I'm not a robot

Stringing is one of biggest frustrations in 3D printing. In these notes, I'll summarize simple steps to reduce stringing by calibrating slicer retraction settings. Fighting fine stringing is one of the most frustrating problems to pin down, and our slicers provide numerous work-arounds to help us deal with this problem: retraction and de-retraction length retract/de-retract filament speeds wiping z-lift or z-hop coasting (Cura) Unfortunately, the testing process can be a long and tedious process. It is particularly frustrating to test individual small prints with different printer settings such as retraction. Important Be sure you've calibrated your slicer filament settings before trying to find the perfect retraction settings. Retraction and related settings are work-arounds for imperfect extrusion results. Calibrate your filament settings first to match your slicer extrusion settings to your real-world printer and filament settings first to reduce the number of work-arounds required to get good results. One trick is to use a "virtual multi-extruder" printer profile to create a multi-part single print with unique hardware settings for each part. Note These notes are based on my experiences with the Prusa i3 Mk3 and Artillery/Evnovo Sidewinder X1 printers. If you are using a different printer, please verify the hardware details are similar. The idea is to create a printer profile with multiple extruders and enter different settings for each extruder. Assign each part to a different extruder and you can (with a bit of patience) generate several tests with a single print as shown below. Fig. 6.48 Retraction calibration print It's a little cumbersome, but it beats trying to track multiple tiny prints. Here's an example showing several retraction setting modifications: Retraction length is set to 0.2mm Lift Z is set to 0mm Retraction speed is set to 50mm/s Deretraction speed is set to 25mm/s Fig. 6.49 Extruder settings Provided each part is small enough, you can produce a single print that compares multiple settings. Fig. 6.50 Retraction calibration print results Note Thanks again @Ringarn67 for catching the filament override error in the sample file! I've removed the example settings for now to avoid confusion. I use the following retraction settings as a starting point: 0.2-0.4mm retraction 0-0.2mm z-lift 50mm/s retract speed 25mm/s de-retract speeds Contact and feedback You can find me on the Prusa support forums or Reddit where I lurk in many of the 3D printing-related subreddits. I occasionally drop into the Official Prusa 3D discord server where I can be reached as bobstro (bobstro#9830). You can email me directly at projects@ttlexceeded.com. Last edited on Aug 13, 2021. Last build on Aug 30, 2021. Reddit and its partners use cookies and similar technologies to provide you with a better experience. By accepting all cookies, you agree to our use of cookies to deliver and maintain our services and site, improve the quality of Reddit, personalize Reddit content and advertising, and measure the effectiveness of advertising. By rejecting non-essential cookies, Reddit may still use certain cookies to ensure the proper functionality of our platform. For more information, please see our Cookie Notice and our Privacy Policy. Stringing or oozing, also known as "hairy prints", is the name given for when small strings of filament are left on a printed model. This usually happens when the filament keeps flowing from the nozzle while the extruder is moving to another object. You can see this as a marginal line of filament left between the objects. This issue is caused by very high printing temperatures and/or using incorrect retraction settings. This can be solved by changing a couple of settings in PrusaSlicer and checking your hardware. Stringing from material left on the nozzle If you print for a long time from a single type of filament, such as PET-G, the filament can create a thin layer in the nozzle. This can cause stringing as the strands of the filament stick to the surface of the print. Therefore, thoroughly clean the nozzle before printing and make sure that any dirt or remnants of previous filaments are removed from the nozzle. We highly recommend using our official (factory default) preset settings in PrusaSlicer. However, if you are printing with your own settings, make sure that you have the retraction settings configured correctly. PrusaSlicer settings You should start by checking a parameter known as Retraction. What does retraction do? When the extruder has finished printing one section/object of your g-code, the filament is pulled back into the nozzle. Once the extruder moves to the next location the printing process continues – the filament is pushed back out and it starts extruding from the nozzle again. Retraction settings can be found in PrusaSlicer in Printer Settings -> Extruder 1. Flexible filaments usually need longer retractions, because the material stretches while being pulled back to the nozzle. Flexible materials are a special case and can need a lot of tweaking and tuning. The retraction settings are: Retraction length: Amount that the filament is pulled back when a retraction is triggered. On the MK2.5/S and MK3/S/+, the retraction length should be a maximum of 2 mm. Due to the Bowden extruder, the retraction length of the Original Prusa MINI/MINI+ presets are much longer (default 3.2 mm). Lift z: lifts the extruder during movement. Having this setting lower will improve stringing. Note that disabling this feature may cause the nozzle to hit the printed part. Retraction Speed: Extruder motor speed on retraction. A higher value improves stringing, but if it is too high it may skip steps in the motor. Minimum travel after retraction: Amount of move that will trigger a retraction (mm). The preset number in PrusaSlicer is 1 mm, which is a low amount. Having this number higher will get lower printing times, but increase oozing and stringing. Retract on layer change: Activates retraction when the layer changes to the next one. It is recommended to leave this option on. Wipe while retracting: Moves the nozzle (wipe) while the retraction is happening. It is recommended to leave this option on. Retract amount before wiping: This option does a quick retract before doing the wiping movement. More suitable for the Original Prusa MINI/MINI+. Other settings that affect retraction: Only retract when crossing perimeters: (Print settings -> Infill -> Advanced): Disables retraction when the travel path does not exceed the upper layer's perimeter. Any oozing that happens will be within the walls and should be invisible. Avoid crossing perimeters: (Print settings -> Layers and perimeters -> Quality): Optimize travel moves in order to minimize the crossing of perimeters. This will lower the stringing amount, especially in the MINI/MINI+. Sequential printing: (Print settings -> Output options -> Sequential printing): Printing each object individually has a smaller chance of causing stringing between the parts. When using this feature, PrusaSlicer will warn you of any possibility of the extruder colliding with an already printed part, but follow the print closely. Nozzle temperature: (Filament settings -> Filament -> Nozzle): Lowering the temperature decreases the occurrence of strings. Try decreasing the nozzle temperature by 5 – 10°C and check whether there's less stringing. Filament Try using a different spool than the one that is causing the stringing. The filament might have gathered moisture, which will cause a lot of stringing. Hotend Stringing in the nozzle might be caused by a heat dissipation issue in the nozzle. Try re-applying thermal paste on the threads between the heatbreak and the heatsink. If you have recently changed any component in the hotend, it is also possible that some individual parts are not in place. Go over the assembly of the hotend and check for any parts that may be different from the instructions. Enough settings, pass me my heat-gun! If you don't feel like tweaking any of the settings, well, then there is an alternative. You can get rid of the strings with a heat gun (or often with a lighter – but be very careful). Set your heat-gun to around 200 °C and aim at the strings for one or two seconds. This will melt the strings, and the printed object should remain undamaged. Do not leave the heat source on the printed model for longer than one or two seconds, as this may deform the part. I am currently using Length 3.2mm and Retraction Speed 30 mm/s and results are good! I am looking for something a little better than that. I have tweaked and tweaked (maybe I should twerk lol) and end up coming back to the 3.2mm / 30 mm/s because all the tweaks had detrimental effects. I have not seen any info or advice on the other retraction settings as shown highlighted below. Does anyone have any experience of changing these settings? Would very much appreciate your help! Posted : 04/12/2021 8:58 am @briancadtec3d-co-uk welcome to the forum. well, for starters, you probably haven't seen advice on the other retraction settings as shown highlighted, because you posted in the "Awesome prints (Hall of Fame)" section of the forums. beyond that, you made no mention what so ever of what kind of printer your using, what kind or brand of filament your using, and apparently no one was kind enough to tell you this. So please let this be your official notification. normally we would see these posted in the "how do I print this" section, "Hardware Firmware & Software help" section or even the "PrusaSlicer" section of the forum. This information should be accompanied by Printer type, filament type, brand and potentially a project file, as all settings are specific to the model being printed. I can tell you right now if your printing on a Prusa direct drive printer such as an MK3 model your 3.2mm of retraction is way to much. Please follow up in one of the aforementioned sections for further help. Good Luck Swiss_Cheese Posted : 05/12/2021 12:29 am Hi Thank you for your response and advice. I am actually using a PRUSA Mini + and the filament is 1.75mm PLA from 3DFilaPrint. I will check out the link you sent me. Thank you Posted : 05/12/2021 3:37 pm I'm still looking for the perfect retraction settings. You'll be looking for a long time... "Perfection" is elusive. Different filament types or even brands may require different settings. I am currently using Length 3.2mm and Retraction Speed 30 mm/s and results are good! I am looking for something a little better than that. Better than good? How good is "good"? Remember perfection is the enemy of good. Little angle hairs (in particular with PETG)? You may spend the rest of your life toiling with improvements while a blast with the heat gun can take care of it in five seconds. Absolutely nothing wrong with tweaking parameters, in particular if you're having fun tinkering! I'm just offering a pragmatic alternative—let good enough alone—, as chasing down perfection can quickly take you down a rabbit hole Formerly known on this forum as @fuchsr -- Posted : 06/12/2021 12:45 pm BogdanH, languer and liked It sounds like maybe you're already at the optimal setting if you keep coming back to that one and any changes are worse. Mini+ (kit) - Revo Micro | Antler Cooling | WiFi (4.4.0RC1) Posted : 07/12/2021 3:19 pm It sounds like maybe you're already at the optimal setting if you keep coming back to that one and any changes are worse. Mini+ (kit) - Revo Micro | Antler Cooling | WiFi (4.4.0RC1) Posted : 07/12/2021 3:20 pm Can you post a picture of your print results? What problems do you have? So we can help you to get a better results, it's hard for us to tell you the better settings if we didn't know what the problems are. Posted : 08/12/2021 6:05 pm Better than good? How good is "good"? Remember perfection is the enemy of good. Little angle hairs (in particular with PETG)? You may spend the rest of your life toiling with improvements while a blast with the heat gun can take care of it in five seconds. Absolutely nothing wrong with tweaking parameters, in particular if you're having fun tinkering! I'm just offering a pragmatic alternative—let good enough alone—, as chasing down perfection can quickly take you down a rabbit hole This is easily one of the best advices I've seen. And it is so true, and hits so close to home. Thanks for the reminder. This post was modified 3 years ago by languer Posted : 10/12/2021 3:26 am Stringing or oozing, also known as "hairy prints", is the name given for when small strings of filament are left on a printed model. This usually happens when the filament keeps flowing from the nozzle while the extruder is moving to another object. You can see this as a marginal line of filament left between the objects. This issue is caused by very high printing temperatures and/or using incorrect retraction settings. This can be solved by changing a couple of settings in PrusaSlicer and checking your hardware. Stringing from material left on the nozzle If you print for a long time from a single type of filament, such as PET-G, the filament can create a thin layer in the nozzle. This can cause stringing as the strands of the filament stick to the surface of the print. Therefore, thoroughly clean the nozzle before printing and make sure that any dirt or remnants of previous filaments are removed from the nozzle. We highly recommend using our official (factory default) preset settings in PrusaSlicer. However, if you are printing with your own settings, make sure that you have the retraction settings configured correctly. PrusaSlicer settings You should start by checking a parameter known as Retraction. What does retraction do? When the extruder has finished printing one section/object of your g-code, the filament is pulled back into the nozzle. Once the extruder moves to the next location the printing process continues – the filament is pushed back out and it starts extruding from the nozzle again. Retraction settings can be found in PrusaSlicer in Printer Settings -> Extruder 1. Flexible filaments usually need longer retractions, because the material stretches while being pulled back to the nozzle. Flexible materials are a special case and can need a lot of tweaking and tuning. The retraction settings are: Retraction length: Amount that the filament is pulled back when a retraction is triggered. On the MK2.5/S and MK3/S/+, the retraction length should be a maximum of 2 mm. Due to the Bowden extruder, the retraction length of the Original Prusa MINI/MINI+ presets are much longer (default 3.2 mm). Lift z: lifts the extruder during movement. Having this setting lower will improve stringing. Note that disabling this feature may cause the nozzle to hit the printed part. Minimum travel after retraction: Amount of move that will trigger a retraction (mm). The preset number in PrusaSlicer is 1 mm, which is a low amount. Having this number higher will get lower printing times, but increase oozing and stringing. Retract on layer change: Activates retraction when the layer changes to the next one. It is recommended to leave this option on. Wipe while retracting: Moves the nozzle (wipe) while the retraction is happening. It is recommended to leave this option on. Retract amount before wiping: This option does a quick retract before doing the wiping movement. More suitable for the Original Prusa MINI/MINI+. Other settings that affect retraction: Only retract when crossing perimeters: (Print settings -> Infill -> Advanced): Disables retraction when the travel path does not exceed the upper layer's perimeter. Any oozing that happens will be within the walls and should be invisible. Avoid crossing perimeters: (Print settings -> Layers and perimeters -> Quality): Optimize travel moves in order to minimize the crossing of perimeters. This will lower the stringing amount, especially in the MINI/MINI+. Sequential printing: (Print settings -> Output options -> Sequential printing): Printing each object individually has a smaller chance of causing stringing between the parts. When using this feature, PrusaSlicer will warn you of any possibility of the extruder colliding with an already printed part, but follow the print closely. Nozzle temperature: (Filament settings -> Filament -> Nozzle): Lowering the temperature decreases the occurrence of strings. Try decreasing the nozzle temperature by 5 – 10°C and check whether there's less stringing. Filament Try using a different spool than the one that is causing the stringing. The filament might have gathered moisture, which will cause a lot of stringing. Hotend Stringing in the nozzle might be caused by a heat dissipation issue in the nozzle. Try re-applying thermal paste on the threads between the heatbreak and the heatsink. If you have recently changed any component in the hotend, it is also possible that some individual parts are not in place. Go over the assembly of the hotend and check for any parts that may be different from the instructions. Enough settings, pass me my heat-gun! If you don't feel like tweaking any of the settings, well, then there is an alternative. You can get rid of the strings with a heat gun (or often with a lighter – but be very careful). Set your heat-gun to around 200 °C and aim at the strings for one or two seconds. This will melt the strings, and the printed object should remain undamaged. Do not leave the heat source on the printed model for longer than one or two seconds, as this may deform the part. Stringing or oozing, also known as "hairy prints", is the name given for when small strings of filament are left on a printed model. This usually happens when the filament keeps flowing from the nozzle while the extruder is moving to another object. You can see this as a marginal line of filament left between the objects. This issue is caused by very high printing temperatures and/or using incorrect retraction settings. This can be solved by changing a couple of settings in PrusaSlicer and checking your hardware. Stringing from material left on the nozzle If you print for a long time from a single type of filament, such as PET-G, the filament can create a thin layer in the nozzle. This can cause stringing as the strands of the filament stick to the surface of the print. Therefore, thoroughly clean the nozzle before printing and make sure that any dirt or remnants of previous filaments are removed from the nozzle. We highly recommend using our official (factory default) preset settings in PrusaSlicer. However, if you are printing with your own settings, make sure that you have the retraction settings configured correctly. PrusaSlicer settings You should start by checking a parameter known as Retraction. What does retraction do? When the extruder has finished printing one section/object of your g-code, the filament is pulled back into the nozzle. Once the extruder moves to the next location the printing process continues – the filament is pushed back out and it starts extruding from the nozzle again. Retraction settings can be found in PrusaSlicer in Printer Settings -> Extruder 1. Flexible filaments usually need longer retractions, because the material stretches while being pulled back to the nozzle. Flexible materials are a special case and can need a lot of tweaking and tuning. The retraction settings are: Retraction length: Amount that the filament is pulled back when a retraction is triggered. On the MK2.5/S and MK3/S/+, the retraction length should be a maximum of 2 mm. Due to the Bowden extruder, the retraction length of the Original Prusa MINI/MINI+ presets are much longer (default 3.2 mm). Lift z: lifts the extruder during movement. Having this setting lower will improve stringing. Note that disabling this feature may cause the nozzle to hit the printed part. Retraction Speed: Extruder motor speed on retraction. A higher value improves stringing, but if it is too high it may skip steps in the motor