

3D HELP EN

[slovak version...](#)



We have collected for you a few of the most common, as well as less common cases when 3D printing causes wrinkles on your face and their possible solutions, tips and advice..

We will be happy to add your experience and solutions to any point, if you send them to us.
Happy printing !!! :)

1. Blocked Nozzle

2. Nozzle too close to print bed

3. Print head misses the bed

4. Cracked filament

5. Stripped filament

6. Printing stopped mid-print

7. Print doesn't stick to bed

8. Print stuck to bed

9. Supports fall apart

10. Messy first layer

16. Gaps between infill and outer wall

17. Visible infill from the outside

18. Cracks in high objects

19. Print leans when it shouldn't

20. Messy overhangs

21. Fine details not printing correctly

22. Ripples and "echoes" in print (Ringing)

23. Diagonal scars on print

24. Print looks stringy and droopy (Over-extrusion)

25. Under-extrusion

11. Print bows out at bottom - elephant foot

12. Missing layers

13. Layer misalignment

14. Deformation - Warping

15. Broken / chaotic infill

26. Print looks melted and deformed

27. Pits and hollows in top layer (Pillowing)

28. Stringing

29. The print has offset in some places

30. Bridges are messy.



1. Blocked Nozzle

I. Needle cleaning

With little bit of luck, the unblocking of nozzle could be a quick a easily done. Start by removing of the filament. Then heat up the nozzle throug your control panel (if it has the function) to the melting point of the stuck filament. Or you can connect the printer to compatible control software and heat up the nozzle. For the PLA set up the temperature to 220 C. Once the nozzle heat up at right temperature, use a thin pin or special needle for that use to clear the hole of the nozzle (attention: to not burn your fingers).

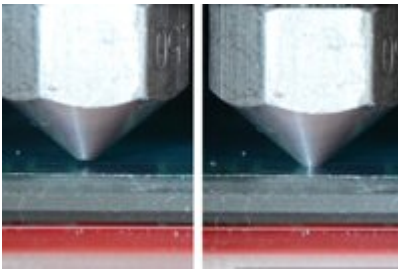
II. Pushing the blocked filament

If the nozzle is always blocked, you can try to push the filament through with another piece of filament. Firstly remove the stucked filament as before by heating up and remove the feeder tube from the print head. Heat up the hotend up to 220 C for PLA or adequate temperature for the stucked material. Then use another piece of filament to push it through from the top, in order to force the stucked filament out of the nozzle. Attention, do not push too much as you risk to bend the horizontal rods of your printer.

III. Disassemble and rebuild your hotend

In hard stucked cases of the nozzle, you will need to do some disassembly of your hotend. To be sure you will reassemble everything correctly, its good point to take notes and photos when you disassamble the hotend. First of all, rempove the stucked filament as previously, then check your printers manual to know exactly how to proceed to dismantle the hotend.

Back to the top.



2. Nozzle too close to print bed

I. Z-axis offset

Even minor raise of the height of nozzle can often help. A lot of printers have in their system settings possibility to set a Z-axis offset. In order to raise the nozzle away from the bed, you have to move the offset settings in positive value. The contrary process – negative offset value will help you when your prints are not well sticking to the print bed.

II. Lower the print bed

Eventually, the printers with function of lowering the bed, can achieve the same effect. However, this is more difficult fix, as it requires the new calibrating and leveling the print bed for your prints.

[Back to the top.](#)



3. Print head misses the bed

I. Checking the slicer software for correct print settings

First of moving forward, check that you have the right printer selected in the slicer software. There is a lot of printer types and even if the bed is the same for two different printers, they have for sure different other settings that may not match.

II. Firmware update

Make sure your firmware is up to date – latest version, especially if you have a new printer. After successful update, run the setup process and check the size of the print area.

III. Check the end stops

Suggested for advanced users. Watch the moves of your printer head. If the head tries to move beyond the furthest point of any of axes, check that your endstops are not disconnected. If not, and everything looks OK, then next step should be replacing the endstops for the new ones.

[Back to the top.](#)



4. Cracked filament

I. Remove the filament

First step to do is removing the filament from the printer as obviously. In some cases of printers as for example Ultimaker, select the maintenance and then change material. The filament is often broken in the tube, so you will have to remove the tube as from hotend also from extruder. Heat up the nozzle and pull out the filament.

II. Test with another filament

If the issues occurs again, try to use another filament to ensure its not because of old filament that should be no more used.

III. Release the tension of filament

Check if the iddler tensioner is not too much tight – release it all the way and than after print starts, tighten it till the filament doesnt slip.

IV. Nozzle check

Make sure that the nozzle is not blocked – eventually good clean up will be not bad idea.

V. Temperature and flow rate control

Check out the correct temperature of the hotend and also if the flow rate of the filament is at 100%, not higher, if the issue still continue.

[Back to the top.](#)



5. Stripped filament

I. Manual forcing of the filament

If the filament starts to slip, then press lightly on the filament to help it pass through the system. This will often help to get the machine to print again without any problems.

II. Adjusting of the idler tension of the filament

Release the feed wheel, insert the filament and tighten it until it stops sliding. The fibers have different diameters, so although the tension wheel absorbs some differences in diameter, some filaments will need some adjustment.

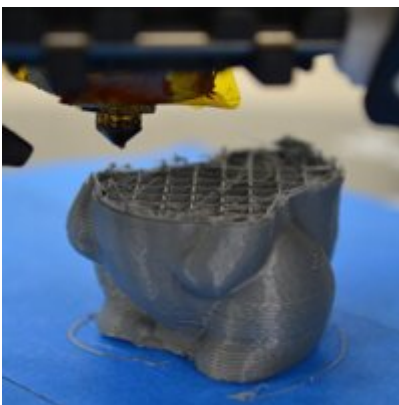
III. Removing the filament

In most cases, you will need to remove and replace the filament and then load it back into the system. After removing the filament, cut it under the area that shows signs of damage, and then return it to the system. If it gets stuck again and you find that the filament appears brittle, discard it and use a different filament.

IV. Hotend temperature check

If you just inserted a new filament when the problem started, check again to make sure you have the correct temperature.

Back to the top.



6. Printing failure mid-print

I. Make sure, you have enough filament

Certainly obvious, but even the best of us have some loss of concentration. Many slicer softwares now provide a material estimation for your projects, and based on the weight of your spool, make sure you have enough filament left for the print.

II. Check for the damaged filament

A damaged fiber can be responsible for a mid-process print failure and can be caused by a lot of problems. Check out point 5 – stripped filament.

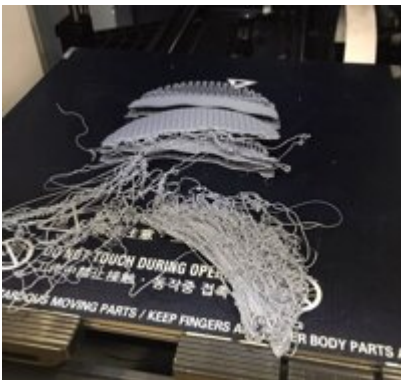
III. Nozzle check

A nozzle blocked with old burnt fiber can cause several different printing problems, one of which is blocking the deposition of the layers. Check out the point 1 – blocked nozzle.

IV. Check for the broken filament

This problem mainly affects bowden printers, as a broken filament can cause disconnection between the extruder and the hotend. It's easy to diagnose and fix, but it can be a sign that your filament is no longer in the best condition. Check out the point 4 – cracked filament.

Back to the top.



7. Print doesn't stick to bed

I. Put some adhesive texture

To ensure that the filament will stick well to your print bed, you will need to add some adhesive texture. The best solution is to put a thin layer of stick glue to the bed that can be easily cleaned up with hot water. For some material as PLA is another alternative to put on print bed decorative tape. For materials which need a heated bed of 40 C and more, there is plenty of various adhesives as for example 3D lac or Dimafix that will help you to stick your print well.

II. famous: LEVEL THE BED !!

Every printer has its own procedure how to level the bed correctly. Some of the printers use reliable auto leveling process, others use their own step-by-step process to align your print bed. Use your printer manual for this procedure.

III. Correct set-up of the nozzle height

If the nozzle is too high from the print bed, the filament will not stick correctly to the bed, in reverse, if it is too low, it can scrap and damage your print bed. Use your Z-axis offset setup to make the right corrections. Adjusting into positive value will raise the nozzle away from the bed, negative value will lower it closer.

IV. Clean print bed

If your print bed is glass and you are using often glue for better sticking, it's not a bad idea to give a platform some cleaning from time to time. Touching your bed with fingers, accumulation of glue textures are often contributing to non-stickness of your platform.

V. Adding a build plate adhesion – brim/raft

Bigger prints are often printing well, with good adhesion to the platform, but smaller models or models with a small footprint on your print bed will often need some type of build plate adhesion as brim or raft, which can be easily added in your slicer software.

Brim adds a single layer of a few number of perimeter lines to improve adhesion contact of the print to the bed. Brim is less wasteful of these two options.

Raft - depending on the parameters you enter, the raft adds a certain shadow of your print to the platform, which is printed in a thicker and better adhesive layer. Then your model is printed on the raft in the usual way. Rafts tend to create a rough and uncomfortable surface where it touches your print, and consume more material than Brim. However, the advantage of Raft is that it can be easily rip off the platform.

VI. Adding supports

If your model has complex overhangs or ends, be sure to add supports to hold the print together during the printing process, in addition to adding an adhesive layer.

[Back to the top.](#)



8. Print stuck to bed

I. Patience

You were waiting for hours for the print, so waiting a little longer for the print to cool completely will not hurt. After some time of cooling may be released on his own will.. When the print cools, it hardens and loses its stickiness.

II. Use a sharp cutter

They come with almost every 3D printer, and you're sure to find them in every hardware store. If you have allowed it to cool for an hour and the print is still stuck firmly, release the bed, and then carefully use the knife/cutter to separate the edge from the bed and peel off the print. Beware of fingers and injuries.

III. Clean print bed

If you use any type of adhesives, keep the print bed clean to avoid tight bonding of the printouts in the future. It will not help for actual issue, but will help you with your future prints.

IV. Do not use cheap and poor quality filament

Back to the top.



9. Supports fall apart

I. Correct type of supports

See the type of model you are going to print. If there are large overhangs that connect parts of the model and these have good contact with the bed, try using lines or zig-zag supports. If the model has less contact with the bed or needs much stronger supports, use grid or triangular supports.

II. Adding a build plate adhesion

Ensure, you added some type of adhesive layer, such as Brim, so that the supports have enough foundation to stick to.

III. Better support density

Increasing the support density will give your model a denser structure to hold on, and will be less affected by any movement of the model, but will be much more difficult to remove.

IV. Vymeňte filament

The filament may become brittle over time when it reaches the end of its life, and this usually affects the quality of the supports. Replace the filament with a new one to see if the problem improves.

V. Overall check of the printer

Various shocks and vibrations with the printer during printing can be a real problem. Check the machine to make sure everything is in place, as tight as it should be, and tighten and recalibrate as needed if necessary.

[Back to the top.](#)



10. Messy first layer

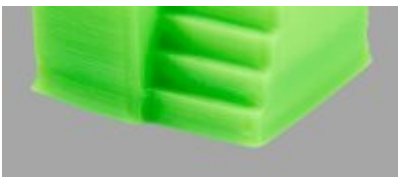
I. LEVEL THE BED !!

Every printer has its own procedure how to level the bed correctly. Some of the printers use reliable auto leveling process, others use their own step-by-step process to align your print bed. Use your printer manual for this procedure.

II. Lower the bed temperaturee

Try lowering the bed temperature by 5 degrees until you reach the correct adhesion point, without losing detail.

[Back to the top.](#)



11. Print bows out at bottom - elephant foot

I. Balance between bed temperature and cooling

To prevent an "elephant's foot" from appearing in your 3D prints, you must cool the base layers of the model enough to support the higher layers. However, if you use too much cooling, you risk that the base will deform. Proper balancing can be difficult, starting by lowering the bed temperature by 5-degree intervals (to +/- 20 degrees from the recommended temperature). If your bottom/top thickness is set to 0.6 mm, start the fan slightly lower.

II. LEVEL THE BED !!

Most printing problems can be traced more often till to the print bed level. Each printer has a slightly different technique for bed leveling. Start the calibration according to the procedure recommended by your printer manufacturer. By printing the calibration cube you can watch, how your printer lays your layers and if it needs calibration or not.

III. Raise the nozzle

Only a slight increase in nozzle height can often help, but be careful too high and it will not stick to the bed.

IV. Chamfer the base

Another option is to chamfer the base of the print. Of course, this is only possible if you designed the model yourself or have access to the original file. Start with a 5 mm and 45 ° bevel, but for a best result, experiment a bit.

Back to the top.



12. Missing layers

I. Mechanical check

Missing layers are the first sign something is going on with mechanical parts of your 3D printer. Try to check all the rods, bearings, gears, belts etc. If you will find something strange (vertical or horizontal clearance of spare parts) please consider to replace it with new spare part.

II. Check the rods

There is dirt cumulation on rods during prints. This can cause resistance at running of the printing head. Result is irregular layering. To fix the problem clean up the rods and oil them smooth.

It might happen you will see the printing head is slowing down at several points (locations) while printing (or it will stop/block). This is caused by rods bending. During the time leading rods can bend. Solution is to move the printing head manually over the rods and check/find the place where is the resistance point.

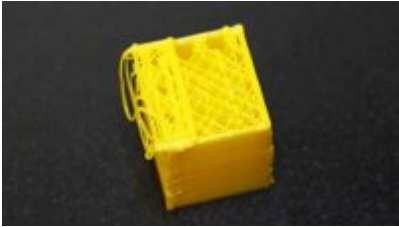
III. Checking drive pulleys, gear wheels and bearings

Drive pulleys and gear wheels are driving the belts and are connected to stepper motors. Try to check if there is all tight like it supposed to be. The same is valid for bearings.

IV. Check lubrication of the moving parts

Each mechanical part which is making some movement must be lubricated. The best way is to apply machine oil. Please note too much oil can harm the machine and too less oil as well. It's strictly recommended to clean up the rods, bearings etc. before putting oil on.

[Back to the top.](#)



13. Layer misalignment

I. Check the belts

Check if the belts are tight, (ensure they are not tight too much). When you press to belts against them you should feel soft resistance. If you find out the top part of the belt is more tight than the bottom part of the belt then the belt is not well tight. You have to fix it by adjusting the belt tensioners. Belts of the printers are in fact one loop which are running around gear wheels. During the time of 3D printing the belt can jump over one or more teeth of the pulley and this can cause the belt tight problem. Result of this issue is irregular layering during 3D print.

II. Check the rods

There is dirt cumulation on rods during prints. This can cause resistance at running of the printing head. Result is irregular layering. To fix the problem clean up the rods and oil them smooth. It might happen you will see the printing head is slowing down at several points (locations) while printing (or it will stop/block). This is caused by rods bending. During the time leading rods can bend. Solution is to move the printing head manually over the rods and check/find the place where is the resistance point.

III. Checking drive pulleys and gear wheels

Drive pulleys and gear wheels are driving the belts and are connected to stepper motors. Try to check if there is all tight like it supposed to be.

[Back to the top.](#)



14. Deformation - Warping

I. Heated print bed

The most simple solution is to use heated print bed. If you set the print bed temperature just below the melting point of the filament (glass transition temperature of the plastic material), then the first layer will stick correctly to the platform. The print bed temperature is normally set by slicer software. The recommended temperature of the print bed for your filament you should find on the packaging or on the spool.

II. Use adhesives to the bed

If the print still lift up especially at the edges, apply a small volume of glue or other adhesives evenly to your print bed to increase adhesion.

III. Experiment with different types of print bed

Change your printing bed to offer better grip. Manufacturers like Prusa use a PEI (polyetherimide) surface, which offers excellent adhesion – no need of additional adhesives. XYZPrinting comes in a box with some of their printers textured tape, essentially a large sheet of masking tape, and re-adding works great, but if only on unheated printing platforms. Zortrax 3D printers have a perforated printing surface, models are almost welded to this surface, which completely eliminates the problem.

IV. LEVEL THE BED !

Another cause may be the calibration of the bed. During the calibration process, check that the platform is leveled correctly and that the nozzle height is correct.

V. Improve the contact between model and the bed

A simple solution is to increase the contact between the model and the bed, and most printing software has the ability to add rafts or brims.

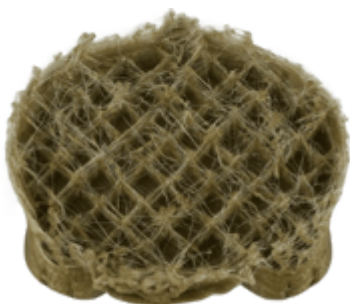
VI. Advanced temperature settings

If all options fail, you will need to see the advanced print settings on both the printer and the printing software. Try raising the bed temperature by 5 degrees.

In the Slicer, look at the fan cooling, which is usually set so that the cooling fans switch to full power at a height of about 0.5 mm. Try extending them to 0.75 so that the base layers have some time for natural cooling.

Even if your printer has a heated print pad, it is always recommended that you use adhesives and calibrate the platform level regularly.

[Back to the top.](#)



15. Broken / chaotic infill

I. Density of the infill

Check the infill density in your slicer software. Normal value is around 20%, lower than 20% might cause this problem. For the larger models you should increase this value, so the print has enough support.

II. Lower print speed of the infill

Print speed of the infill has the main effect on his quality. If the infill looks weak and poor, try decreasing his print speed.

III. Try another pattern of the infill

Mainly all slicer softwares has the possibility to change the internal structure of the infil. There can be grid pattern, triangular, honeycomb and more. Try another type of infill i fit helps.

IV. Check the nozzle

Check if the nozzle is not slightly clogged. Je možné, že tryska je mierne upchatá. Although this does not affect the printing of thicker outer walls, as the inner structure causes less flow, the filament may getting trapped.

Back to the top.



16. Gaps between infill and outer wall

I. Check the infill overlap

This is the most common problem and is really easy to solve. In your slicer, look for the "infill overlap" option and increase the value.

- in Cura its set to 15% by default. Increase it to 30%.
- In Simplify3D, you can find this option in the "Edit Process Settings> Fill> Outline Overlap" section. Increase the value again.

When adjusting this setting, always keep it below 50%, otherwise you will see the effects of overlap in the outer perimeters of the model.

II. Print infill before perimeter

If you print with a relatively thin outer wall, the fill structure may be visible through. If this occur, you can try changing the order in which the printer print infill and perimeter layers. For example, in Cura, make sure the "Infill prints after perimeters" check box is selected.

III. Increase hot-end temperature

Some of the latest special materials, such as carbon fibers, are slightly less forgiving when printed due to the carbon fibers that form part of their structure. When you print them, you will find that raising the temperature by 5-10 °C will make wonders.

IV. Decrease the print speed

You may need to rush to print, but printing quicker can cause a variety of problems if the printer is not correctly calibrated. If you need to print quickly, you can still avoid gaps by reducing the speed of the top layer.

Back to the top.



17. Visible infill from the outside

I. Shell thickness

Ensure that the value you selected for the outer layer thickness is a multiple of the nozzle size.

II. Increase the shell thickness

The simplest solution is to increase the thickness of the outer layer. Doubling the size should cover any overlap caused by the infill.

III. Print infill after perimeter

Most slicers allows you to print the infill after the perimeter.

- In Cura, open "Expert Settings" and in the infill section, check the "Infill prints after perimeters" checkbox.
- In Simply3D, click "Edit Process Settings", then select "Layer" and select "Outside-in" under "Layer Settings".

IV. Check the print bed

Check the surroundings of the model and if you see that the effect prevails on one side more than the other, it may be due to calibration. If so, perform the usual calibration process.

Back to the top.



18. Cracks in high objects

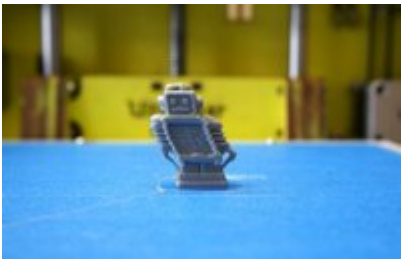
I. Hotend temperature

Start by raising the extruder temperature – at the beginning would be fine to increase it by 10 ° C. Try to keep the temperature setting within the values of the filament manufacturer, according to the label or the values on the filament packaging.

II. Direction and speed of the fan

Be sure, that your fans are on and pointing at the model. If they are, try reducing their speed.

Back to the top.



19. Print leans

I. X and Y axis check

If your print is leaning left or right you might have a problem with X axis. If your print is leaning forward or backward you might have a problem with Y axis. Finding out which axis it is you can continue fixing this problem by checking the rods, pulleys, belts, gears and bearings.

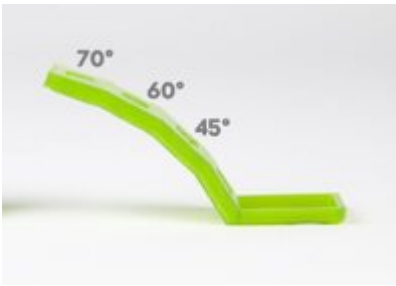
II. Belts friction

. Check all the belts to ensure none is rubbing against another part or construction of the 3D printer. Check if the belts are correctly tight.

III. Tighten stepper motors

Once you have found out which axis caused the problem you can tighten the gear to the stepper motor by hex key (Allen wrench/key).

Back to the top.



20. Messy overhangs

I. Supports

The fastest and easiest solution is to add supports. Most slicers will allow you to do this quickly. In Simplify3D, click Edit Process Settings> Support> Generate Support Material; you can adjust the quantity, pattern and settings.

In Cura, just select the desired type of support from the basic settings.

II. In-model supports

Software-generated supports can sometimes be disruptive and can lead to jamming of the support material in an impossible place to remove. A good alternative is to create your own file in your modeling application. It takes a little more skill, but it can enable fantastic results.

III. Support platform

When printing a figurine, the most common areas that cause problems are, for example, the hands. Using supports from the bed can also cause problems because they often have to travel relatively large vertical distances.

A great solution may be to create a solid block or wall under the arms and then create a smaller support between the arm and this block.

IV. Angle the walls

If you have a shelf style overhang then an easy solution is to slope the wall at 45° so that the wall actually supports itself and removes the need for any other type of support.

V. Break the model apart

Another way to look at the model is to divide it into several separate prints. For some parts, this allows the printout to be flipped and what should be an overhang will become the base. The only problem is that then you have to find a way to connect the individual parts.

[Back to the top.](#)



21. Fine details not printing correctly

I. Resolution

Increasing the resolution - a tighter, lower layer height will give successful prints a smoother finish.

II. Diameter of the nozzle

The smaller the nozzle diameter, the more detail you can print. But a small nozzle also means lower tolerances, so your machine needs to be well calibrated.

III. Printer maintenance

Any additional friction from slightly misaligned bars or loose belts will be immediately reflected on your printout. Make sure everything is tight and aligned.

IV. Clean nozzle

Ensure, your nozzle is clean before starting a detailed print.

V. Slow down your print

Decreasing the print speed – slower print has less mistakes.

VI. Good quality filament

Good quality prints has more chance to be done with good quality filament.

VII. Level the bed !!

Calibrate the printer and make sure the bed is correctly leveled.

Back to the top.



22. Ripples and "echoes" in print (Ringing)

I. Reduce the vibrations

Ensure that the surface on which the 3D printer is placed is solid and that the printer does not generate any visual vibration.

II. Check the bearings

Linear bearings wear out over time, so when the printer is turned off, make sure they are all running smooth.

III. Ensure everything is tight

It's amazing how one loose screw can affect print quality. As part of your routine maintenance, make sure everything is bolted and tight.

IV. Oil check

Check that all rods are free of dust and dirt, and then add a drop of oil so that everything is well lubricated and run smoothly.

V. Slow down

Decrease the print speed.

Back to the top.



23. Diagonal scars on print

I. Combing

Combing keeps the print head above the already printed areas of the print, and reduces the need for retraction. Although it increases the printing speed, it can cause scars. Try to turn off combing, and in most cases, this will fix the issue, but it will extend the printing time.

II. Retraction

If you turned off the combing and the problem persists, try increasing the retraction rate. If the problem persists, look for over-extrusion or nozzle temperature.

III. Extrusion

How the filament flow is set will vary depending on your printer. Reduce the flow rate by 5% and print out the calibration cube to check that the filament is extruded correctly and eliminates the problem.

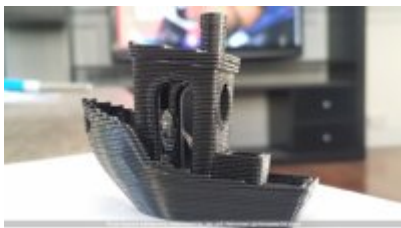
IV. Nozzle temperature

Tolerance of good quality filaments should in principle eliminate this problem, but if your filaments have been set aside for a while, exposed to moisture or sunlight, you may find that the temperature tolerances of the filaments have decreased. Lower the hot end temperature by 5° and try again.

V. Z-Lift

Filament is not the only problem that can cause this issue. If the head does not rise high enough from the print surface, then the nozzle itself may cause a scratch when moving from one layer to another. On older printers, you will need to recalibrate if Z-lift or Z-hop settings are not available, otherwise increase Z-Hop or Z-Lift in 0.25 mm increments.

[Back to the top.](#)



24. Print looks stringy and droopy (Over-extrusion)

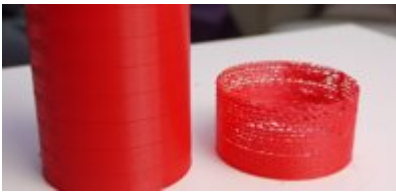
I. Extrusion multiplier

Make sure the correct extrusion multiplier is selected in your slicer.

II. Flow rate settings

If all this looks right, reduce the flow setting in your printer software.

[Back to the top.](#)



25. Under-extrusion

I. Filament diameter – slicer software

Start with the simplest problem, whether you have the correct filament diameter set in the slicer. If you are unsure of the diameter, the value, along with the recommended temperature, is usually printed on a box or spool.

II. Filament diameter - measurement

If you still do not get the desired results and the filament flow is a problem, then check the filament diameter several times with a set of calipers.

III. Hotend – debris check

Check that the nozzle is free of deposits, filament residues and dirt.

IV. Extrusion multiplier

If there is no mismatch between the actual fiber diameter and the slicer settings, then the extrusion multiplier (also referred to as flow rate or flow compensation) may be too low. Each slicer software will handle this a little differently, but the principle is to increase this value in increments of 5% until you see that the problem has disappeared.

Back to the top.



26. Print looks melted and deformed

I. Recommended material settings

This may seem obvious, but make sure you have provided the printer with the correct material details. The latest filaments temperatures are between 180 - 260 ° C or around them, so it is very easy to set them incorrectly.

II. Hotend temperature

Reduce the hotend temperature in the printer or software settings. Depending on the severity of the overheating, lower the temperature by 5 ° C.

III. Increase the print speed

Try to increase the print speed.

IV. Fans adjustement

Make sure the cooling fans are pointing at the hotend. Check that they are in the correct position and, if possible, increase their speed to increase the airflow to the cooling filament.

Back to the top.



27. Pits and hollows in top layer (Pillowing)

I. Check the orientation of the fans (ventilators position)

Reason can be cooling. Usually fans are set up for off mode or low performance at the beginnig of the printing. After few layers are printed off fans start to blow and cool the layers. Please check if your fans are blowing air during the print, especially at towards end of the printing. If yes and the pillowing persists, then check if the performance of the fans is set up correctly in your slicer software.

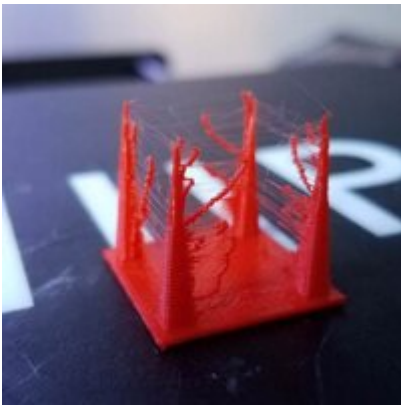
Setting fan speed (performance) in your slicer software

Why its important ? every layer of molten plastics must be cooled down short after its applied. Its to avoid molten plastics will leak in to gabs or holes of the model. Cooling fan speed or performance can be set up in your slicer software or directly during printing process by cotroler of your 3D printer. Please follow manual of your printer.

II. Top layer thickness increasing

Increasing the top layer thickness is the easiest solution. Most common softwares will enable you to do this. Check the advanced section, under the 'Bottom / Top Thickness setting'. Usual its set up fot 6 layers of material normally and up to 8 for nozzles under 0,4 mm. If your layer height is therefore set to 0.1mm then set the 'Bottom / Top Thickness setting' to 0.6mm. If the effect of pillowing still persist then increase to 0.8mm.

Back to the top.



28. Stringing

I. First of all ENABLE retraction !

What does it mean „retraction“ By easy words its sucking back the filament in to the nozzle while extruder is moving from end point of the print to start point of the print. If not set up correctly you will see „hairs“ on your print. Retraction is important to achieve high quality prints. Retraction can be managed through your slicing software.

Usually default option set up in Cura for example is more than sufficient but of course its depending on filament material your print is made of.

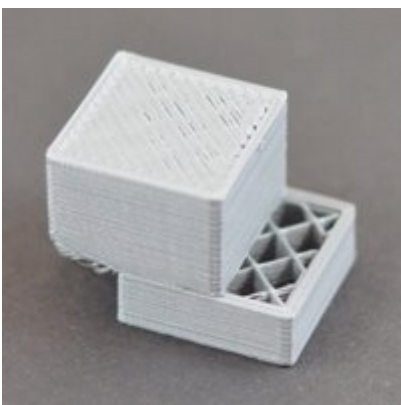
II. Travel distance or what is the minimum travel

Stringing (hairy print) is mostly caused by wrong set up of the retraction or too fast movement of the extruder head. This movement is called „travel speed“. Reducing the minimum travel is usually the first and quickest solution to avoid stringing if the standard retraction isn't working well. Put the value down in 0.5mm increments until the stringing has stopped.

III. Cutting strings ????

The easiest solution how to get off the string is to cut them off after the print is finished. Its not very attractive solution but its relatively easy to do. This option should be really considered as the last one. You just have to take something sharp and carefully cut the strings off. Just watch out your finger and dont cut your self.

Back to the top.



29. The print has offset in random places

I. Is your printer stable based ?

First of all check if your printer is stable based and its not moving during the print. Ensure during printing no vibrations or other outside circumstances will appear which could influent the print.

II. Heating bed is stable fixed (secured) ?

There are a lot of types of detachable print beds. Its a big advantage at removing prints and it avoids damage to the printer but its also causing over time clips and screws can work loose. Make sure that when you reinstall the print platform it's clipped or bolted well in place to avoid any slip or another movement.

III. Upper layers are wrapped ?

If you set up too much cooling while print, then the upper layer of the print can warp very easilly. Please watch out ! Cooling is causing warping and warping is causing rising of the layers and this will lead to an contact (mostly fatal contact by destroying your model or damage of your hot end) to the nozzle as it moves. Of course in most cases the print will release from the platform, but if it doesn't it can really casue damages of teh print or of the printer parts. In the upper layers try reducing the speed of the fans slightly if your prints are suffering from warping.

IV. Reducing print speed ? why not

When you increase the temperature and speed up flow of the filament you will be able to print really fast. There is big BUT ! High speed printing is bringing a lot of problems which can occure. If you hear a clicking during printing this could be a sign that the printer is going to fast. That means filament is running faster in to the noozle than viscosity change of the filament. Reducing printing speed can help a lot. You can adjust this parameter in your slicer software or directly in printer settings. For more info please see user manual of your 3D printer.

V. Belts check

Shifting layers ? immediatelly check the belts please ! Focus on tension in each belt because it must be the same, if its not then you'll need to adjust the belt position. Please be aware to tight belts can damage your stepper motors.

VI. Check , Clean and lubricate rods

There is dirt cumulation on rods during prints. This can cause resistance at running of the printing head. Result is irregular layering. To fix the problem clean up the rods and oil them smooth.

It might happen you will see the printing head is slowing down at several points (locations) while printing (or it will stop/block). This is casued by rods bending. During the time leading rods can bend. Solution is to move the printing head manually over the rods and check/find the place where is the resistance point.

[Back to the top.](#)



30. Bridges are messy

I. Bridges distance check

Make a test print by printing columns and bridges of different distances to check how far your printer will go- Best is to start with a 5cm gap and then increase. Anything between 5 and 10 is perfect. 15 cm would be more than perfect .

II. Supports adding

Make your printing easy as possible. Add supporting in your slicer at making the G-Code file.

III. Increase fan speed

Increase the extrusion fan speed to ensure quick cooling of the melt filament, the faster the filament is cooled down the larger the bridge can be created.

IV. Reduce extrusion and print speed

Fast printing is absolutely not compatible in relation to bridges. Why? Filament must cool down , it need enough time for it. At fast print you can forget about. So, slow down the print, while printing models applying bridges print .

V. Simplify3D can help

This software is able to set up optimal cooling settings and extrusion slowing down once the bridging of the model is needed.

[Back to the top.](#)