Modelling and Analysis of 3D Printer Thermal Liquefier

Pushkal Awasthi[1], Rajeev Srivastava[2], Saty Dev[3]

[1]Department of Mechanical Engineering,
Motilal Nehru national institute of technology Allahabad, Prayagraj 211004, India
pushkalme0081@gmail.com

[2]Department of Mechanical Engineering,
Motilal Nehru national institute of technology Allahabad, Prayagraj 211004, India
rajmnnit@mnnit.ac.in

[3]Department of Mechanical Engineering,
Motilal Nehru national institute of technology Allahabad, Prayagraj 211004, India
satyadevyadav08@gmail.com

Abstract: ***The fused deposition modelling (FDM) technology is widely preferred advanced digital manufacturing system. The governing component of heating and deposition of the FDM system should manage the quality of product and energy saving. The quality of the final product also depends upon the proper heating, melting and deposition of material. Thermal liquefier is one of the most influencing components of FDM for governing of filament feed and heat energy saving. The efficient design of liquefier governs the temperature which controls continuous flow of material and blockage at nozzle tip and heat sink. It also maintains the desired minimum temperature at top of liquefier which prevents the lifecycle and functioning of surrounding hardware from overheating. The objective of this study is to modelling and analysis of a modified thermal liquefier to improve its performance for low melting point materials and to avoid the existing problems. In this paper models of thermal liquefier with different heat sink designs (circular heat sink with and without perforation) are developed through the SolidWorks software and the finite element analysis of the model is performed by ANSYS software. The liquefier with a perforated heat sink model found with reduced temperature on the top of liquefier which is desirable for the performance of the system. The modified model avails more part of heat for part fabrication and minimizes the waste. It also reduces the weight of the system which helps the easy movement of the print head.***

Keywords: FDM, liquefier, heat sink, modelling, simulation

1. Introduction

Additive manufacturing (AM) is an advanced technology to make parts from 3D digital data by joining the material in layers. AM technology is capable to fabricate intricate and complex parts with minimum waste of material and energy. AM is capable to use a variety of materials application including plastics, ceramics and metals. Therefore this technology can help users to think more innovative designs and customized production at comparatively low cost. AM supports the pollution free environment, strong economy and healthy societal factors. These capabilities of AM technologies aids sustainability in manufacturing. There are wide range of application areas including aerospace, automobile, medical industry, and education [1] [2] [3][4].

Extrusion-based AM process is one of the most familiar technologies. It enables companies to fabricate customized parts for different applications. The control of thermal parameters are most important for qualitative production without affecting the system component and energy consumption. The thermal liquefier is the main heat sensitive part of FDM technology. The heat sink of thermal liquefier plays an important role in pushing the filament towards the nozzle without its premelting and maintaining the temperature between part bed and extruder. Therefore, an advanced model of thermal liquefier is required to be developed which will be suitable for low melting point material, and also provide more difference in temperatures at the nozzle and surrounding of liquefier. The present study is carried out for the modelling and analysis of a thermal liquefier with the advanced design of heat sink. The results are compared with the findings of published studies.

1.1 FDM technology and its physical mechanism

In FDM process, more commonly material used is a polymer, in which filament material is feed into heat sink by a feed mechanism and then passing through heat break, reaches at heat block where the temperature is near about melting point range. At heat block, the filament is converted into the semisolid form of material and it extruded through liquefier nozzle and laid down in layers to shape a part.

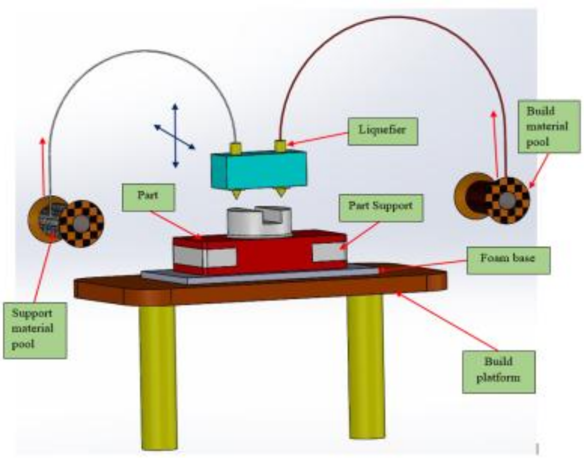

Figure 1: FDM processing components

1.2 Liquefier mechanisms

In the extrusion-based additive manufacturing process, FDM is the primary method of the manufacturing process. In this AM process, the filament in wire form is entering through the top of liquefier by using feeding mechanism via the heat sink and heat break and reaches to heat block where the wired filament is to be melt and the melted filament is extruded through nozzle set down layer by layer to fabricate the part.

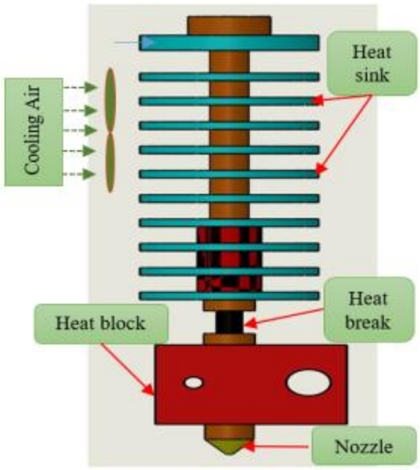

Figure 2: Parts of liquefier

The properties of the finished part mainly depend upon input factors that are the temperature of the nozzle tip, cooling fan speed, nozzle diameter, bonding's between layers, filament diameter and feed rate. Out of these factors, Temperature plays an important role in both part properties as well as liquefier performance. If the temperature is more than the melting point of filament material inside the liquefier heat sink, the filament will melt inside the heat sink before reaching the heat block and due to this filament is not worked as a plunger and it creates a blockage. And filament material will not move next to the heat sink.

1.3 Significant role of liquefier

The geometry of liquefier plays a significant impact on the polymer filament melt behaviour in the melt-chamber. Liquefier is an essential component of FDM. It has four parts i.e. Heat sink, nozzle, heat beak, and heat block. For better performance of FDM, liquefier should maintain the following essential properties…

- ❖ The temperature at the liquefier nozzle must be kept constant for continuous flow of material and should not create a barrier at the tip of the nozzle.
- ❖ The temperature must be properly decreased along the liquefier from hot end to cold end and should maintain ambient temperature at its top of liquefier.
- ❖ In heat sink, the temperature should be kept below the melting point of polymer filament material.
- ❖ Heat break should be made of low conductive material i.e. stainless steel.

If liquefier will not maintain the above properties, the following problems will arise in the FDM printing process due to temperature variations…

- ❖ If the liquefier nozzle temperature is fluctuating, then filament material is not melted uniformly inside the nozzle. Due to this, melted filament material is not in the form of homogeneous form of state, which causes certain problems like porosity, and have not maintained a uniform property of end products.
- ❖ If the heat break is made of high conductive material, then there will be more heat transfer upwards from heat block. It require more energy/power from heating resistor to maintain nozzle at constant temperature.
- ❖ In heat sink, if the temperature is more than the melting point of polymer filament material, the filament will melt inside the heat sink, which further will causes two problems.
  a) In order to extrude the filament material melt, the feed polymer filament should be a solid form, is used as a plunger. This is not achieved if the temperature is more than the melting point of the filament.
  b) If polymer filament melts inside the liquefier heat sink, it creates blockage and not to flow material uniformly in a proper way[8][9].

The methodology is to analyze the performance of FDM liquefier by reviewing its temperature behaviour, and major attention on heat dissipation of convective approach at liquefier heat sink during fabrication of 3D printed parts, is discussed.

2. Methodology

The different steps in the algorithm of modelling and analysis have been explained in this section.

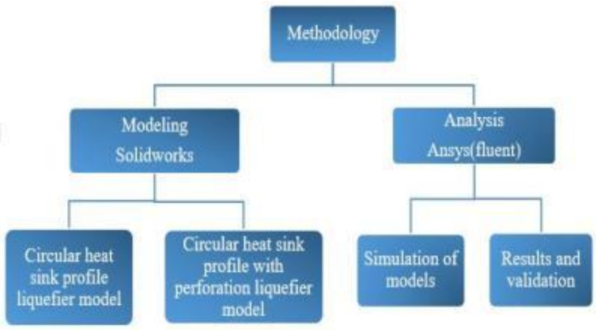

Figure 3: Methodology for modelling and analysis of liquefier

2.1 Finite element model

The thermal effects of FDM process are governed from the two heating mechanisms. In the first mechanism, liquefier heat block is heated by a heating resistor which is embedded inside heat-block. In the second mechanism, the conducted heat of heat block is dissipated from the heat sink by using the forced convective method with the help of a cooling fan[8]. A finite element model is generated to measure its thermal performance of extruder during the printing process. In this study we have taken the variable parameter is the temperature on top of liquefier surface. In addition with in this study, we developed two FEM models, the first one is perforated heat sink liquefier model and the second one is non-perforated heat sink liquefier model. The preferred software to implement this finite elements analysis (FEA) is ANSYS FLUENT 18.2.

2.2. Liquefier modelling

The whole geometry of liquefier system is created in solid works CAD software and simulation is carried out in ANSYS. Two CAD models [figure 4 (a) & (b)] are developed, which are shown in the figure below.

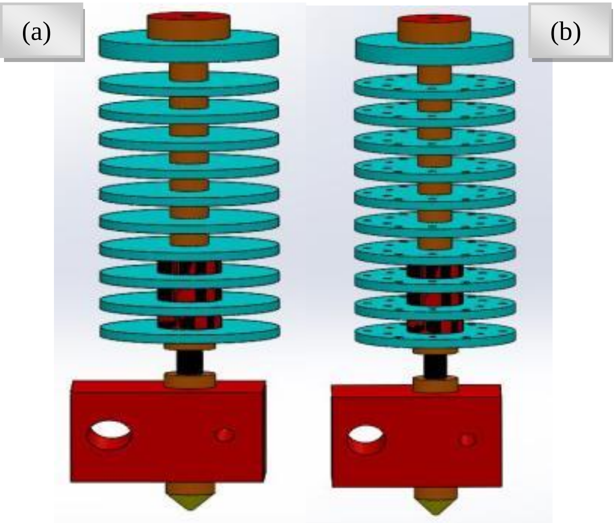

Figure 4: (a) circular heat sink liquefier, (b) circular heat sink with perforation liquefier

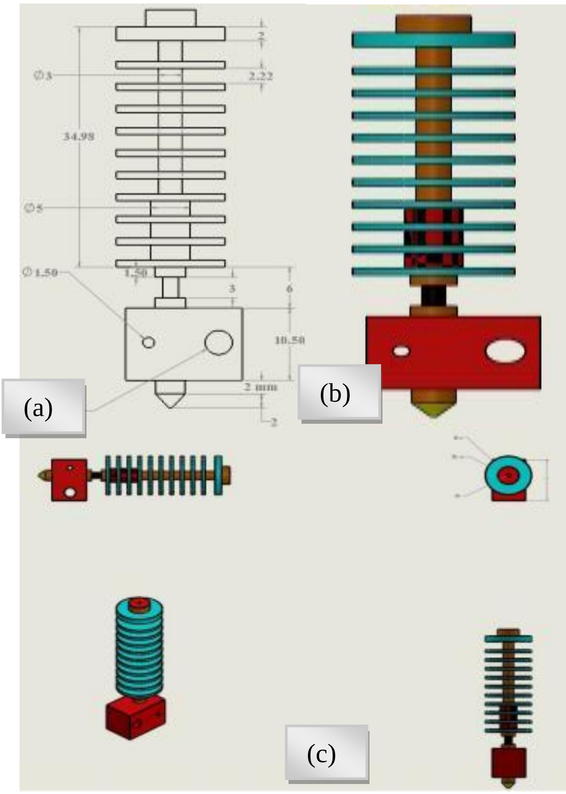

Figure 5: (a) (b) & (c) shows diferent views of liquefier

At simulation, eight different domains have been considered with different physical characteristics. In eight domains, four are a heat sink, heat block, heat break and nozzle, and other four domains are defined by ansys software. These are surrounding air, fluid domain where surrounding air flows, and two heat flow domains.

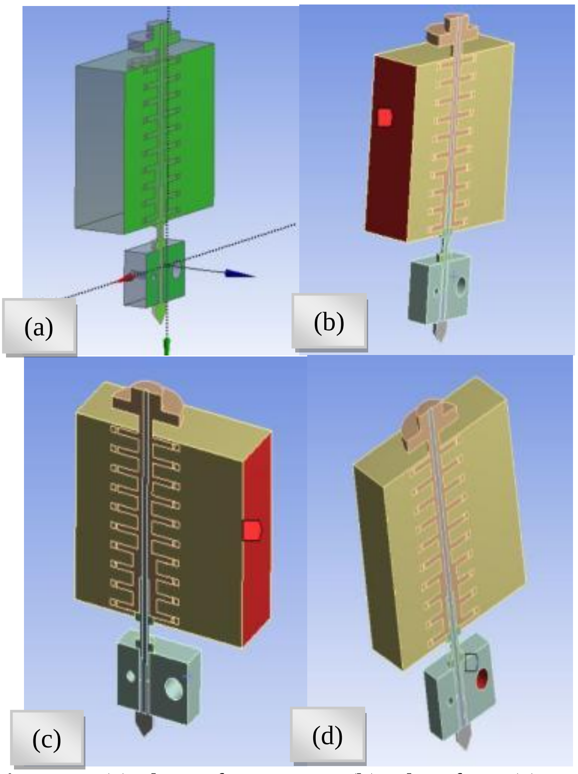

Figure 6: (a) plane of symmetry, (b) inlet of air, (c) outlet surface of air, (d) heating resistor surface

2.3. Convergence of mesh

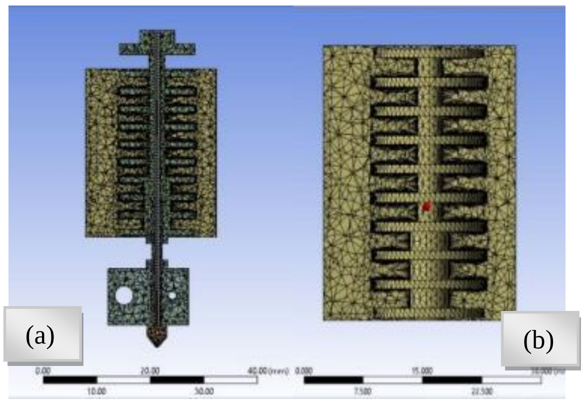

Figure 7: (a) fine mesh with 338456 elements, (b) fine mesh of enclosure

2.4. Governing equations of liquefier

The governing mathematical equations of heat transfer is defined already in the selected appropriate modules which are available in ANSYS. To solve this problem we have taken some appropriate modules that are explained below…
In this study, we have taken heat transfer in both conduction and convection. Because heat transfer through the heat sink surface is with the forced convection method, that's why here we selected 'conjugate heat transfer and turbulent flow

analysis modules'. And the second module is 'steady state heat transfer'.

The general equations for heat transfer are following:

$$Q_{COND} = K.A.\frac{\Delta T}{L} \quad (1)$$
$$Q_{CONV} = h.A.\Delta T \quad (2)$$

This heat is convected through heat sink to decrease the temperature of liquefier. Here we can say that

$$Q_{Conv} \propto A \quad (3)$$
$$S.A. \propto \frac{1}{\Delta T} \quad \&$$
$$Q_{Cond} \propto \frac{1}{L} \quad (4)$$

Here 'A' is the surface area of convected zone i.e. area of the heat sink. And 'L' is the length of the liquefier heat sink. Here from the above equation, it can say that if we increase the surface area, change in temperature is decreasing. And if we increase in length of liquefier, the rate of conduction of heat is decreased.

2.5. FEM simulation

The FEM models were executed at different cases, by varying the air velocity, from 0 m/s to 1 m/s, generated by the cooling fan. And collected solutions showing on graphs represents the temperature profile of each case. The simulation required at least 12 hours of CPU processing on a 4 Intel(R) Core (T) i5 CPU m560 @3.2GHz with 12 GB RAM, on a Windows 10 operating system with 64 bits.

2.6. Numerical solutions: The FLUENT software which is based on the finite volume method has been utilized to examine the model with boundary conditions. In the pipeline of CFD analysis the RNG k- ε turbulence module with standard wall function for Reynolds-average Navier-Stokes (RANS) model has been used to result in the desired performance[10]. During the iteration of the process to monitor the convergence rate the governing equations have been used. It is found that the residuals less than 10^{-10} which shows the convergent solution. The non-dimensional wall distance (NDWD) i.e. y+ (ρUfy/μ) used in turbulent flow to find out first layer thickness. The value of the factor y+ (ρUfy/μ) has considered 10 for the simulation purpose.

3. Results and discussion

In this study two analysis has been performed, the first is to find the temperature at top of liquefier with varying the air velocity of the cooling fan where it is used for force convection heat dissipation at the heat sink. Second, a comparative study between the perforated heat sink liquefier model and without perforated heat sink liquefier at a particular air velocity.

The Figure 8(a), (b), (c), (d), (e) are showing the analysis of circular heat sink model with different fan speed, carried out in ANSYS. Table 1 is indicating the different values of temperature at top of liquefier at varying fan speed (0m/s to 1.5 m/s). Near to stable position of fan the temperature is maximum. Whereas, the temperature is declining up to a certain limit by increasing the fan speed. The decrease in temperature is not considerable after a speed of 1 m/s.

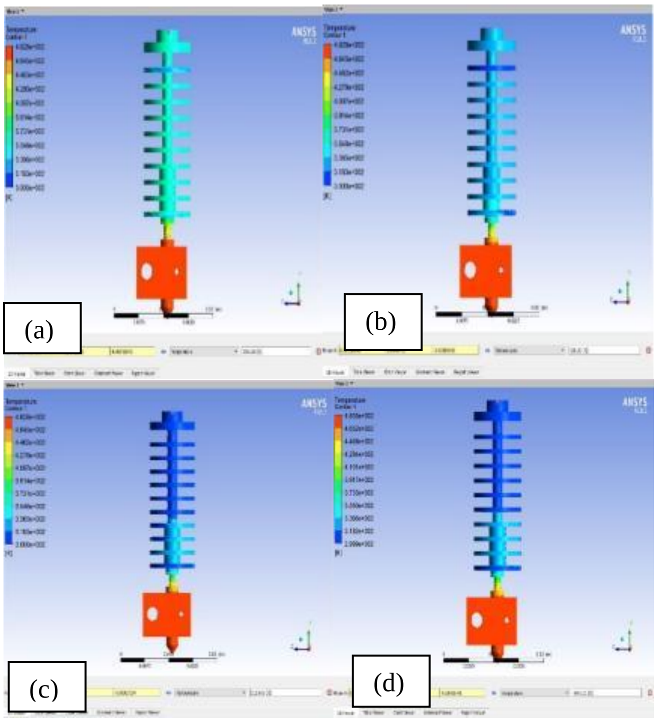

Figure 8 [a, b, c, d, e]: Analysis of circular heat sink model with different fan speed

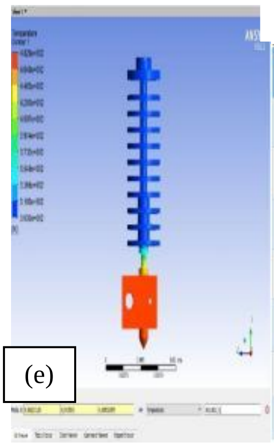

Table: 1 Temp at different fan speed

Fan speed [m/s]	Temperature at top [K]
0	355.182
0.25	312.933
0.5	309.311
0.75	306.463
1.0	303.802
1.5	303.801
2.0	303.081

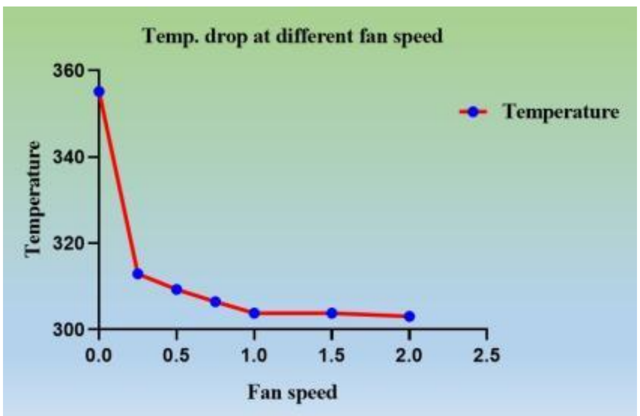

Figure 9: Temperature drop at different fan speed

3.1. Comparison between circular heat sink profile liquefier with and without perforation

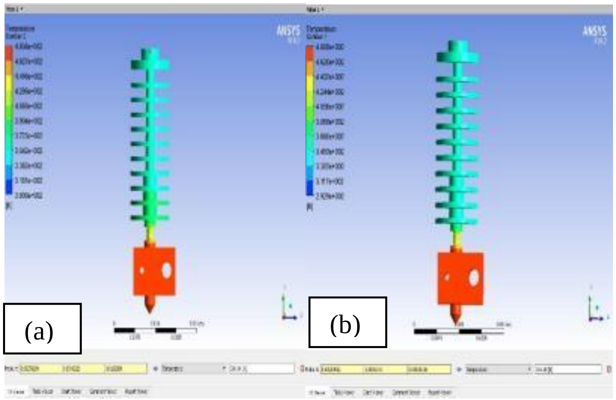

Figure 10: Comparison between circular heat sink model with and without perforation

In this comparison, it has been observed that temperature at top of liquefier is less in case of perforated heat sink liquefier model [figure 10(a)] than non- perforated heat sink model [figure 10(b)]. This shows that if the temperature of the heat sink is less than the melting point of the filament, the filament is not melted inside the heat sink. It is beneficial for liquefier, that filament is used as a plunger and it would not create blockage inside the liquefier sink. So it can be concluded that perforated heat sink liquefier model is dissipating more heat than non-perforated liquefier model and causes effective cooling.

4. Validation of CFD model

In order to validate the current CFD model of the heat sink which is used in order to analyze the heat dissipation rate from the heat sink. R. Mesa experimentally calculates the temperature at the heat sink surface for the different velocity of air which is coming from the exhaust fan. This velocity of air is responsible for forced convection at the heat sink. Mesa had given the mathematical model which is used to calculate the velocity of air coming from the fan in order to get the desired temperature. The mathematical model by Mesa is

$$Y = 30.574 . X^{-0.567} \quad \text{(i)}$$

Where Y is the temperature in Kelvin at top of liquefier and X is fan speed in m/s.

Table 2: Comparative study of temp drop at a different fan speed

Fan speed	Present data	Existing data 1	Existing data 2
0.25	312.933	345.2	367
0.5	309.311	323.5	339
0.75	306.463	309.4	318
1.0	303.802	303	311

The existing data [1 & 2] of liquefier temperature is taken from reference [2] for comparison.

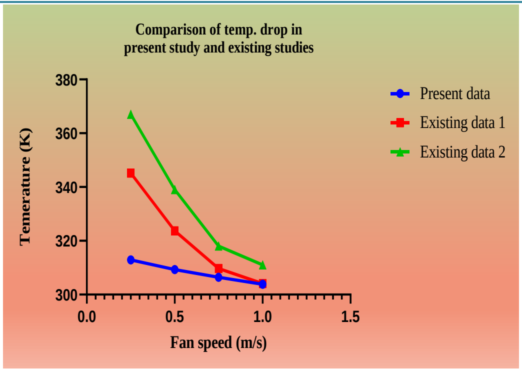

Figure 11: Comparative study of temp drop at the different fan speed

5. Conclusions

In this study two models of heat sink, thermal liquefier (Circular heat sink model with and without perforation) have been modeled and analyzed for comparative thermal behaviour. The convective dissipation mechanism plays an important role in different air inflows during simulation.

Following conclusions have been drawn from this study:

- ❖ The circular shape is found to be most favourable for heat transfer in liquefier.
- ❖ The temperature drop in case of a perforated heat sink model is larger than without the perforated model.
- ❖ The temperature drop also depends on fan speed. The temperature decreases up to 1.0 m/s of fan speed. Thereafter no change in temperature even on increasing the speed.
- ❖ Circular heat sink with perforation improves the heat transfer rate and also saves the material consumption which further favourable for system performance by reducing the weight

Finally, it is found that the proposed perforated heat sink model is more efficient than the existing model. The existing model shows the temperature approximately from 481 K to 312.95 K. The model with perforation drops the temperature to 303.93 K which shows the high performance of analyzed liquefier.

The future studies can be conducted with different parameters variation including the diameter of perforation and number of holes.

References

[1] H. Bikas, P. Stavropoulos, and G. Chryssolouris, “Additive manufacturing methods and modelling approaches a critical review,” pp. 389–405, 2016.

[2] M. Kumar, K. Singh Verma, S. Nagpure, and M. Tech scholar, “Numerical Simulation of Rep-Rap 3D Printer Liquefier to Determine the Thermal Behavior of Heat Sink,” 2017.

[3] S. H. Huang, P. Liu, and A. Mokasdar, “Additive manufacturing and its societal impact : a literature review,” pp. 1191–1192, 2013.

[4] A. D. Amico and A. M. Peterson, “An adaptable

FEA simulation of material extrusion additive manufacturing heat transfer in 3D," *Addit. Manuf.*, vol. 21, no., pp. 422–430, 2018.

[5] R. Huang, M. Riddle, D. Graziano, J. Warren, S. Das, S. Nimbalkar, J. Cresko, and E. Masanet, "Energy and emissions saving potential of additive manufacturing : the case of lightweight aircraft components," vol. 135, 2016.

[6] F. Rayegani and G. C. Onwubolu, "Fused deposition modelling (FDM) process parameter prediction and optimization using group method for data handling (GMDH) and differential evolution (DE)," no. January 2016, 2014.

[7] Y. Zhang and Y. K. Chou, "MSEC2006-21132," pp. 1–8, 2016.

[8] R. Jerez-Mesa, J. A. Travieso-Rodriguez, X. Corbella, R. Busqué, and G. Gomez-Gras, "Finite element analysis of the thermal behavior of a RepRap 3D printer liquefier," *Mechatronics*, 2016.

[9] M. Nikzad, S. H. Masood, and I. Sbarski, "Thermo-mechanical properties of a highly filled polymeric composites for Fused Deposition Modeling," vol. 32, pp. 3448–3456, 2011.

[10] T. Y. Emre Koc, Onur GUNEL, "Investigation of thermal effect on 3d printer liquefier of different cooling fan nozzle geometry," in *international conference on energy and thermal engineering: ISTANBUL 2017 25-28*, 2018.

Author Profile

Pushkal Awasthi did his graduation in Mechanical Engineering from United College of engineering and research Allahabad, India, and pursuing M. Tech in Production Engineering from MNNIT Allahabad India. His areas of interest are Software based modelling and simulation (solid works, ANSYS, Abaqus) & additive manufacturing.

Rajeev Srivastava is working as a Professor in Mechanical Engineering Department at Motilal Nehru National Institute of Technology Allahabad, India. He did his under graduate in Mechanical Engineering from Institution of Engineers India Calcutta, M. Tech in CAD/CAM and Ph.D. in Mechanical Engineering from Motilal Nehru Regional Engineering College, Allahabad, India. He has over 25 years of teaching and research experience. He has published papers in many journals of international repute. His current research interest are Computer Aided Design & Manufacturing, Modelling and Optimization of Manufacturing Processes and Mechatronics.

Saty Dev did his B.Tech in Mechanical Engineering from Gautam Buddh Technical University Lucknow, Uttar Pradesh, India (Formerly UPTU) and received M.Tech. From Kurukshetra University, Kurukhsetra, Haryana, India. He is currently pursuing Ph.D. in Mechanical Engineering at Motilal Nehru National Institute of Technology Allahabad, UP, India. He had worked as an Assistant Professor in MIIT Meerut and SITE Meerut (Affiliated to UPTU) and ACEIT Jaipur, (Affiliated to RTU) Rajasthan, India for over three years. His area of interest includes optimization of additive manufacturing process, multi criteria decision modelling and system design and analysis.