iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** I **Month of publication:** January 2018

DOI: http://doi.org/10.22214/ijraset.2018.1149

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor :6.887
Volume 6 Issue I, January 2018- Available at www.ijraset.com

Troubleshooting for FDM Technology

Ishtiaq Ahmed[1], Mohammed Shoaib Shariff[2], M Syed Ismail Zeeshan[3], Prashanth S [4]
[1, 2, 3, 4] *UG Students, Department of Mechanical Engineering, Atria Institute of Technology, Bengaluru, Karnataka, India*

Abstract: 3D printing is a technology where the objects are by the addition of multiple layers one top of another to form a solid object. Different processes involved in 3D printing technology are as follows (1) FDM (Fused Deposition Modeling), (2) SLS (Selective Laser Sintering), (3) EBM (Electron Beam Machining), (4) LOM (Laminated Object Manufacturing), etc. In this paper, we are focusing on the problems faced while 3D printing and solutions to those problems. The most common problems faced during printing may be Clogged Extruder, layer shifting, grinding filament, Weak Infill, overheating, etc.The technology adopted by us is FDM for which the problems and solutions are discussed in this paper. Without resolving all these problems, it is not possible to get a good print. By resolving all these problems, we can improve the quality of the 3D printed parts that are being printed.

Keywords -3D printing, filament, extruder or nozzle, FDM (Fused Deposition Modeling)

I. INTRODUCTION

A. Fused Deposition Modeling (FDM)

This is a process by which a machine deposits a material (Thermoplastics or wax). One-layer top of other layer of the same material, in order to create a solid joint by adhesion or heat. fused Deposition Modeling (FDM) was produced by Stratasys in Eden Prairie, Minnesota. In this procedure, a plastic or wax material is expelled through a nozzle that follows the part's cross-sectional geometry layer by layer [1]. The fabricate material is generally provided in fiber shape, however a few setups use plastic pellets nourished from a container. The nozzle contains resistive warmers that keep the plastic at a temperature simply over its liquefying point with the goal that it streams effortlessly through the nozzle and structures the layer. The plastic solidifies instantly subsequent to spilling out of the nozzle and bonds to the layer underneath. Once a layer is fabricated, the stage brings down, and the expulsion nozzle stores another layer. This will dramatically decrease the bond strength between the layers and overall build quality. For example, if a 3D printer is using a 0.6mm nozzle, then the maximum layer height should not exceed 0.5mm[2]. In the X-Y plane, 0.001inch determination is achievable. A scope of materials are accessible 25 counting ABS, polyamide, polycarbonate, polyethylene, polypropylene, and venture throwing wax (16, 17). Today in FDM process there are many problems facing by us such as extruder is clogged, extra filament etc. So to over this we have come up with various problem and their solution by FDM process:

II. TROUBLESHOOT

A. Clogged Extruder

The 3D printer must soften and expel numerous kilograms of plastic over its lifetime. To make things more convoluted, the greater part of this plastic must leave the extruder through a little gap that is just as large as a solitary grain of sand. Definitely, there may come a period where something turns out badly with this procedure and the extruder is never again ready to push plastic through the nozzle[3]. These jams or obstructs are more often than not because of something inside the nozzle that is hindering the plastic from uninhibitedly expelling. While this might plague the first occasion when it happens, yet we will stroll through a few simple investigating steps that can be utilized to settle a stuck nozzle. This problem is due to the following reason:

1) Manually pushing the filament into the extruder

2) Reload the filament Clean out the nozzle

B. Layer Shifting

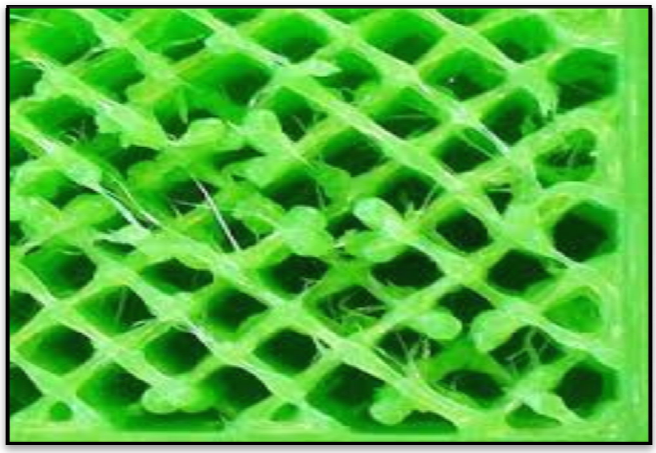
Many of the 3d printers nowadays use an open loop control system that they have no feedback to the actual location of the nozzle. The printer attempts to move the nozzle to a location but it doesn't happen due to powerful stepper motors that will drive the printer. if something goes wrong the printer would never detect it .and this causes misaligned layers in the print. The layer shifting problem can be avoided as what problem we may occur and how to fix it given below:-

1) Nozzle moves too fast
2) Mechanical or Electrical issue

C. Weak Infill

The infill inside your 3D printed part assumes a critical part in the general quality of your model. The infill is in charge of associating the external shells of your 3D print, and should likewise support and upper surfaces that will be imprinted over the infill. On the off chance that your infill gives off an impression of being powerless or stringy, you might need to change a couple of settings inside the product to add extra quality to this segment of your print.

1) Trying different infill pattern
2) Reducing the print speed
3) Increasing the infill extrusion width



D. Grinding Filament

The 3D printers use small drive which grab the filament and sandwich it against another bearing .The sharp teeth that allows the filament to push it forward or backward .If the filament is unable to move then the drive gear will keep spinning and will grind away plastic from the filament as this situation is termed as stripped .To overcome this problem there are some fix to the problem given below

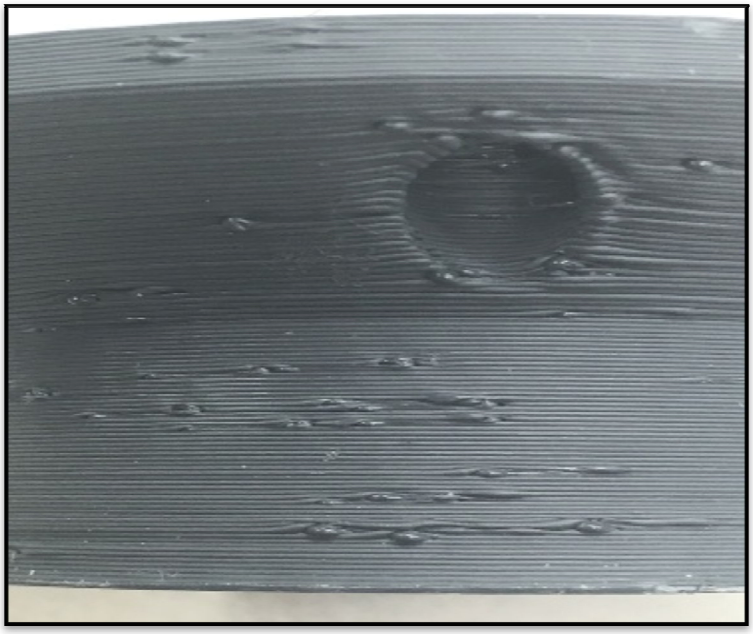
1) Increase the extruder temperature
2) Print too fas
3) Check for nozzle clog

E. Blobs and Zits

In 3D print, the extruder should always stop and begin expelling as it moves to various bits of the fabricate stage. Most extruders are great at delivering a uniform expulsion while they are running, be that as it may, each time the extruder is killed and on once more, it can make additional variety. For instance, in the event that you take a gander at the external shell of your 3D print, you may see a little stamp at first glance that speaks to the area where the extruder began printing that segment of plastic. The extruder needed to begin printing the external shell of your 3D show at that particular area, and after that it in the end came back to that area when the whole shell had been printed. These imprints are normally alluded to as blobs or zits. As you can envision, it is hard to join two bits of plastic together without leaving any check at all, yet there are a few devices in 3D printer that can be utilized to limit the presence of these surface imperfections.

1) Movement Behavior
2) Choose start point that is closest to specific location

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor :6.887
Volume 6 Issue I, January 2018- Available at www.ijraset.com



F. Stringing

Stringing will occur when the small strings of plastic is left behind on 3d printed model .This is due to the plastic coming out of nozzle while the extruder is moving to a new location. The common setting which is used to avoid excessive stringing is retraction where the extruder is done printing at one part of the model, the filament is pulled backwards to the nozzle to act against oozing. To avoid or eliminate stringing there are several settings that can be used given below :-

1) Retraction Distance
2) Retraction Speed
3) Temperature is high

G. Overheating

The plastic which comes out of the extruder its temperature is between 190 to 240 degreesCelsius. If the plastic is hot then it can easily be formed into different shapes. As it cools it quickly becomes solid and retains the shape so we need to correctly balance between heating and cooling temperatures so that the plastic can easily flow through the nozzle but can solidify as it poured on the heat bed. To overcome the overheating problems there are several causes and how to prevent them which are given below :-

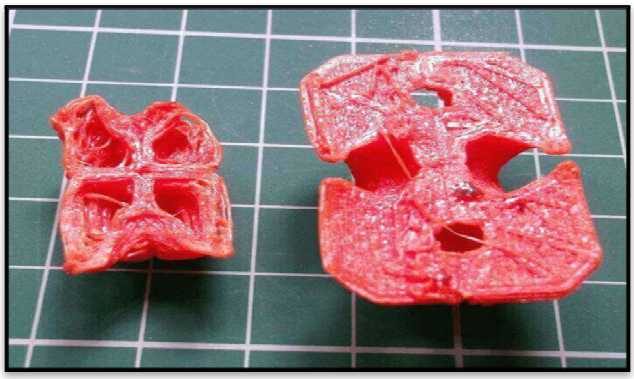
1) Insufficient cooling
2) Print at too high temperature
3) Printing too fast

H. Poor Surface Above Supports

one of the huge preferences of 3d printer is the ability to make creative help structures which empower you to make incomprehensibly complex parts that would be hard to make by and large. For instance, on the off chance that you have a precarious shade or part of your model with nothing beneath it, at that point a help structure can give an establishment to these layers. The help structures made by 3D printer are expendable and can be effectively isolated from the last part. In any case, contingent upon your settings, you may locate that a few changes are expected to idealize the surface quality on the underside of your parts, appropriate over the help structure establishment. we will illuminate the key settings underneath and how they can impact your prints.

1) lowering your layer height
2) Horizontal Offset from the part
3) Use of multiple extruder
4) Support Infill Percentage

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor :6.887
Volume 6 Issue I, January 2018- Available at www.ijraset.com



I. Extruder Not Extruding Enough Plastic

The software used for 3D printing have some settings to determine how much plastic to be extruded. However, the 3D printer does not give any feedback that how much material must be extruded from the nozzle[4]. There may be some chance that less plastic exiting the nozzle than what software expects (else it would be a under extrusion process). In this case we may notice gaps between extrusion of each adjacent layer. There are some typical ways to solve this problem

1) diameter of the filament is not correct
2) high extrusion speed

J. Inconsistent Extrusion

For 3d printer to have the capacity to make exact parts, it should be fit for expelling an exceptionally steady measure of plastic. On the off chance that this expulsion fluctuates crosswise over various parts of your print, it will influence the last print quality. Conflicting expulsion can for the most part be distinguished by viewing your printer intently as it prints. For instance, if the printer is printing a straight line that is 20mm long, however you see that the expulsion appears to be fairly rough or appears to fluctuate in estimate, at that point you are likely encountering this issue. We have outline the most widely recognized foundations for conflicting expulsion, and clarified how everyone can be tended to.

1) Filament becomes tangled
2) Low layer height
3) Low quality filament
4) Mechanical issues with extruder

K. Extruder Not Extruding At The Beginning Of Print

This is the very common problem faced by the 3D printer owners, but this problem can be easily resolved. But If this issue is not resolved then it may affect the print quality. There are some possible causes if the extruder is not extruding at the beginning of the print. The possible solutions for this problem may be due to

1) extruder not loaded before the printing
2) gap between the bed and nozzle is too close
3) blockage of extruder
4) grinding filament against the drive gear

L. Print Not Adhering To The Bed

The first layer of the print must be strongly adhered to build platform because it will the foundation for any print model. If the first layer is not adhering to the bed it will create a problem at the end of the print. There are some typical ways to solve this problem

1) bed not levelled
2) extruder extrudes far away from the bed
3) first layer being printed too fast
4) temperature settings
5) extra materials to use for the adhesion(tape, glues, etc)

M. Extruder Not Extruding Enough Plastic

The software used for 3D printing have some settings to determine how much plastic to be extruded. However, the 3D printer does not give any feedback that how much material must be extruded from the nozzle. There may be some chance that less plastic exiting the nozzle than what software expects (else it would be a under extrusion process). In this case we may notice gaps between extrusion of each adjacent layer. There are some typical ways to solve this problem

1) diameter of the filament is not correct
2) high extrusion speed

N. Too Much Extrusion Of Filament Material

To obtain a good print quality the extrusion must be precise. The software constantly keeps tracking the nozzle to extrude the correct amount of filament material. But however the printers have no way to monitor that how much plastic is in flow. If the extrusion flow settings are not properly configured then the printer may extrude excess plastic as expected by the software. This excess extrusion may affect the dimension of the model. In order to solve this problem, the extrusion speed must be reduced in the printer settings to obtain the filament material as required.

O. Gaps And Holes Present In The Top Layer

In the 3D printed parts there are solid shells created to cover the partially hollow interior in order to save the material. For example, the interior part of an object may use 20% of the interior area which is filled with a solid plastic material and rest of the portion is air. This technique can help in saving the material and time for printing, while can still give a strong part. The interior part may be anything but the exterior must be a solid. To do this we must make settings in the software like top-bottom layers and the infill required as per requirement. However what ever settings we use, we may notice that the layers on the top of the print are not completely solid. We may notice some gaps and holes between the extrusion of solid layers. There are some typical ways to fix this problem

1) insufficient top solid layers
2) low infill percentage
3) under extrusion

P. Layer Sepration And Splitting

3D printer works by building object one layer at a time so after a successive layer sis printed on the top of previous layer and at the end we will get the desired shape. The final layers must be strong and reliable to bond the layers below it. if it doesn't it will split or separate. The layer separation causes and to resolve the causes are given below:

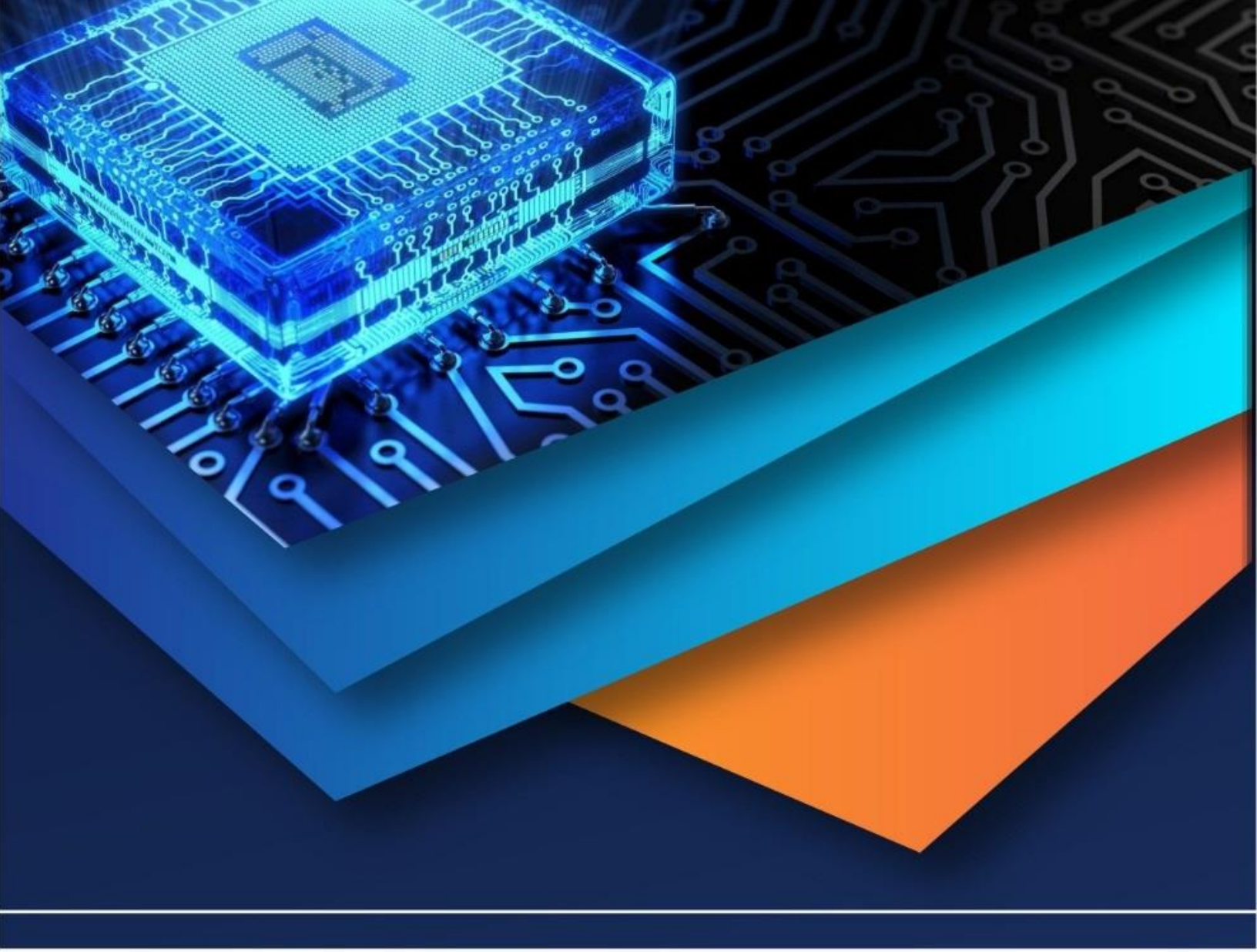
1) Layer height is too large
2) Print temperature is too low

III. CONCLUSIONS

In this paper we have focused on the problems faced while 3D printing. The problems focused in this paper are Clogged Extruder, layer shifting, grinding filament, Weak Infill, overheating, etc. The outcome of this paper was to list out the all the problems faced during 3D printing and solution to those problems. By solving all these problems, the 3D print quality can be improved and strong parts can be obtained.

REFERENCES

[1] Dr. Rajashekar Patil, Ishtiaq Ahmed, Mohammed Shoaib Shariff, Syed Ismail Zeeshan, Prashanth S, Harsha N and Pradeep Kumar K, Design and Fabrication of Portable 3D Printer, International Journal of Mechanical Engineering and Technology 8(11), 2017, pp. 129–135. http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=8&IType=11
[2] Fused Deposition Modeling (FDM) Mechanism by Fawaz Alabdullah, International Journal of Scientific & Engineering Research, Volume 7, Issue 5, May-2016 41 ISSN 2229-5518
[3] file:///C:/Users/HP/Downloads/D-R-516-Banding-and-Clogged-Nozzles.pdf
[4] http://www.egr.msu.edu/classes/ece480/capstone/spring14/group08/MartezAN.pdf



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY