Spot and scar solution for 3D printouts

Note: This setting can only be set in simplified3d, so the following explanation is S3D content. If you are interested, you can contact us to give you software and set parameters for you.



During 3D printing, when the extruder moves to a different location, it must continue to stop or start extruding. Most extruders produce a consistent extrusion line during operation. Then each time the extruder is turned on again, it will be a noticeable change. For example, if you look at your 3D print case, you will notice some minor signs of performance that appear in the area where the extrusion started. The extruder must start printing from a certain position in the shell of your 3D model. When the entire shell is printed, the nozzle will return to that position. This is often referred as a spot or scar. As you can imagine, it is difficult to connect two pieces of plastic together without leaving any marks. However, Simplify3D provides several jobs that can minimize this surface break. (Note: This can only be set by S3D.)

Retraction and taxiing

If you notice the small flaws in the printouts, the best way to find out the reason is to carefully observe each edge on the printout. Are these defects present when the extruder starts printing edges? Or does it only appear after the edge has been completed and the extruder stops? If you have just appeared in the beginning, it is likely that your withdrawal settings need to be adjusted slightly. Click on "Edit Process Settings" to open the "Extruders" tab. Below the withdrawal distance setting there will be a name "Extra Restart Distance". This option determines the difference between the retraction distance when the extruder is stopped and the distance when extrusion reloads. If it is found that the surface defect is exactly at the beginning of the edge print, the extruder may have extruded too much consumables. You can reduce the loading distance by entering a negative value in the restart filling setup. For example, if the retraction distance is 1.0mm and then the refill distance is -0.2mm (note the minus sign), then each time the extruder stops, it will retract 0.1mm of consumables. However, each time the extruder restarts, it will only need to push the 0.8mm consumables back to nozzle. Adjust this setting until the extrusion does not appear when you begin to print edges.

If this defect only ends up along the edge and the extruder has to stop, then there is another different setting. This setting is called coasting. You can find it on the Extruder tab page, under Back to settings. When the edge is about to end, the gliding will close the extruder a short distance to eliminate nozzle down pressure. Turn on this setting, and increase the value until you no longer see flaw appear near the edge and the extruder will stop in the future. In general, the sliding distance is set between 0.2 and 0.5 mm, and a very significant effect can be obtained.

Avoid unnecessary withdrawal

The retraction and gliding setups mentioned above can help avoid the flaw

produced by each nozzle retraction, and in some cases, it is better to avoid retraction

completely. In this way, the extruder does not have to reverse the direction, but can perform a pretty uniform extrusion. This is even more important for machines using the Bowden extruder because the large distance between the extruder and the nozzle makes retrieving more troublesome. To adjust the settings for this control fallback, open the Advanced tab and look for the Ooze Control Behavior section. This paragraph contains many useful settings that can modify the behavior of your printer. As we mentioned in the "drawing" section, withdrawal is mainly used to prevent the nozzles from slopping when the nozzle moves between different print portions of the printouts. However, if the nozzle does not move to an open area, the sag will occur inside the model and cannot be seen from the outside. For this reason, many printers need to turn on the setting of “only draw back when you move to open space” to avoid unnecessary withdrawal.

Another related setting can be found in the "Mobile behavior" section. If your printer only withdrawal when it is moved to the open space, then it is better to avoid such open space. Simplify3D has a very useful setting that allows the extruder's path of travel to be steered so that it does not intersect the outer edge of the contour. If you avoid the intersection with the contour by modifying the extruder movement path, then withdrawal will not be needed. To use this feature, simply turn on the "Avoid crossing paths with outer contour" option.

Non-fixed withdrawal

Another very useful feature of Simplify3D is the ability to implement non-fixed withdrawal. This is still useful for the Burton extruder. When printing, there is a lot of pressure in the nozzle. Usually after this type of machine stops extruding, the extruder stands still and it still squeezes a small amount of pressure due to internal pressure. So, Simplify3D adds a unique option that allows you to keep the nozzle moving all the time while performing a withdrawal action. This means that you are less likely to see stationary moustaches because the extruder has been moving during this process. To tear this option, we need to modify some settings. First, click on "Edit Process Settings" to open the "Extruders" tab. Make sure the "Wipe Nozzle" option is on. This will tell the printer to wipe the nozzles before the end of the segment printout. The "Wipe Distance" option, set to 5mm, is a good starting. Then open the Advanced tab page and turn on the option "Extract after wiping". This will avoid a static withdrawal because the printer has been commanded to wipe the nozzle when backing out. This is a very powerful and useful feature. If you are still facing problems with printed surfaces, you can give it a try.

Select starting position

If you still see defects on the print surface, Simplify3d also provides another option to control where these points appear. Click "Edit Process Settings" to open the "Layer" tab. In most cases, the selection of these locations is to optimize the printing speed. Then, you can also randomize these locations or set them to a specific location. For example, if you are printing a statue, you need to set all the starting points and start on the back of the model so that they cannot be seen from the front. turn on the option to “Select the starting point closest to a location,” and enter the XY coordinates you want to use as the starting point if it needs to do.