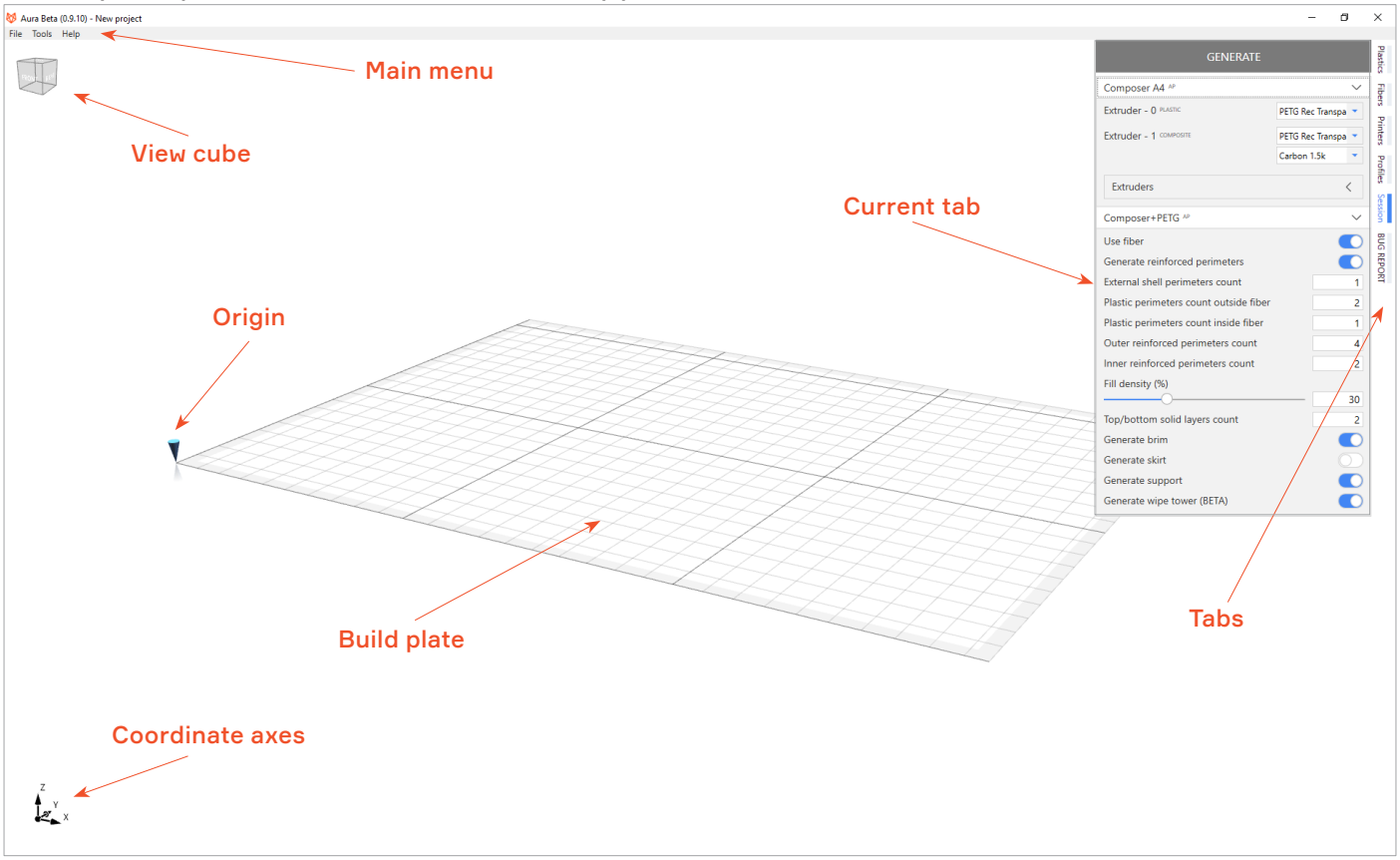
143026, Bolshoi bulvar 42-1, Skolkovo innovation center, Moscow, Russia
email: info@anisoprint.com

Starting Anisoprint Aura for the first time

After successful installation you can launch Aura for the first time. Just launch application on the last installation step or open it from your applications.

Main window

When you open Aura the main window appears.



Main window

First thing that you may want to do in Aura is to load 3D model of a part. Aura supports *.stl, *stp, *.3ds and *.obj file formats.

- To load model choose File > Open and then select a model in open file dialog. Also you can press Ctrl+M to call model open dialog.

After you have loaded model it appears on a build plate. Now you can save a project. Project is the combination of session settings (profile + printer + materials), models on a build plate and their layup settings. Project has *.auproj format.

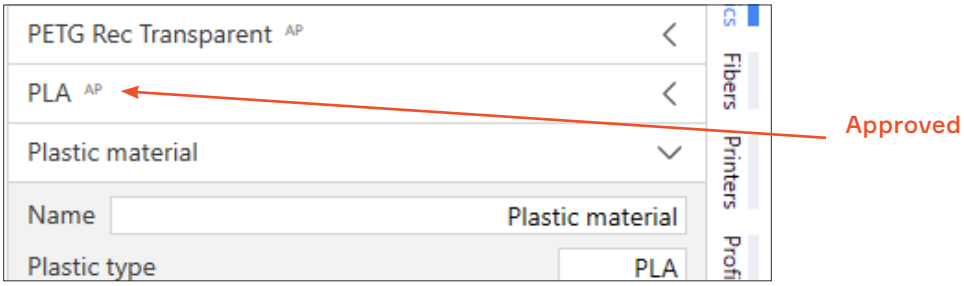
- To save project choose File > Save project as menu item, specify filename and press Save. Or click File > Save (Ctrl + S) to overwrite the current project.
- To load project choose File > Open project (Ctrl + P). If loaded project has conflicts with current session a conflict dialog will appear.

Settings

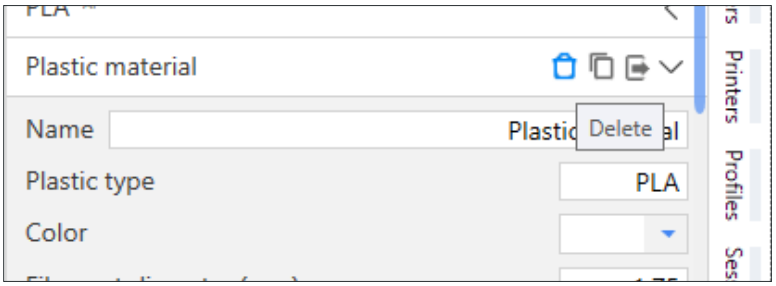
Aura has modular settings system. It means that you have 4 settings modules - plastics, fibers, printers and profiles. These modules are combined in a session, which contains all parameters required for slicing process. Session contains one printer, one or several plastic and fiber materials and one profile.

For each item you can set Name, Version and Description. Item names should be unique. Keep in mind that item name is important to compare settings in case of conflict.

- Some items have approved mark. APproved mark means that this settings are approved by Anisoprint. Approved items can't be modified, but it doesn't stop you from experiments. If you want to make any changes, you can copy the approved settings. You can find AP mark on the right of settings name in settings list.



- You can delete any non-approved (without AP mark) settings item. Just click on a delete button. If you are trying to remove item which is using by the session you will

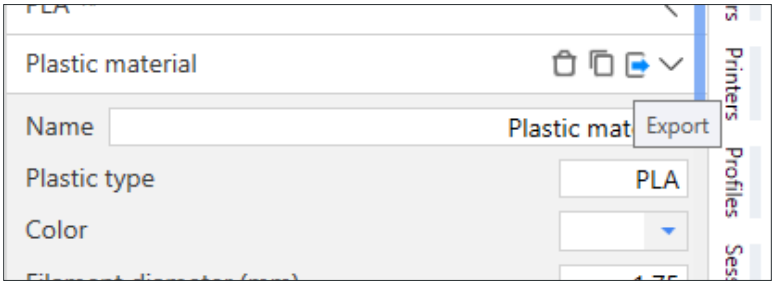


see a warning.

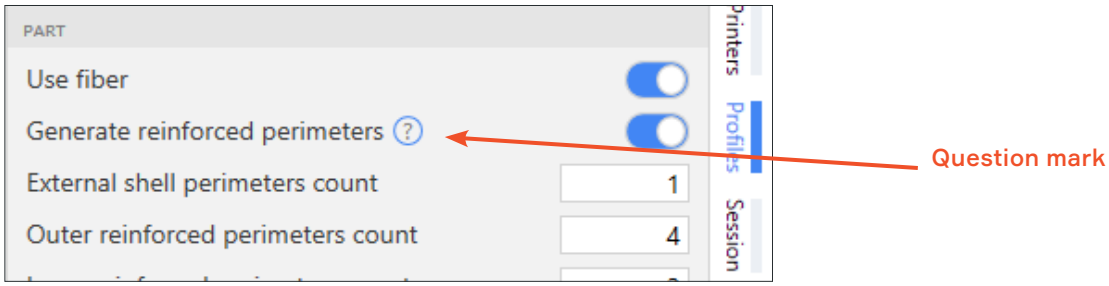
- You can duplicate any settings item. Just click on a duplicate button. Duplication

creates a full copy of your settings item.

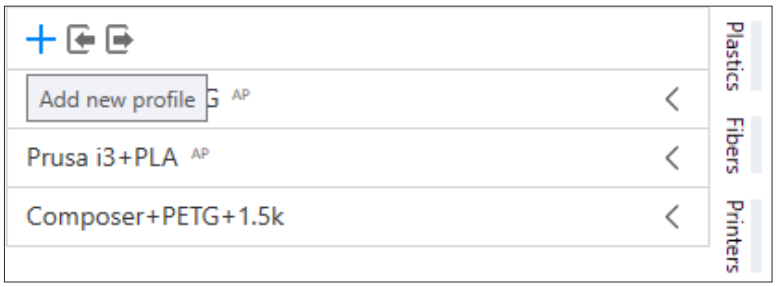
- You can export any settings item. Just click on a export button. Each settings type has



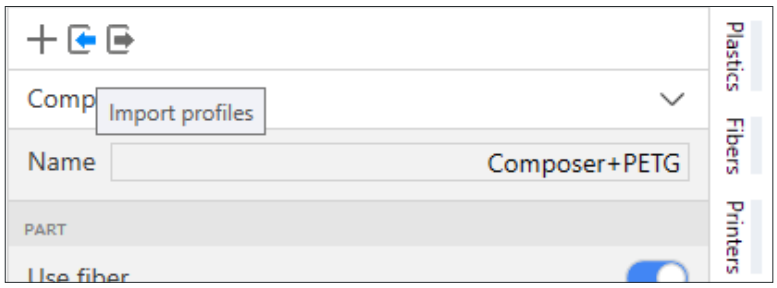
- You can find information about any parameter by clicking on a question mark on the right from the parameter name.



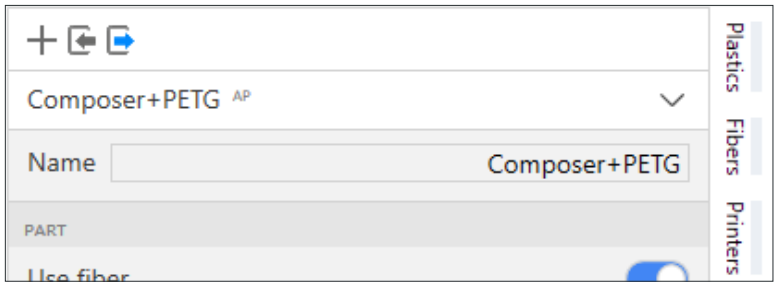
- You can add new item to list. Just click on an add button, and you will see a new item in a list. Item will be filled with default values.



- You can import a set of settings. Just click on a import button.



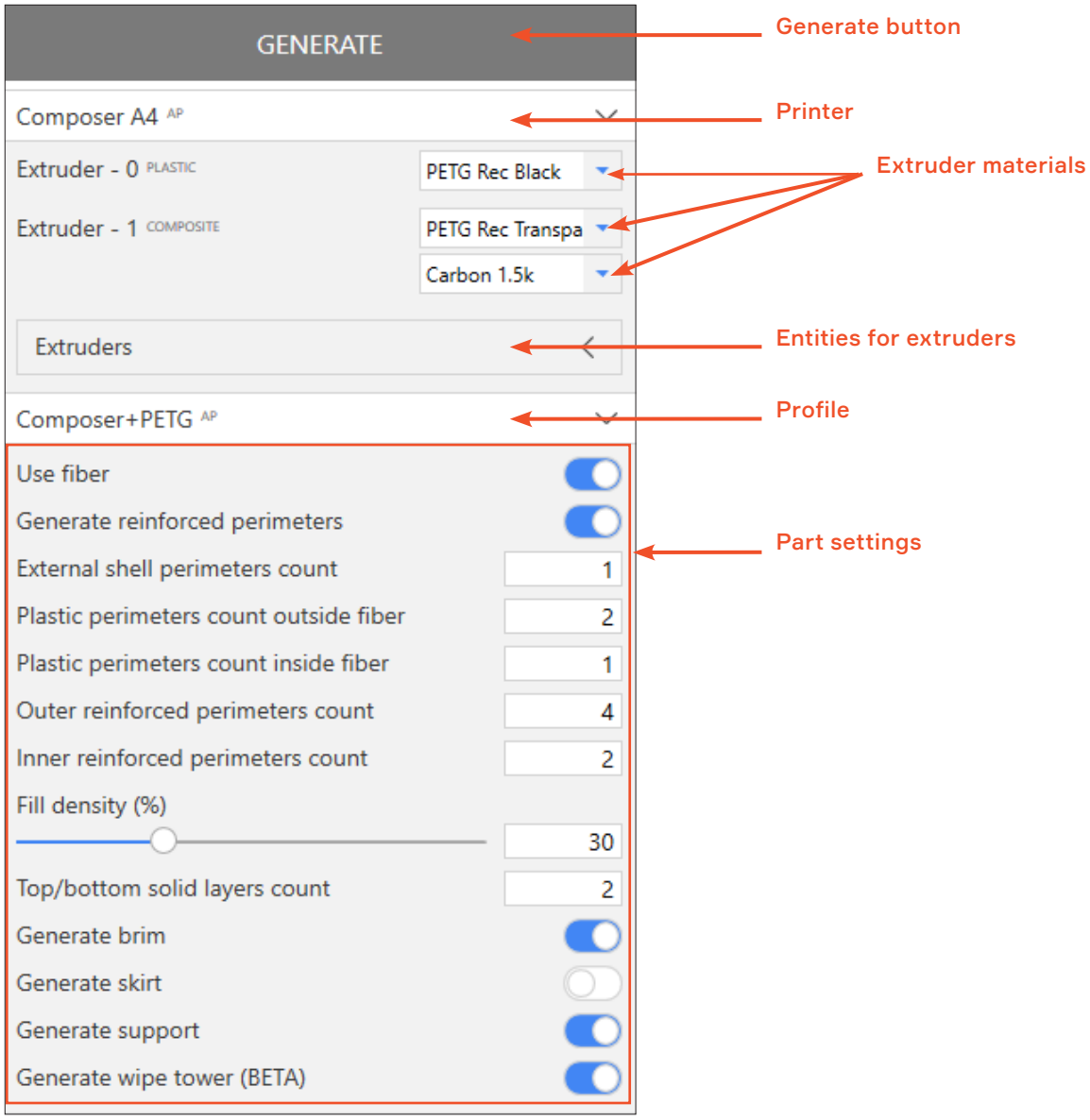
- You can export all settings from your list . Just click on a export button.



Session

Session is the settings aggregator. You should specify printer, profile, materials in extruders, and choose extruder for each print entity. The part settings that you can see at the bottom of session panel can be adjusted individually for each print. Besides, this settings are not taken into account during profile comparison and conflict determination. So if you load a project from a file, this settings are always overrrided without conflict.

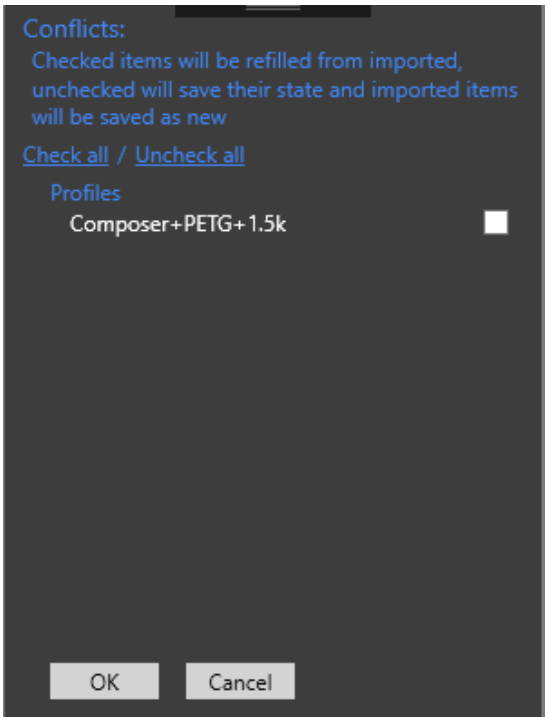
Session panel



After you've configurated session you can press a generate button. If the generate button is disabled check that you have loaded model for slicing and your models have no intersection between themselves.

Conflicts

Conflicts dialog emerges if you have conflict with at least one settings item in your project (material or printer or profile). Conflict means that you already have in your Aura settings an item with the same name but with differing settings. For example, you already have Plastics material named “PLA” and flow multiplier = 1. And you want to load the project which contains material “PLA”, and this material has flow multiplier = 1.1. So, this is the conflict and you should solve it before you can continue to work with slicer. You can do it with the Conflict dialog.

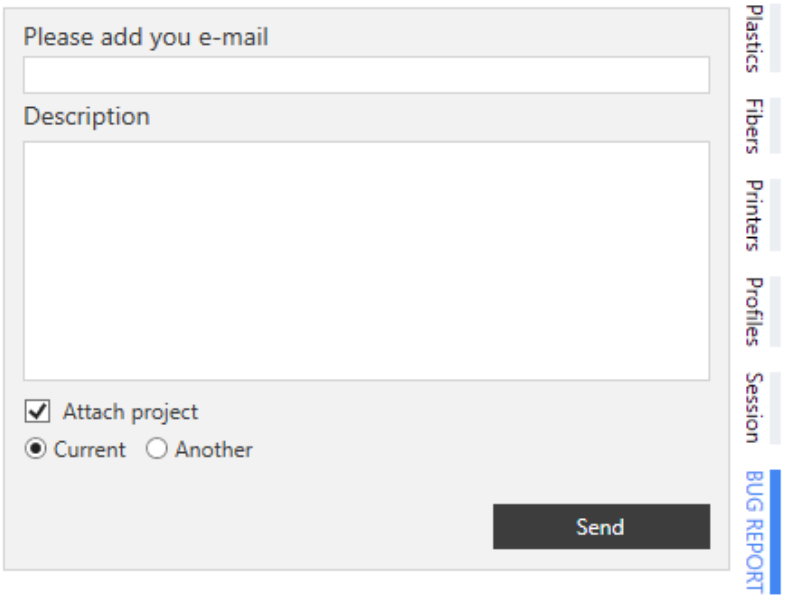


Conflicts dialog

- To override your local settings check a checkbox for appropriate settings.
- To add the settings from the project as a copy uncheck a checkbox for appropriate settings.

Bug report

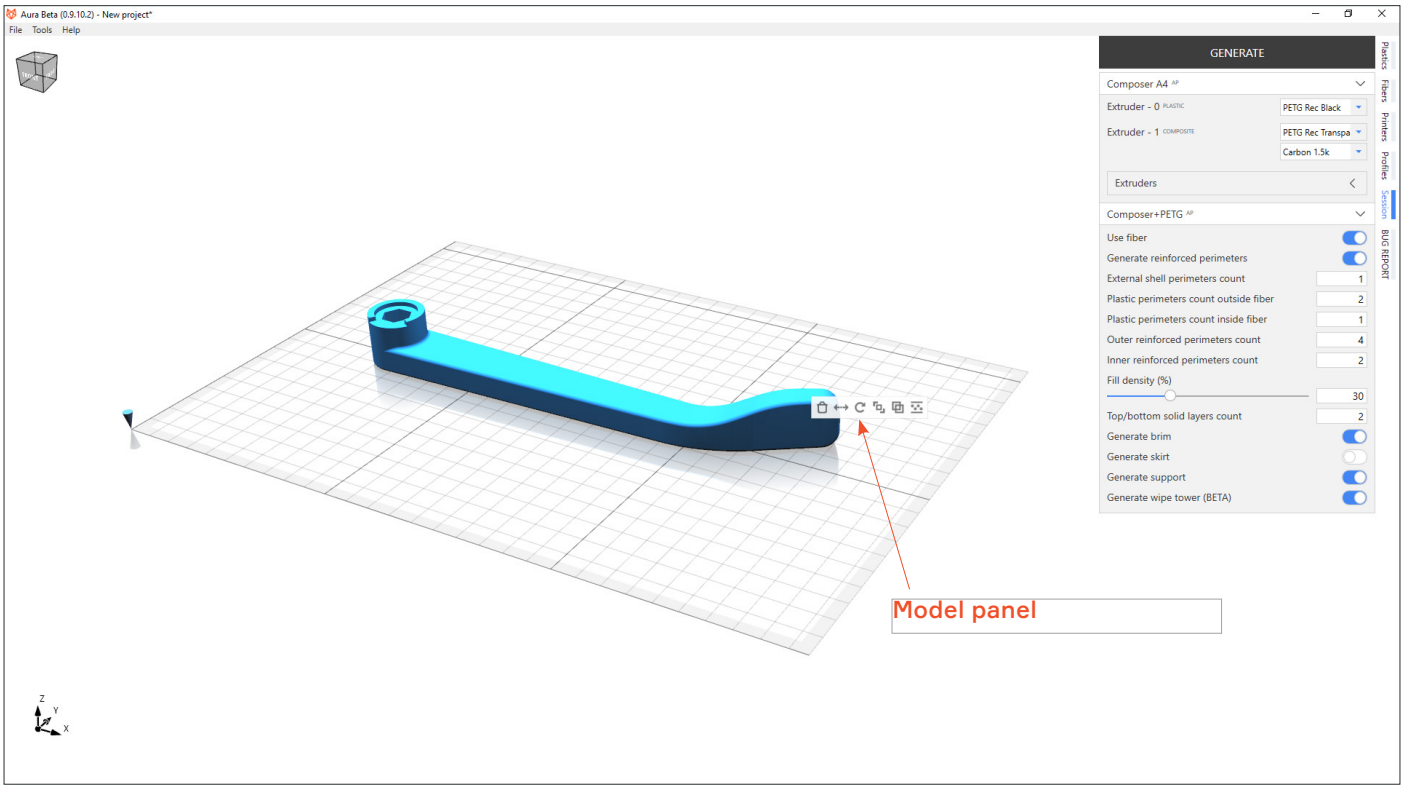
If you've encountered bug while working with Aura or you have any suggestion for Aura improvements, please send us a bug report. You can do this from BUG REPORT panel. Please describe a bug and please attach the project that produce this bug. Full bug description and attachment of project that causes the bug will help us to fix it as soon as possible. If you have a question or you want to attach more files, please send it to support@anisoprint.com.



Bug report form

Model panel

After you've loaded a model you will see it on a build plate placed at the build plate center. You can select a model by double clicking on it. Double click on empty space will deselect a model. Double clicking on a part calls model panel.



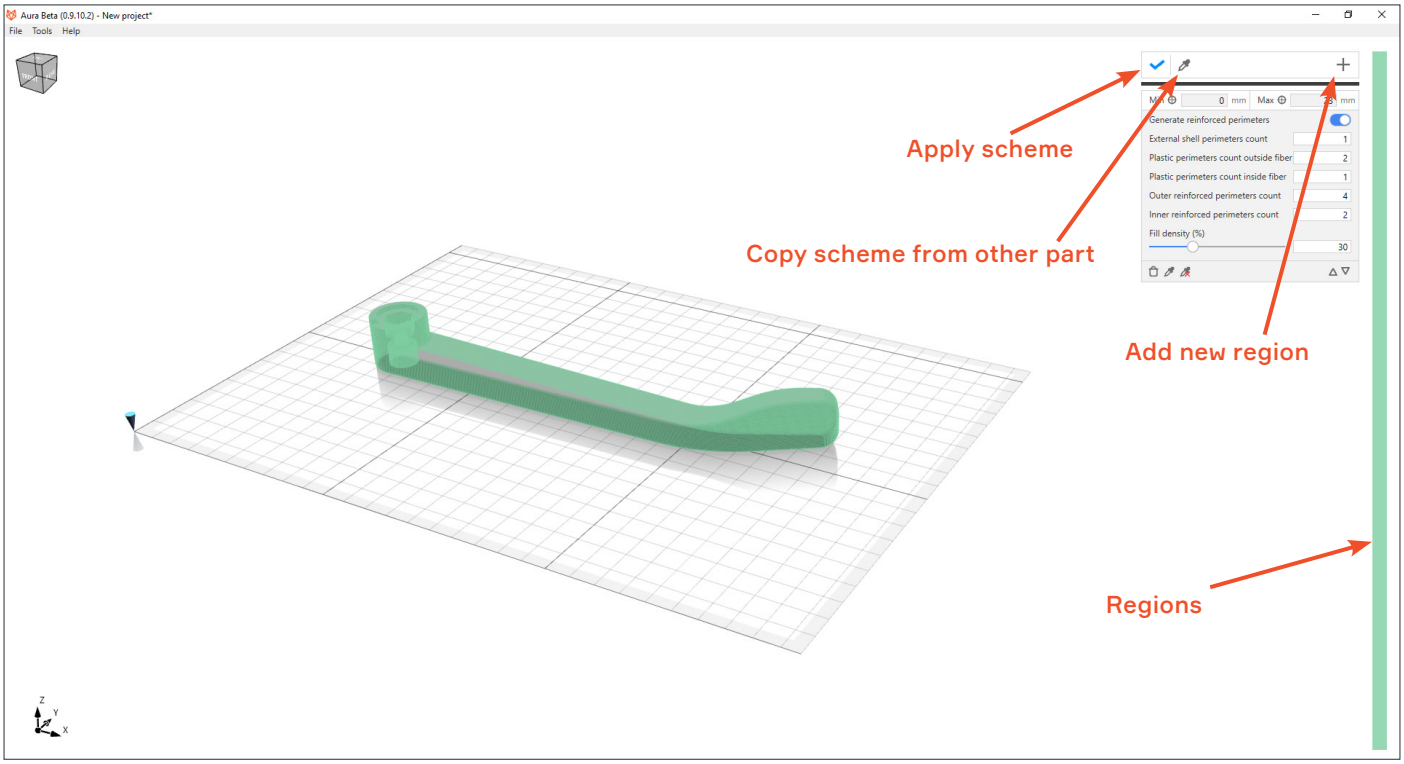
Model panel

- To delete a part from a build plate press a delete button.
- To move a model on build plate surface simply press a move button on the model panel. After that you can simply drag an arrows to shift a part along X and Y axes. Or you can manualy specify X and Y offsets in corresponding textboxes which are located at the bottom of a main window. After a part is placed as needed, press Apply or Enter button to finish movement. Or press Cancel or Esc to cancel movement.
- To rotate a part around it's center press a rotate button on the model panel. After that you can simply rotate a model using an object manupulator. Or you can manualy specify angles in correstponding textboxes which are located at bottom of a main window. Press Apply or Enter button to finish rotation. Press Cancel or Esc to cancel rotation. After you've finished rotation a part will be placed again on a build plate.
- To resize a part you can press a resize button on the model panel. After that you can input scale values in corresponding textboxes. Press Apply or Enter to finish resizing or click Cancel or Esc to cancel resizing.
- To clone a part you can press a clone button on the model panel. After that a new part will be created and placed in the build plate center.
- To set layer structure scheme press a set layer structure button.

Adjusting part internal structure

You may want to have non-homogeneous part structure. For example, you may want to reinforce with fiber only bottom and top layers to obtain maximum bending stiffness with small amount of fiber. Or you may want to renforce the bottom layers more than the top. Generally, if you have specific requirements about how the part should be reinforced at different levels, you may achieve the result with layers structure configuration.

After you press a set layer structure button on the model panel, the window changes it's state to layer structure settings mode.

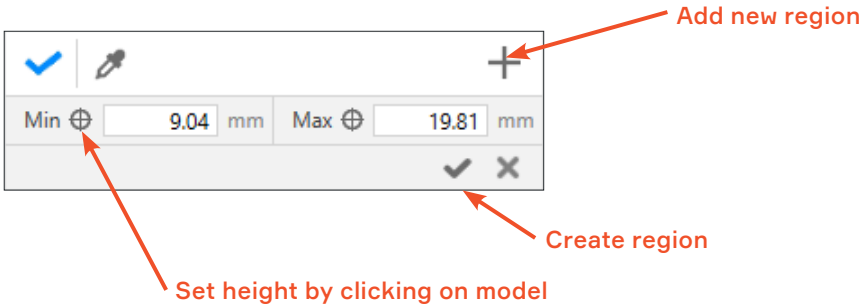


Layers struct panel

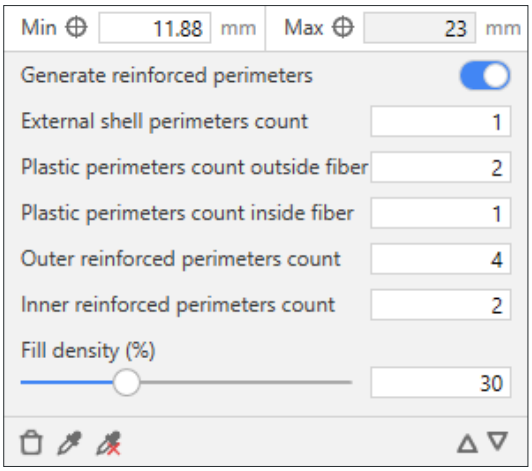
You may copy layer scheme from another model. Just press a button with picker icon and then click on a model from which you want to apply layers structure.

To observe region details and edit region just select it from Regions panel. Configure it's settings as you need. Keep in mind that regions with same color have same properties and they are connected between themselves. It means, that changes made to one regions will affect other regions with the same color.

- To add region click on a new region button or click and drag on regions panel. Arter that you can specify lower and upper bounds of the region and press Create button.



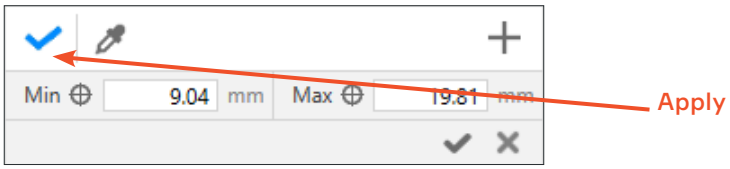
Region creation panel



Region settings panel

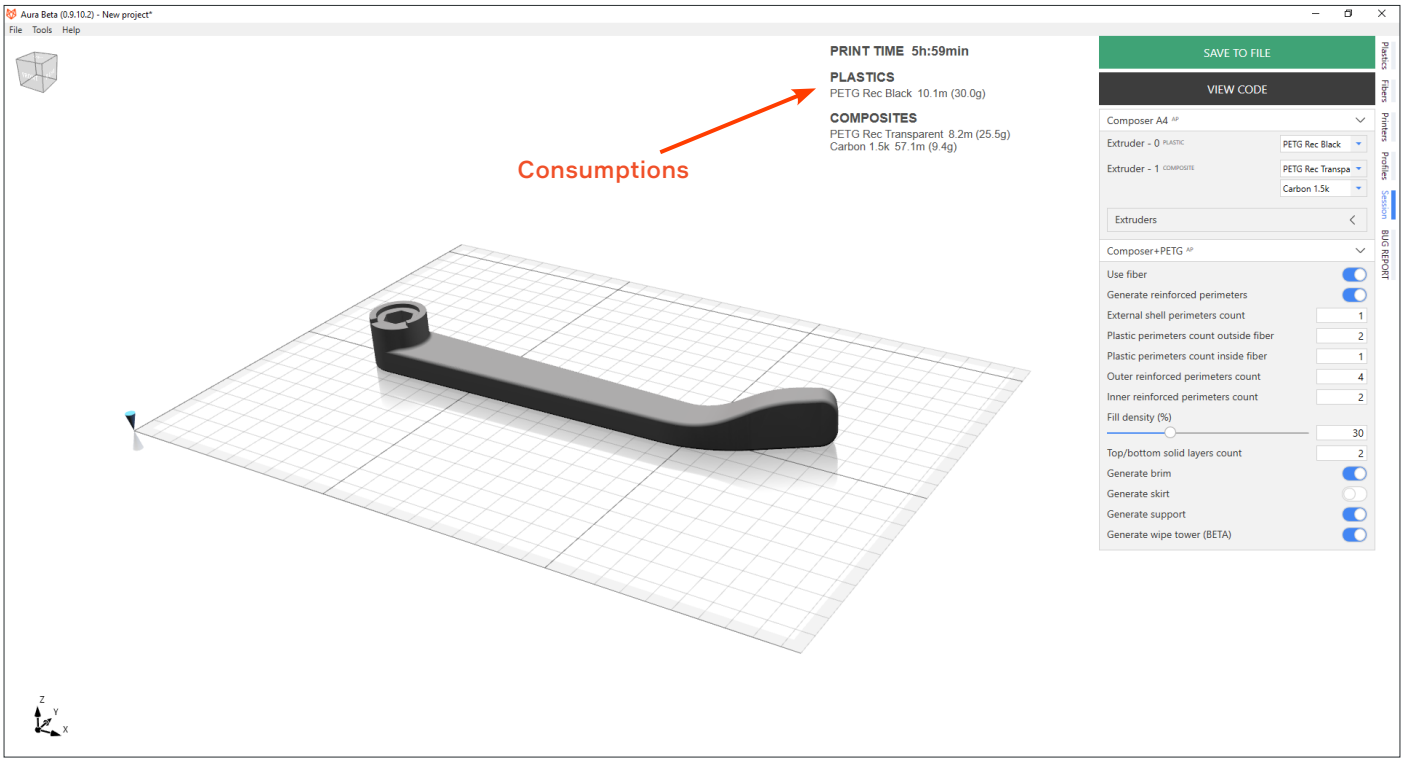
- To delete region click on a delete button. Notice that you can't remove the last region. You scheme should contain at least one region.
- To link region to another click on a button. This opperation fills selected region with the settings from picked region.
- To unlink region click on a button. This operation doesn't change current region settings, but now all changes made to other regions will have no effect on this region and all changes in this region will have no effect on ex-linked regions.
- To switch between regions use buttons. This buttons are only for navigation between regions.

After you have finished to configure layup scheme don't forget to apply changes by pressing Apply button.



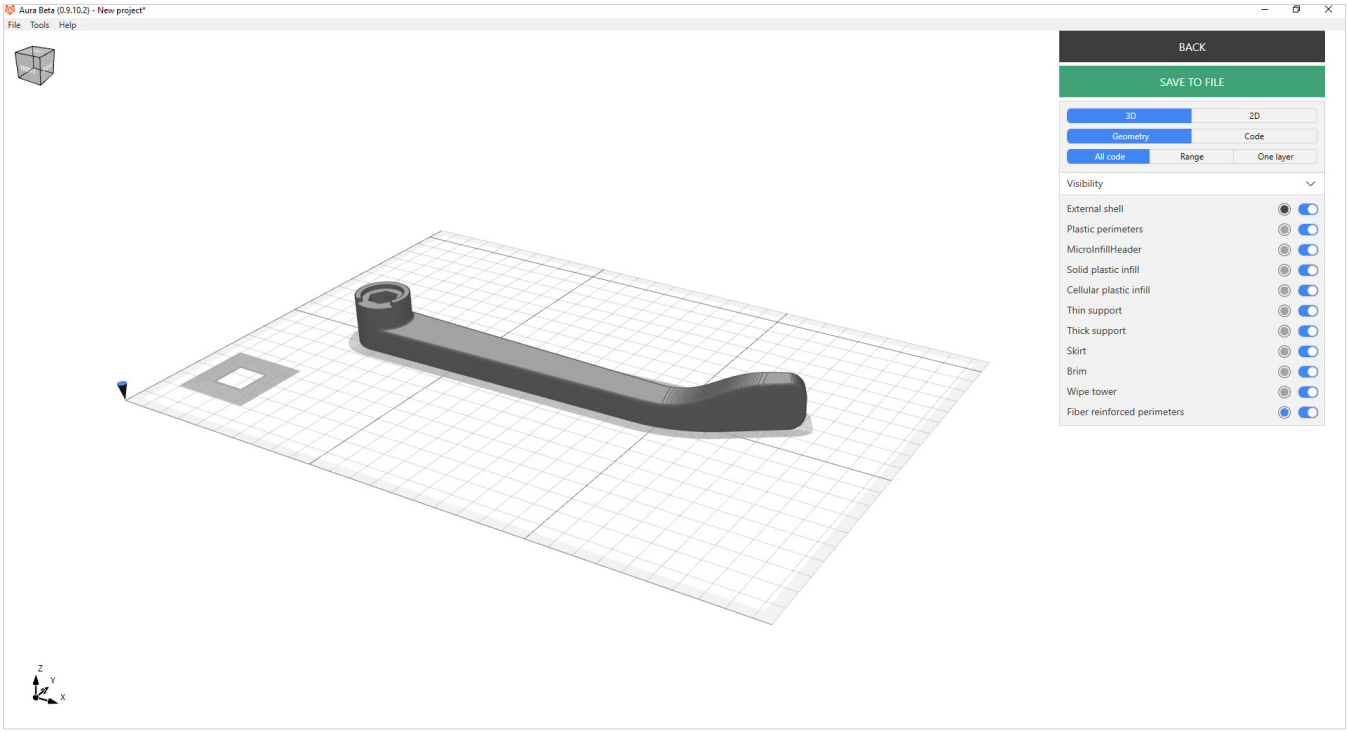
Code generation

After you've configured models and a printing session as you want, press Generate button to generate geometry and translate in to machine code. During process you can Cancel generation with lose of all progress. When generation is successfully finished the printing time and materials consumption information will appear. Materials consumplion is calculated by length and mass. You can Save to file machine code or open code viewer by pressing View code button.

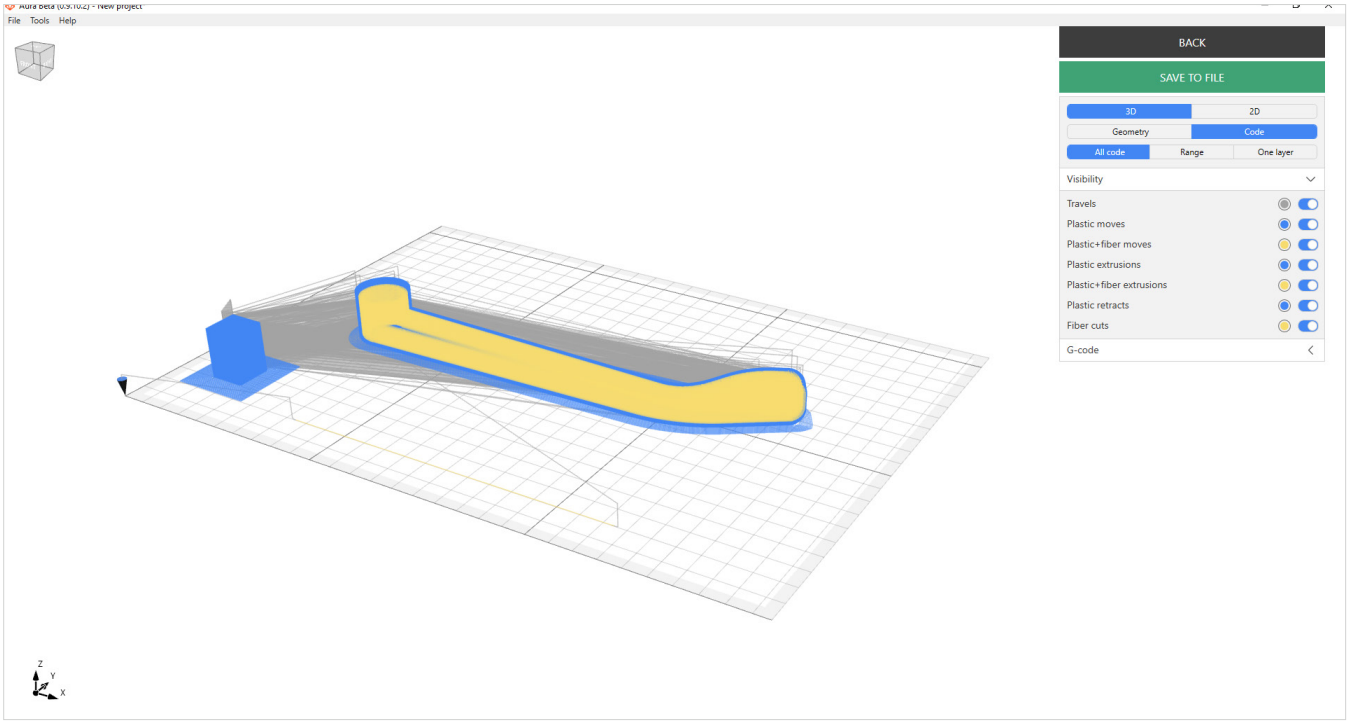


Geometry-Code viewer

Aura has two print process visualizers - geometry and code explorers. Geometry view helps you to understand how a part will be printed while code explorer shows exact print head trajectories. Geometry viewer shows entities such as external shell, plastic perimeters, micro, solid and cellular infills, supports, fiber perimeters and etc. Code viewer represents entities such as travels, moves, pure extrusions, retracts, moves with fiber etc. In both modes you can switch between 3D and 2D mode, you can observe all layers, layer range or one layer. And you can show or hide individual entities and change their colors. In code mode you can also display and read G-code, in future Aura versions we will add possibility to select G-code lines and highlight appropriate code blocks on the model.



Geometry viewer



Code viewer

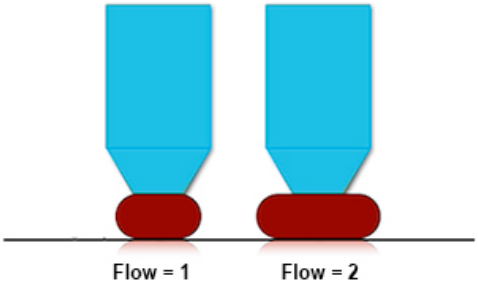
Settings (extended). Plastic.

Filament diameter

The filament diameter of plastic thread is measured in cross section. Usually it is 1.75mm or 3mm. You can find this value on the package of plastic filament.

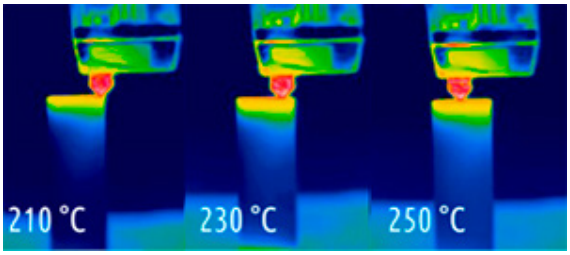
Flow multiplier

The flow multiplier is a correction factor of plastic extrusion. Vary this parameter if all parts have under- or overextrusion.



Extruder temperature (°C)

The extruder temperature is the temperature at which plastic will be melted for extruding from a nozzle. You can find an appropriate temperature range on the plastic filament package.

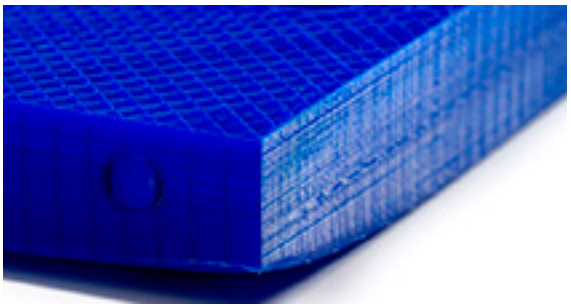


Standby extruder temperature (°C)

The standby extruder temperature is the extruder temperature at which it will be in an inactive state (another extruder is active, i.e. is printing). It should be as close as possible to printing temperature for faster heatup and print start, but low enough to prevent plastic free flow out from an extruder by gravity.

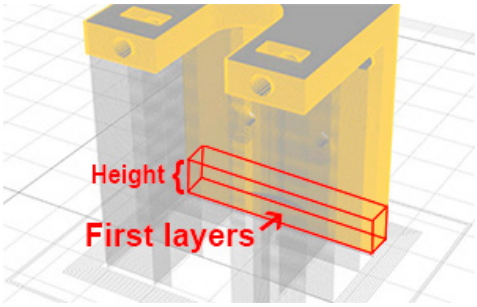
Build plate temperature (°C)

The build plate temperature is the constant temperature of a build plate throughout all printing process except printing first layers. Set this value high enough so a part won't detache and edges won't bend.



First layers temperatures height (mm)

The first layers temperatures height is the height below which extruders and a table have special heating rules.



Extruder temperature on first layers (°C)

The extruder temperature on first layers is the extruder printing temperature which is kept constant only on first layers - below the height defined in settings.

Build plate temperature on first layers (°C)

The build plate temperature on first layers is the temperature of a build plate which is kept constant only on first layers below height defined in settings. It should be high enough for a part to immediately and securely glue to a table.

Enable print cooling

Enable print cooling is the flag which enables fans to cool the part. For best results, turn on cooling for any print.

Regular fan speed (%)

The regular fan speed is the fan speed for cooling during whole printing process except printing first layers. Indicated as percentage. Choose high enough for saving part geometry undisturbed, but not too high so layers don't start losing good adhesion between themselves.

Maximum fan speed (%)

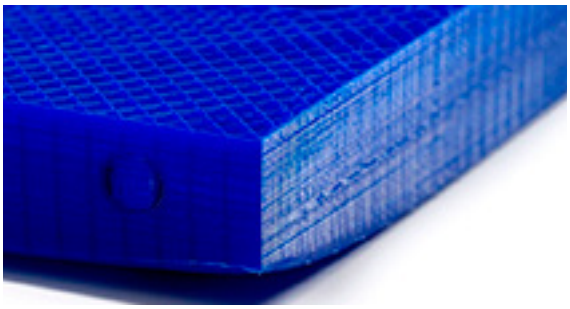
The maximum fan speed is the fan speed for cooling which is used when layer time is less than minimal layer printing time (specified in settings). Indicated as percentage. Choose maximum speed to have no form meltings and geometry distorsions.

Layer time for max fan speed (sec)

The layer time for max fan speed is the time when edges of a part start melting and the part itself starts losing its shape with a regular fan speed.

First layers fan speed height (mm)

The first layers fan speed height is the height below which cooling rules are specific. Choose minimum but sufficient height so that a part is securely glued to a build plate and part won't lose its shape due to bed heating.



First layers fan speed (%)

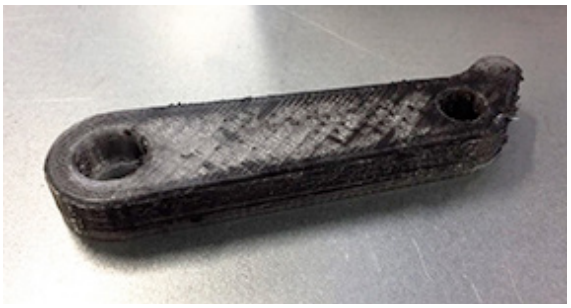
Choose the fan speed low enough for better adhesion to a bed and to prevent geometry changes due to bed heating.

Retraction length (mm)

The retraction length is the length of plastic reverse movement in a feeder. Choose it long enough length to stop pressure onto plastic in an extruder chamber. Keep in mind that during retract, the extruder stays still, so the larger is the retraction length, the longer it stays without movement, which can lead to part melting by the extruder, and leakings for runny plastics.

Retraction length on tool change (mm)

The retraction length on tool change is the length of plastic reverse movement in a feeder. Choose maximum length, taking into account the fact that plastic won't be fed into extruder for a long time and it is necessary to prevent plastic leakings due to gravity.

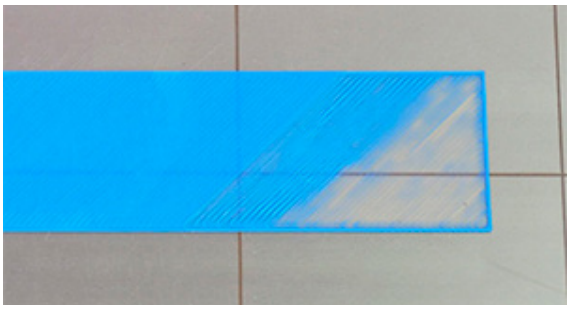


Retraction minimum travel (mm)

The retraction minimum travel is the sufficient travel length before retraction. In case of short travels it makes no sense to stop for retraction because plastic has not enough time to spill out uncontrolled. Choose this value to maximum, but be aware of plastic leaks while travels.

Extra unretract length (mm)

The extra unretract length is the value additional to retract length, when unretracting compensates for retracting. If on print resuming a portion of plastic has spilled from an extruder, then volume of plastic in a nozzle becomes less than what it was before travel. If you have noticed for this plastic (especcially for runny) that at the beginning of print there is an underextrusion migrating to normal extrusion, then you shoud set not null for this parameter.



Retract speed (mm/sec)

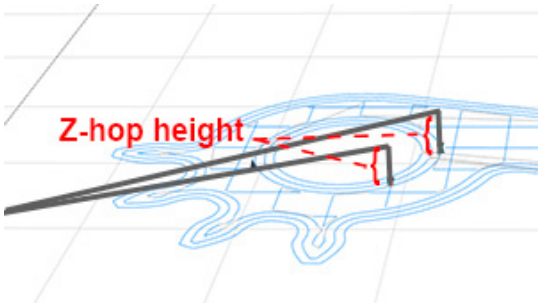
The retract speed is the speed of reverse movement of plastic in a feeder. Choose maximum for fast printing restart, however, keep in mind that some feeders can't handle fast feeding speed.

Unretract speed (mm/sec)

The unretract speed is the speed of direct movement of plastic in a feeder. Choose maximum for fast printing restart, however, keep in mind that some feeders can't handle fast feeding speed.

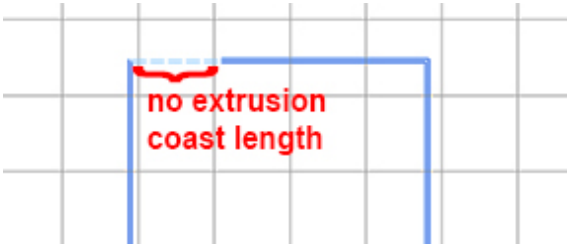
Z-hop height (mm)

The Z-hop height is the extruder lift up height when moving between print areas. Choose minimum to make the printing process faster, but enough to prevent plastic drops on a part shell. Z-hop may lead to formation of strings, however, it is usually easier remove strings than beads.



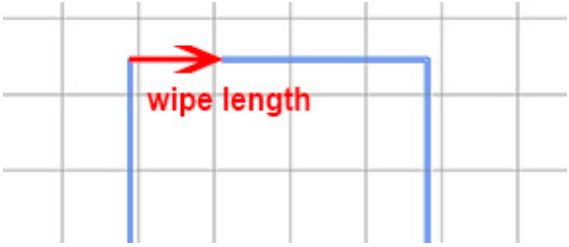
Coast length (mm)

The coast length is the length of the last polygon segment which is printed without plastic extrusion. Plastic goes out only by gravity on this coast segment. Choose long enough to stop free plastic spilling after retract, however, keep an eye on underextrusion.



Wipe nozzle length (mm)

The wipe nozzle length is the length of a reverse print head movement. The print head makes a reverse movement after a retract to wipe spilling plastic on already printed lines. Choose long enough to stop free plastic spilling after a retract.



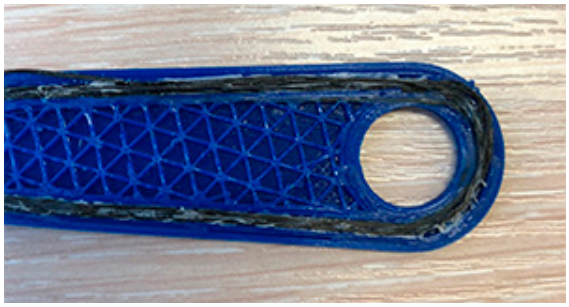
Fiber

Fiber diameter (mm)

The fiber diameter is the average diameter of a fiber thread measured in cross section. For the Anisoprint CCF-1.5k this value is 0.34mm.

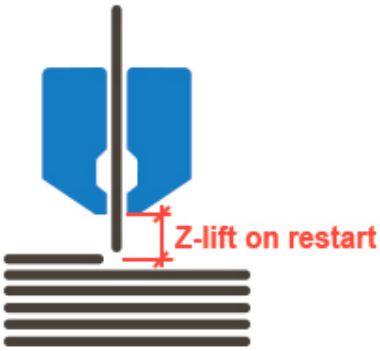
Plastic flow multiplier

The plastic flow multiplier is a factor by which amount of extruded plastic in a composite extruder is multiplied. Vary this parameter if all composite polygons have under- or overextrusion.



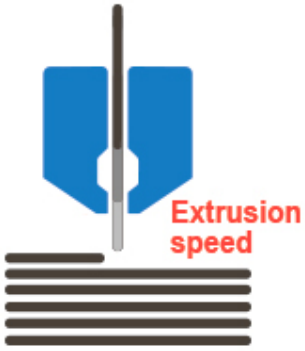
Z-lift on restart (mm)

Z-lift on restart is the height at which a composite extruder is raised to put out a fiber tail and start to print a composite polygon.



Fiber extrusion speed (mm/sec)

Fiber extrusion speed is the speed of the fiber tail extrusion.



Restart pause (sec)

Restart pause is the delay before the start of composite polygon printing. This is needed for the good adhesion between a fiber tail and an underlying layer.

Finish ironing distance (mm)

Finish ironing distance is the length of travel which is performed by a composite nozzle at the end of fiber polygon in the direction of last segment. It happens after cut and full exit of a fiber tail. This movement is needed to iron fiber tail which will be fluffy otherwise.

Do plastic retract

The do plastic retract flag is the flag which indicates that plastic retract should be performed during composite printing.

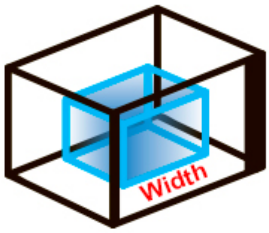
Linear density (tex = g/km)

Linear density is the mass of 1km of fiber in grams.

Printers

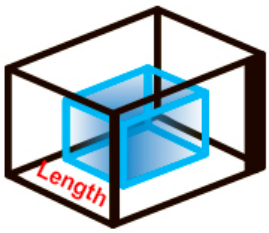
Width (mm)

The width is the size of the build area along the X axis.



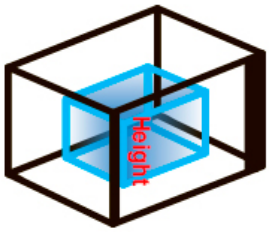
Length (mm)

The length is the size of the build area along the Y axis.



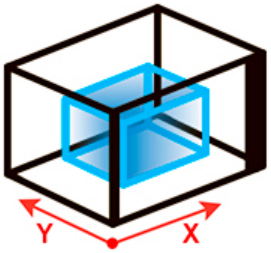
Height (mm)

The height is the size of the build area along the Z axis.



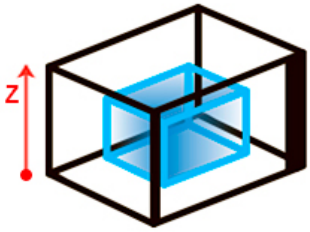
X/Y travel speed (mm/sec)

The X/Y travel speed is the linear speed of print head movements in XY plane without extrusions.



Z travel speed (mm/sec)

The Z travel speed is the linear speed of print head movements in the Z direction without extrusion.

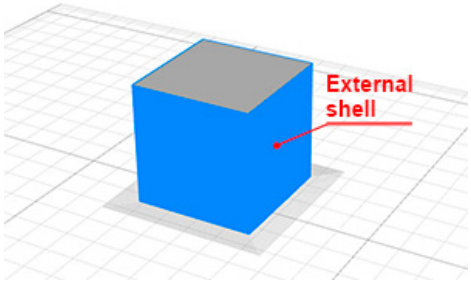


Enable acceleration control

The enable acceleration control is the flag of manual control over acceleartions.

External shell acceleration (mm/sec²)

The external shell acceleration is the acceleration which is used only for external shell printing. It has effect on travels and moves with extrusion.



Regular acceleartion (mm/sec²)

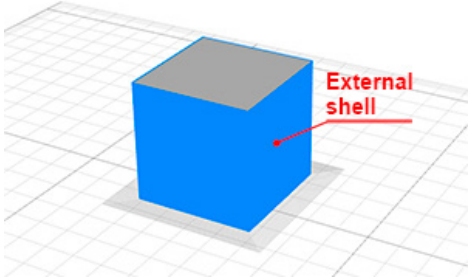
The regular acceleration is the acceleration which is used for all printing areas except an external shell. It has effect on travels and moves with extrusions.

Enable jerk control

The enable jerk control is the flag of manual control over jerks.

External shell jerk (mm/sec)

The external shell jerk is the jerk value which is used only for external shell printing. It has effect on travels and moves without extrusion.

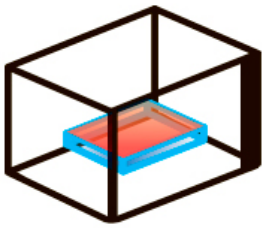


Regular jerk (mm/sec)

The regular jerk is the jerk which is used for all printing areas except an external shell. It has effect on travels and moves with extrusion.

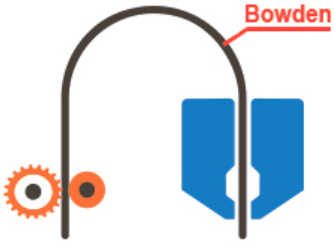
Has heated build plate

The has heated table flag is the flag which indicates if a printer has heatable table.



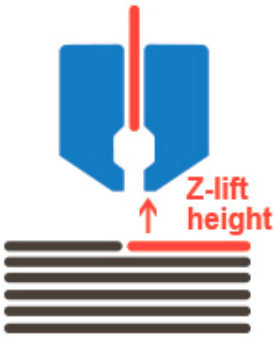
Additional retract length (mm)

The additional retract length is the additional length of plastic reverse movement in a feeder which will be added to the retract length value for used plastics. Use this parameter if your printer has a bowden tube for plastic feeding, because in this case retraction inertia may appear.



Z-lift on tool change (mm)

The Z-lift on tool change is the height at which a print head raises for tool change.



Start G-code

The start G-code is the code which is executed at the start of print process.

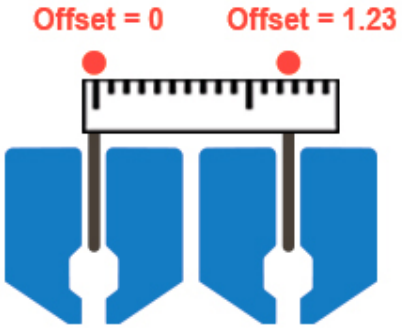
End G-code

The end G-code is the code which is executed at the end of print process.

Extruder (Plastic)

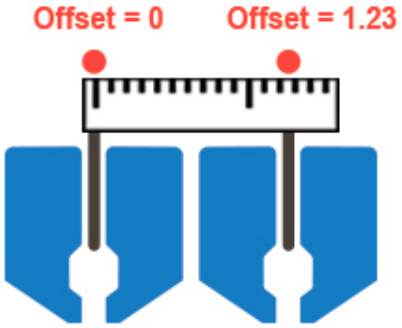
X offset (mm)

X offset is the extruder X offset relative to the base point on a print head. If your printer has only one extruder then set this parameter to zero. If printer firmware doesn't set offset by itself then you should set other extruders offsets relative to base (usually first) extruder. In this case you sholud set offsets for this base extruder offsets to zero.



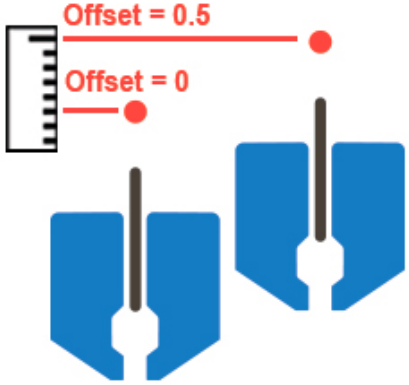
Y offset (mm)

Y offset is the extruder Y offset relatively to the base point on a print head. If your printer has only one extruder then set this parameter to zero. If printer firmware doesn't set offset by itself then you should set other extruders offsets relative to base (usually first) extruder. In this case you sholud set offsets for this base extruder offsets to zero.



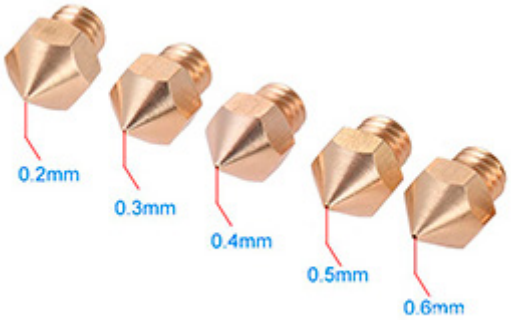
Z offset (mm)

Z offset is the extruder Z offset relatively to the base point on a print head. If your printer has only one extruder then set this parameter to zero. If printer firmware doesn't set offset by itself then you should set other extruders offsets relative to base (usually first) extruder. In this case you sholud set offsets for this base extruder offsets to zero.



Nozzle diameter (mm)

The nozzle diameter is the diameter of the outlet hole of a nozzle.



Has cooling fan

The has cooling fan flag is the flag which indicates thar extruder has a nozzle cooling fan.

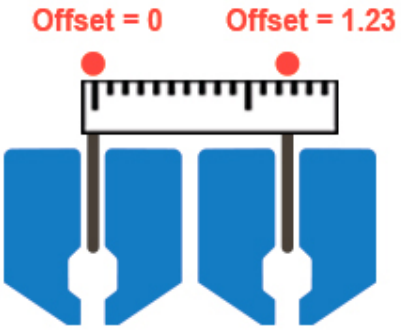
Cooling fan index

The cooling fan index is the number of this cooling fan in printer firmware.

Extruder (Composite)

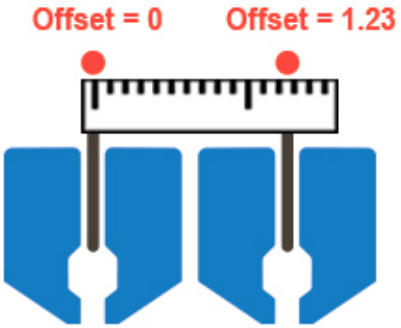
X offset (mm)

X offset is the extruder X offset relatively to the base point on a print head. If your printer has only one extruder then set this parameter to zero. If printer firmware doesn't set offset by itself then you should set other extruders offsets relative to base (usually first) extruder. In this case you sholud set offsets for this base extruder offsets to zero.



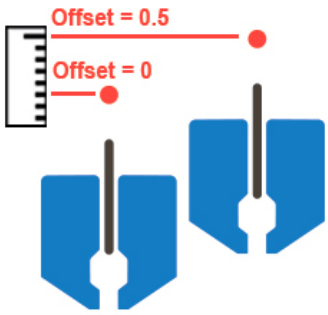
Y offset (mm)

Y offset is the extruder Y offset relatively to the base point on a print head. If your printer has only one extruder then set this parameter to zero. If printer firmware doesn't set offset by itself then you should set other extruders offsets relative to base (usually first) extruder. In this case you sholud set offsets for this base extruder offsets to zero.



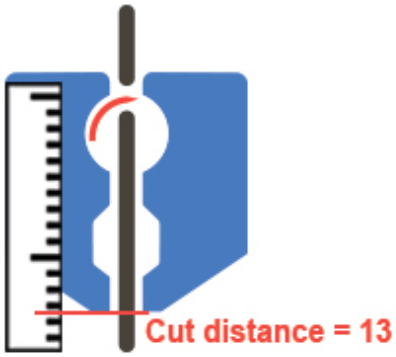
Z offset (mm)

Z offset is the extruder Z offset relatively to the base point on a print head. If your printer has only one extruder then set this parameter to zero. If printer firmware doesn't set offset by itself then you should set other extruders offsets relative to base (usually first) extruder. In this case you sholud set offsets for this base extruder offsets to zero.



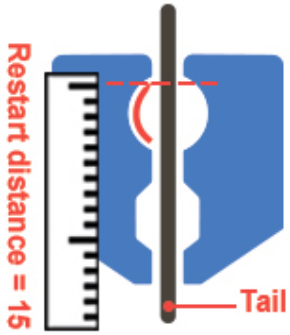
Cut distance (mm)

The cut distance is the distance from a cut point to outlet hole of a nozzle. This value should be set only by printer manufacturer.



Fiber restart length (mm)

The fiber restart length is the length of fiber extrusion during restart. This is necessary for the fiber tail to get out of the nozzle. This parameter in general should equal to the cut distance.



Cut G-code

The cut G-code is the code, that actuates fiber cutting. This value should be set only by printer manufacturer.

Has cooling fan

The has cooling fan flag is the flag which indicates thar extruder has a nozzle cooling fan.

Cooling fan index

The cooling fan index is the number of this cooling fan in printer firmware.

Profile

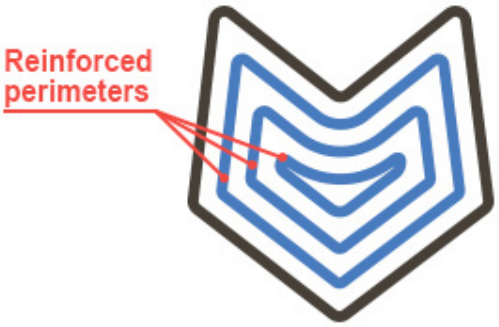
Part

Use fiber

The use fiber flag is the flag which indicates that parts may be reinforced with fiber.

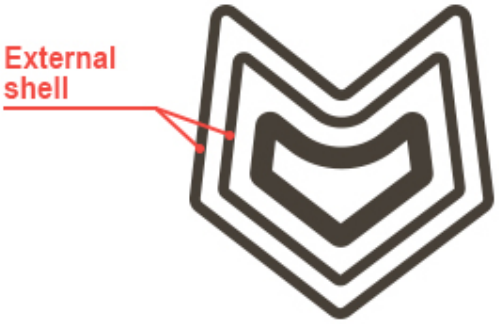
Generate reinforced perimeters

The generate reinforced perimeters flag is the flag which indicates that Aura will generate reinforced perimeters for printed parts.



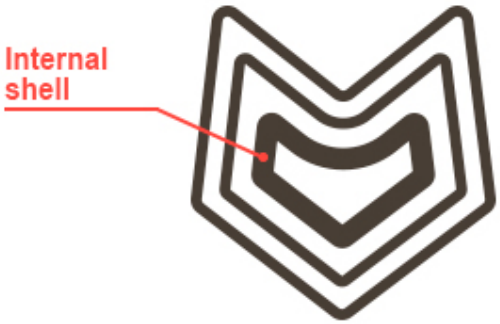
External shell perimeters count

The external shell perimeters is the number of perimeters which build the plastic external shell - the most precise one.



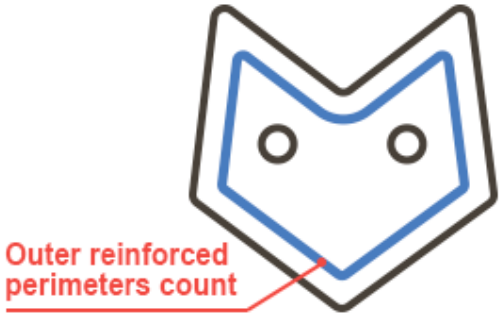
Plastic perimeters count

The plastic perimeters count is the number of perimeters which build the plastic internal shell. It provides solidity of the part surface.



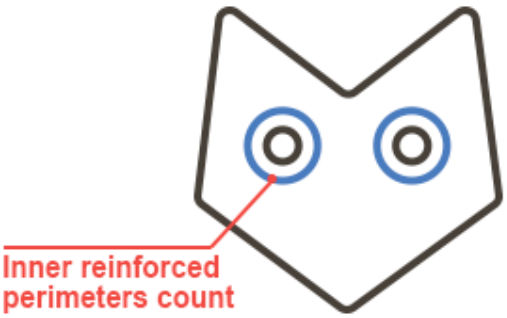
Outer reinforced perimeters count

The outer reinforced perimeters count is the number of perimeters which will reinforce outer surface (not holes) of the part.



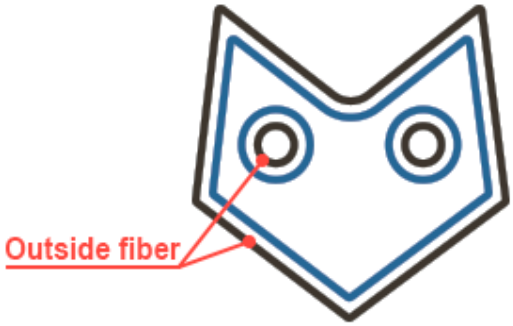
Inner reinforced perimeters count

The inner reinforced perimeters count is the number of perimeters which will reinforce inner holes (not surface) of the part.



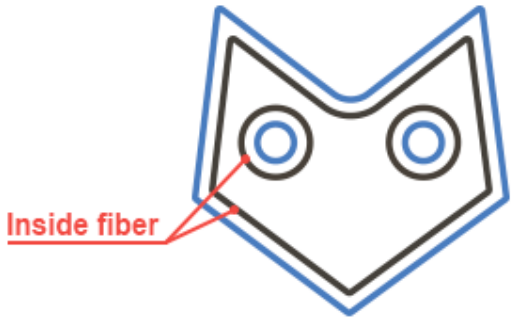
Plastic perimeters count outside fiber

The plastic perimeters count outside fiber is the number of perimeters between an external shell and fiber perimeters.



Plastic perimeters count inside fiber

The plastic perimeters count inside fiber is the number of perimeters between fiber and cellular infill.

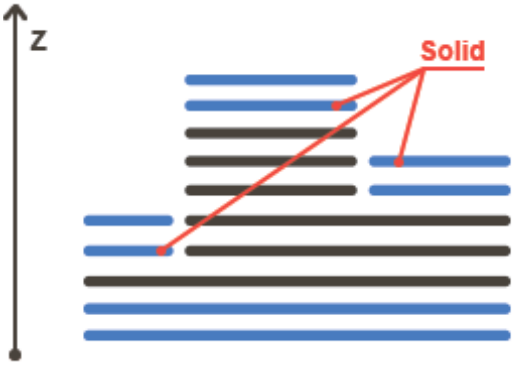


Fill density (%)

The fill density percent is the percent of filled space inside the shell.

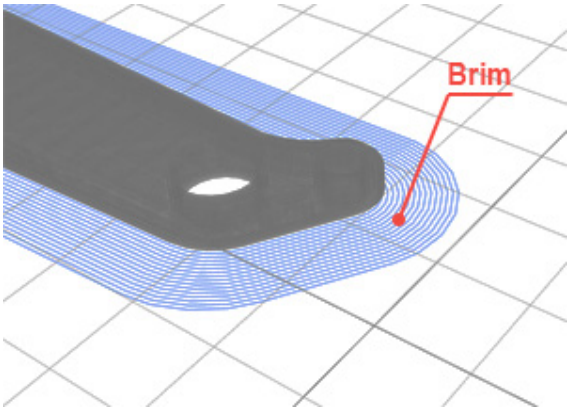
Top/bottom solid layers count

The top/bottom solid layers count is the number of layers in which solid infill will be generated. The solid infill is generated in special spots - where a layer contacts with external environment.



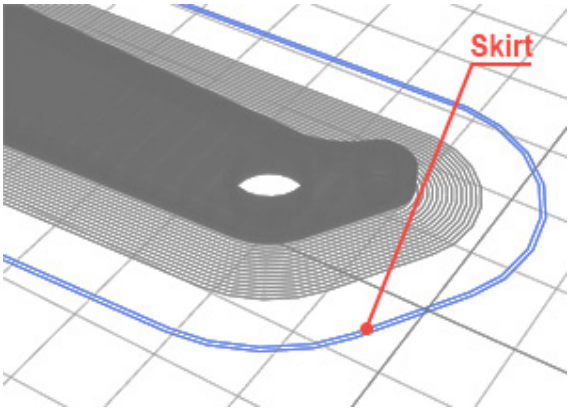
Generate brim

The generate brim flag is the flag which indicates that Aura will generate brims for parts. The brim is the extended contour of a part on the first layer. Use brim to prevent the part from stickung off the table surface during printing process.



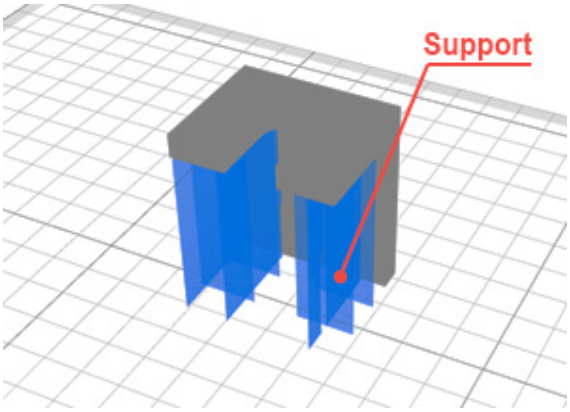
Generate skirt

The generate skirt is the flag which indicates that Aura will generate skirts for parts. The skirt is the extended contour of a part on the first layer. Use skirt to stabilize plastic flow in extruder before main printing.



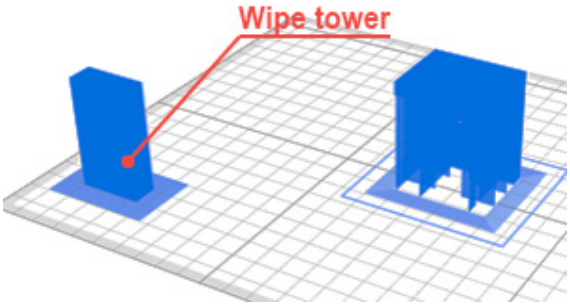
Generate support

The generate support flag is the flag which indicates that Aura will generate supports for the parts. The support is used to prevent the fall of overhanging parts.



Generate wipe tower (BETA)

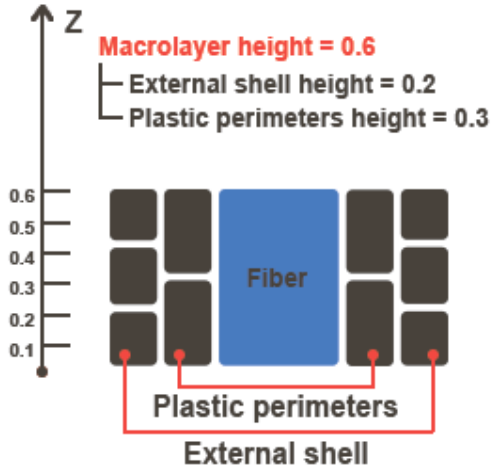
The generate wipe tower flag is the flag which indicates that Aura will generate wipe tower for parts. The wipe tower is a service part that is being printed at the same time as main parts. The wipe tower is used to clean nozzles on tool change.



General

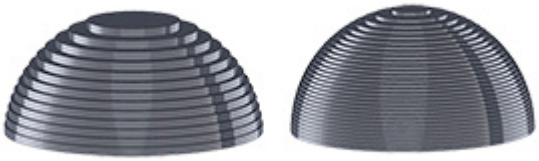
Macrolayer height (mm)

The macrolayer height is the height of a layer package which contains microlayers. The microlayer is the layer which contains one or more print entities with the same height. The macrolayer contains microlayers and its structure is defined by microlayer heights. For example, macrolayer height = 0.6mm, external shell height = 0.2mm, plastic perimeters height = 0.3mm, fiber layer height is always equal to macrolayer height and now it is 0.6mm. Accordinally, in this macrolayer we have 4 microlayers. First on 0.2mm and it has only external shell for printing. Second on 0.3mm and it has only plastic perimeters. Third on 0.4mm and it has only external shell. And the last one on 0.6mm and it has external shell, plastic perimeters and fiber perimeters.



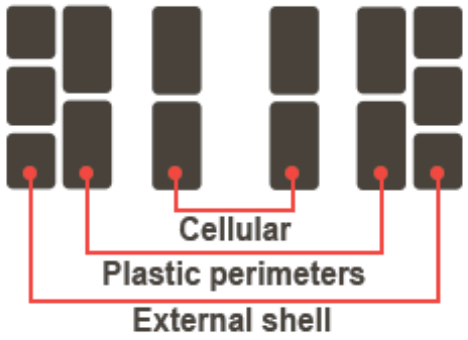
External shell layer height (mm)

The external shell height is the height of thin plastic layers which build up an external shell of a part. The thinner external shell, the more precise and accurate the part looks.



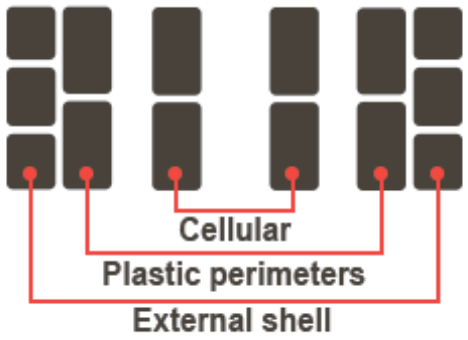
Plastic perimeters layer height (mm)

The plastic perimeters layer height is the height of thick plastic shell. Set this value as high as possible to speed up the print process. Because plastic perimeters are not visible they can be printed thicker than an external shell to speed up the printing process.



Infill layer height (mm)

The infill layer height is the height of layers which contain cellular infill. The cellular infill lies inside plastic perimeters and leaves an empty space in a part.

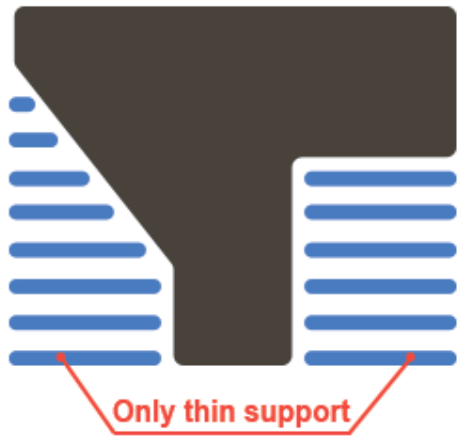


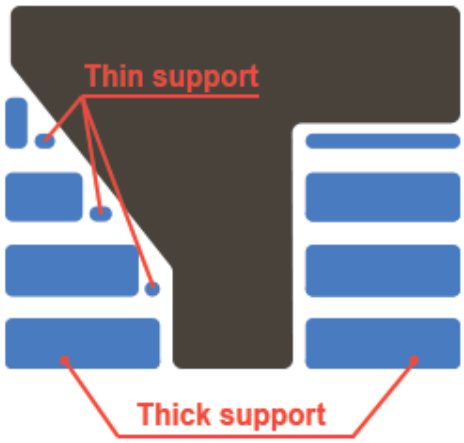
Thick support layer height (mm)

The thick support layer height is the height of layers which contain thick supports. The thick support is calcalated by intersection of thin support layers. Set thick support layer height as large as possible to speed up the print process.

Thin support layer height (mm)

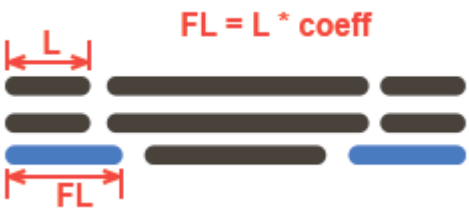
The thin support layer height is the height of layers which contain thin supports. Thin supports are generated in the areas which can't be converted into thick support.





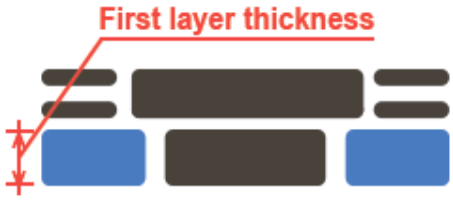
First layer extrusion width coefficient

The first layer extrusion width coefficient is the extrusion width multiplication factor. It has effect on extrusion width for all entities on first layer and consequently on the amount of extruded plastic.



First layer height (mm)

The first layer height is the height of the first layer. Set this parameter a little bit larger that in other microlayers to compensate table relief. The first layer has no microlayers structure as macrolayer and is not a part of a macrolayer.



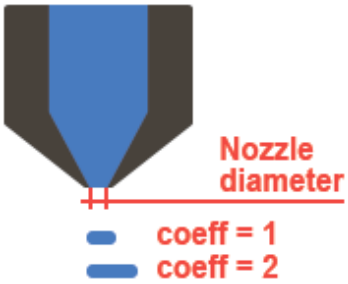
First layer printing speed (mm/sec)

The first layer printing speed is the speed of printing all entities at the first layer. You may want to set this value a little slower than average speed on other layers to improve adhesion between parts and a table.

External shell

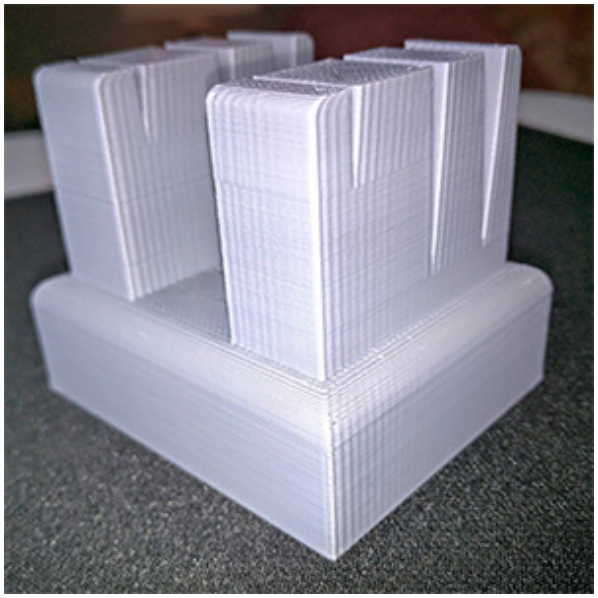
Extrusion width coefficient

The extrusion width coefficient is the nozzle diameter multiplication factor. That is extrusion width = nozzle diameter * coefficient.



Printing speed (mm/sec)

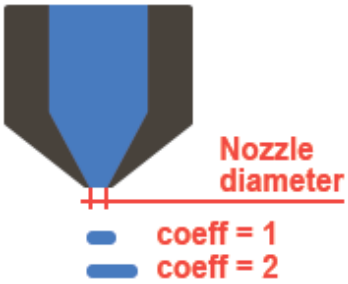
The printing speed is the speed which is applied to external shell printing. Set this parameter to the sufficient minimum to improve surface quality of parts. The slower is the printing speed, the lower is the chance to obtain vertical ringing on parts.



Plastic perimeters

Extrusion width coefficient

The extrusion width coefficient is the nozzle diameter multiplication factor. That is extrusion width = nozzle diameter * coefficient.



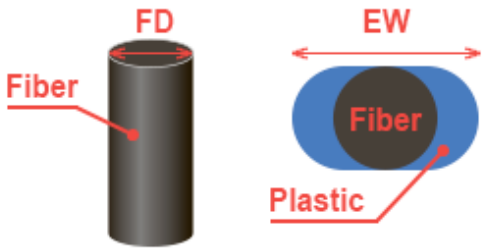
Printing speed (mm/sec)

The printing speed is the speed which is applied to plastic perimeters printing. Set this parameter as high as possible to speed up printing process.

Reinforced perimeters

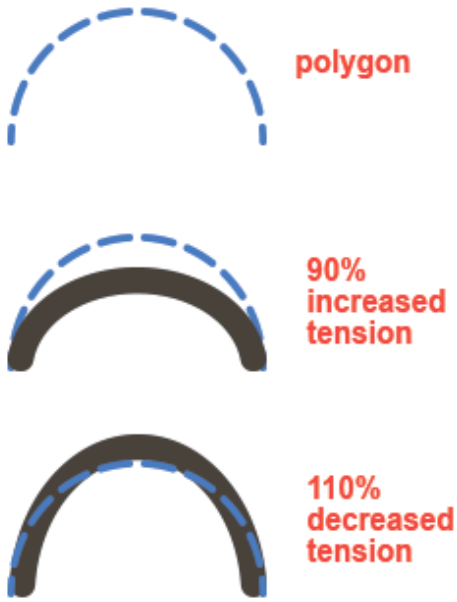
Extrusion width (mm)

The extrusion width is the width of composite material (plastic + fiber).



Fiber feedrate (%)

The fiber feedrate is the percentage of full presumed path length. Decrease this parameter to raise fiber tension and increase this parameter to decrease fiber tension.



Printing speed coefficient

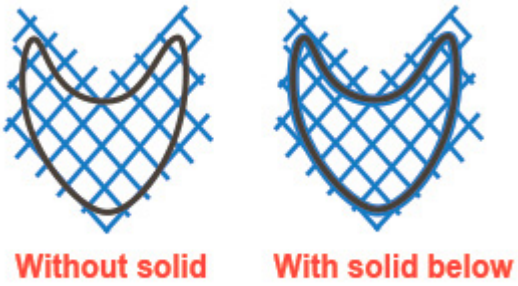
The printing speed coefficient is the multiplication factor which will be applied to fiber print speeds.

Solid plastic layers above fiber count

The solid plastic layers above fiber count is the number of macrolayers in which solid infill will be generated. This parameter is not similar to solid plastic layers below fiber parameter, so you may use it only to cover up fiber polygons from above.

Solid plastic layers below fiber count

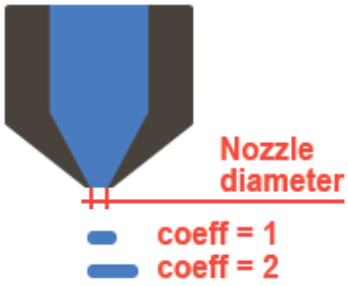
The solid plastic layers below fiber count is the number of macrolayers in which solid infill will be generated. It is necessary to prevent fiber fall through cellular infill.



Solid infill

Extrusion width coefficient

The extrusion width coefficient is the nozzle diameter multiplication factor. That is extrusion width = nozzle diameter * coefficient.

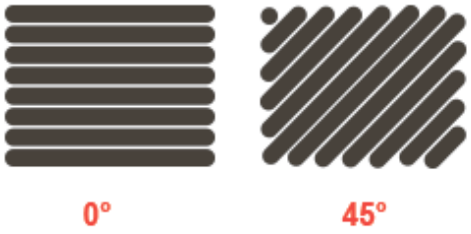


Printing speed (mm/sec)

The printing speed is the speed which will be applied to a plastic infill printing. Set this parameter as high as possible to speed up printing process.

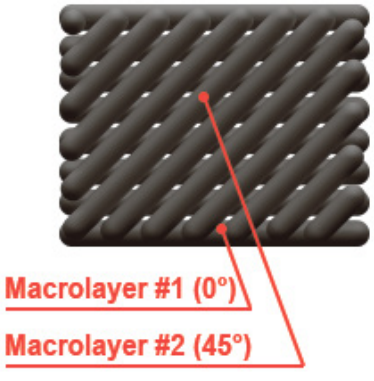
Fill starting angle (°)

The fill starting angle is the starting angle of the infill rotation.



Fill angle offset (°)

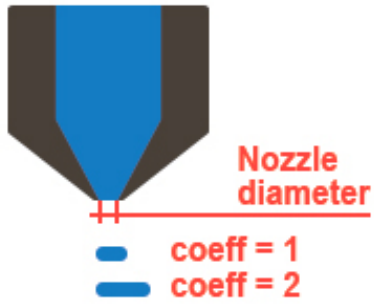
The fill angle offset is the offset angle which is added to current infill rotation angle on each layer. For example, if fill angle offset is 10°, on the first macro layer the angle is 0°, 10° - on the second one, 20° - on the third etc.



Cellular infill

Extrusion width coefficient

The extrusion width coefficient is the nozzle diameter multiplication factor. That is extrusion width = nozzle diameter * coefficient.

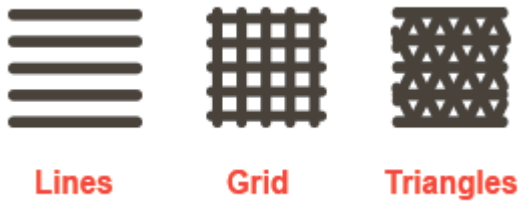


Printing speed (mm/sec)

The printing speed is the speed which will be applied to plastic infill printing. Set this parameter as high as possible to speed up printing process.

Fill pattern

Fill pattern. You could choose one of three patterns: lines, grid or triangles, depending on your preferences.



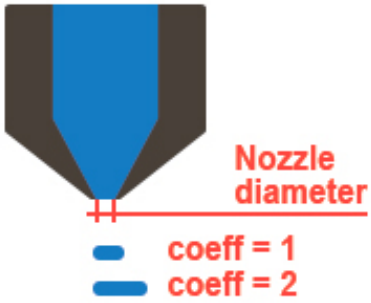
Fill angle (°)

The fill starting angle is the angle of infill rotation.

Brim

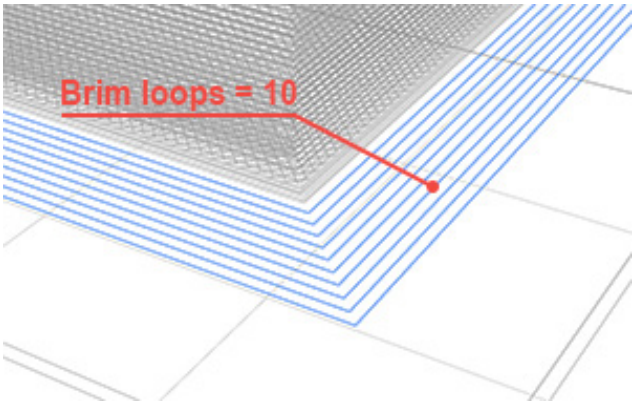
Extrusion width coefficient

The extrusion width coefficient is the nozzle diameter multiplication factor. That is extrusion width = nozzle diameter * coefficient.



Loops count

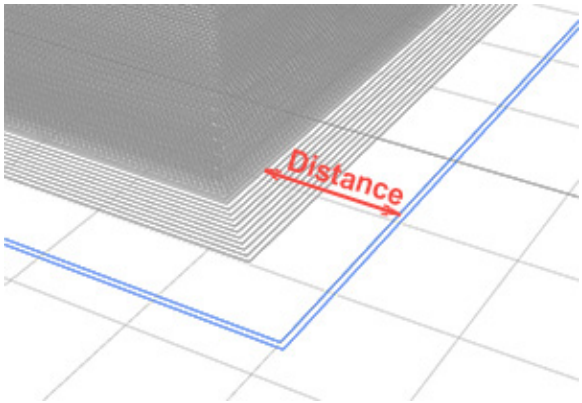
The brim loops count is the number of brim loops around a part. The more loops, the stronger is the adhesion between parts and a table.



Skirt

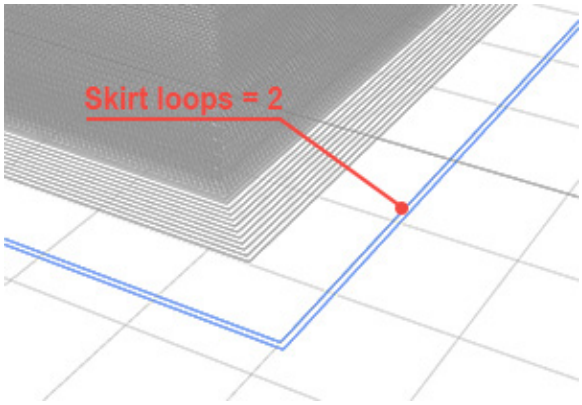
Distance from part (mm)

The distance from part is the distance between a skirt and a part.



Loops count

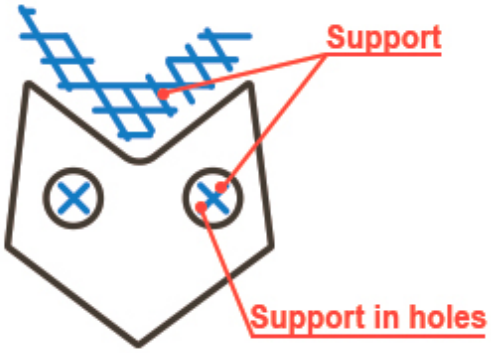
The skirt loops count is the number of skirt loops around a part. Choose the sufficient number to stabilize plastic flow by the end of printing the skirt.



Support

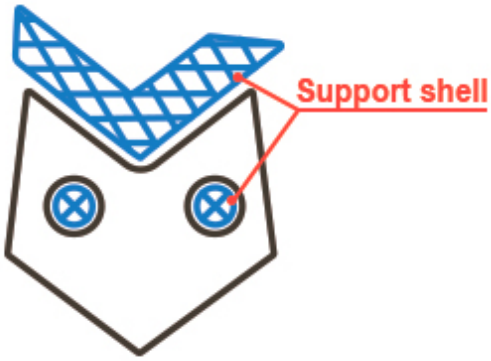
Generate support in holes

The generate support in holes is the flag which indicates that support will be generated not only outside the part, but also inside, in holes.



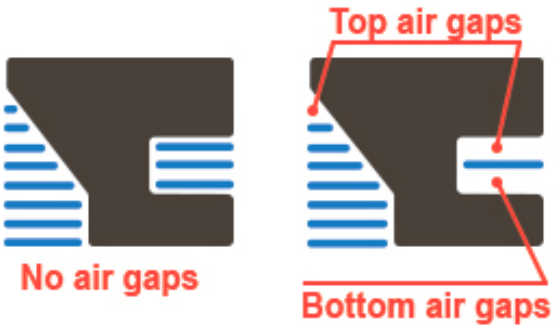
Generate shell for support

The generate shell for support is the flag which indicates that external shell will be generated for supports.



Make airgaps

The make airgaps flag is the flag which indicates that air gaps will be generated onto and/or under supports.



Top airgap thin layers count

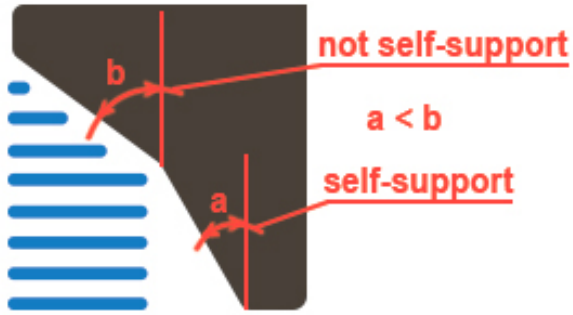
The top airgap thin layers count is the number of layers between the top of a support and the bottom of a part. Choose a value with which a can be easily torn from a part and supported areas will not fall down because of large distance to a support.

Bottom airgap thin layers count

The bottom airgap thin layers count is the number of layers between the bottom of a support and the top of a part. Choose a value so that a support can be easily removed from a part and support will not fall down because of large distance to a part.

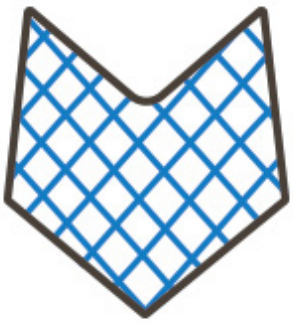
Max unsupported overhang angle (°)

The max unsopported overhang angle up to which part overhang has no need to be supported. If the part's angle exceeds the unsupported angle then the supports will be generated automatically. The angle is calculated from the vertical axe.



Fill density (%)

The fill density is the percentage of filled space inside the support.



Fill pattern

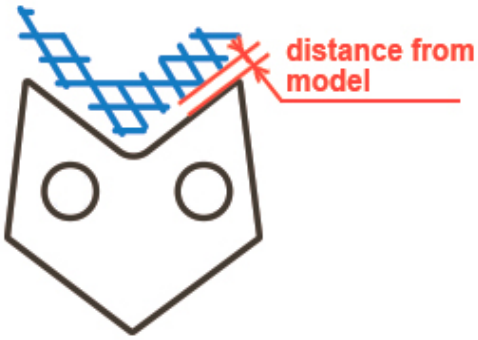
Fill pattern. You could choose from three patterns: lines, grid or triangles, depending on your preferences.

Fill angle (°)

The fill angle is the angle of an infill rotation.

Horizontal X/Y distance from model (mm)

The horizontal X/Y distance from model is the offset from a part to supports. Set this parameter large enough so that a part does not fuse, merge with supports.



Thin support printing speed (mm/sec)

The thin support printing speed is the speed which will be applied to thin support printing. Set this parameter as high as possible to speed up printing process.

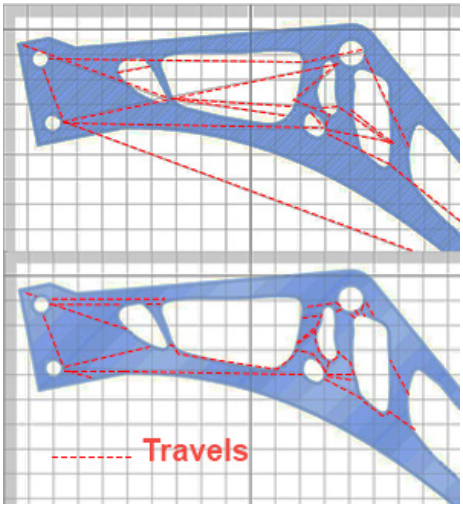
Thick support printing speed (mm/sec)

The thick support printing speed is the speed which will be applied to thick support printing. Set this parameter as high as possible to speed up printing process.

Ooze prevention

Avoid crossing borders

The avoid crossing borders flag is the flag which indicates that border crossing minimization mode in on. It means that if it is possible to travel from one print area to another without crossing borders, that path will be chosen instead of direct path with borders crossing . This mode increases print time, but improves external surface quality.



Do retract only when crossing borders

The do retract only when crossing borders is the flag which indicates that only crossing perimeter travels will have retracts before them. Choose this mode to speed up the print process. Plastic running from nozzle on travels will be left inside a part, so it will not be visible. If you want to do retracts before all travels you should uncheck this flag.

Do retract on changing layers

The do retract on changing layers is the flag which indicates that plastic will be retracted on layer change even if there is no XY movements during layer change.

Do Z-hop when retracted

The do z-hop when retracted flag is the flag which switches on/off a nozzle z-hop mode when travel has retract. Choose this mode to improve external shell surface quality, but keep in mind that it sligtly slow down print process. Besides, it may produce strings between print areas, but usually it is easier to remove them than plastic drops.

Do coast before retract

The do coast before retract is the flag which indicates that plastic polygon will have dry (without plastic feeding) segments in the end.

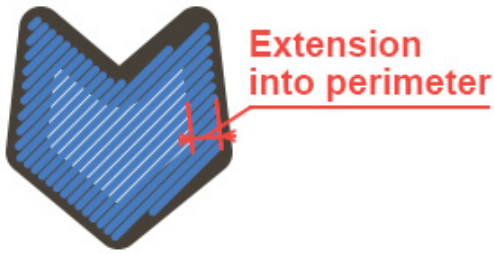
Do wipe nozzle

The do wipe nozzle flag is the flag which indicates that plastic polygon will have reverse movement after full polygon print has ended.

Other

Infill extension into perimeters

The infill extension into perimeters is the distance by which infill penetrates perimeters. Choose the length that provides infills fusion together with perimeters.



Perimeter trim ratio

When this parameter is 0 plastic polygon is fully closed. When this parameter is 1, plastic polygon has empty gap that is equal to extrusion width. Choose the value that is different from 0, if you've noticed the plastic overextrusions on polygon seam.

Minimum layer print time (sec)

When this parameter is 0 plastic polygon is fully closed. When this parameter is 1, plastic polygon has empty gap that is equal to extrusion width. Choose the value that is differ from 0, if you've seen the plastic overextrusions on polygon seam.