EFFECT OF AIR JET VELOCITY AND TURBULENT INTENSITY ON BREAKAGE OF FUEL JET IN AIR BLAST ATOMIZER USING POPULATION BALANCE MODEL

Author[1], Author[2], Author[3]

Shashi Kumar[1], Assistant Professor, Axis Institute of Technology and Management, Kanpur, India.
Anoop Kumar Kushwaha[2], Assistant Professor, Axis Institute of Technology and Management, Kanpur, India.
Rahul Sonkar[3], Assistant Professor, Axis Institute of Technology and Management, Kanpur, India.

Abstract

In gas turbine application the combustion characteristics and emission generally depends on the atomization process. There are many types of atomizing devices like pressure atomizer, air assist atomizer, air blast atomizer, dual orifice, rotary atomizer, plain jet air blast atomizer etc. Present study is based on CFD analysis of the performance parameters of co-axial two fluid air blast atomizers. Air blast atomizers are widely used in aircrafts, marines and gas turbines for atomizing the fuel into small droplets. Efficiency and emission of gas turbine and aero engine is affected by the size and velocity of droplets. In the present study, effect of air inlet velocity and turbulent intensity of air jet on SMD (Sauter Mean Diameter) of water droplet have been investigated. Population balance model with discrete phase method are used for tracking droplets and their size. Realizable k-ε turbulence model is used.

Keywords: *Sauter Mean Diameter (SMD); Population balance model; Discrete Phase Method; Air Blast Atomizer; Air jet velocity.*

---

1. INTRODUCTION

The conversion of bulk fuel into small droplets is known as atomization process. In air blast atomizer large amount of heat is produced by the good mixing of fuel and air. German scientist J. Sauter developed a method to measure particle size known as SAM (Sauter Mean Diameter). SAM is defined as the diameter of sphere that has the ratio of same volume to surface area as a particle of interest. The break-up of liquid sheet or liquid jet to large droplets and further breakage of large droplets into smaller ones depends on the critical Weber number (We) defined as

$$We_{crit} = \frac{\rho_g u_r^2 d}{\sigma}$$

Where ρ_g is the density of the atomizing gas, u_r is the relative velocity between air and liquid, σ is the surface tension of the liquid and d is the particle diameter. In the absence of aerodynamic forces, surface tension force becomes dominant and turns the liquid into spherical form which has the minimum surface energy. First experimental study on co-axial atomizer was performed by Nukiyama et al. [1]. Lefebvre [2] observed the effect of velocity of gas on the SAM of drops which is produced by liquid jet breaking up in high velocity air flow. Further, Engelbert et al. [3] studied the break up phenomena of a liquid jet in co-axial air blast atomizers. The effects of air and liquid properties and atomizer dimensions on the spray characteristics of plain-jet air blast atomizers were studied by Rizk et al. [4]. LUI et al. [5] investigated the effect of liquid jet diameter on performance of coaxial two-fluid air blast atomizer with water–air system in a wide region of Liquid/gas mass flux ratio (0.137 $<m<$ 15.6). OMNI et al. [6] observed the distribution of size and velocity of droplets formed at the end of primary breakup region. Verga et al. [7] analyzed small-diameter liquid jet exposed to a large-diameter high speed gas jet experimentally. They used Flow visualization and particle-sizing techniques for examining the initial jet breakup process and primary liquid atomization. Jaffar et al. [8] analyzed the atomization process occurring in a plain jet air blast atomizer numerically. They used population balance model for determining the droplet size. They compared SMD values obtained numerically with past experimental data only at certain flow conditions. Smith et al. [9] used an Eulerian multiphase flow model coupled with a population balance model for numerical simulation of atomization in swirling. Vast research work has been done on air blast atomizer experimentally. Due to experimental setup costs we used CFD analysis as it is accurate & cheap. Atomization process has been studied numerically using VOF model and Lagrangian particle tracking method. In present study Population Balance model is used. The main objective of this study is to find the droplet size (SMD) of water in air blast atomizer by using discreet method and the effect of variation of flow parameters on SMD and the effect of turbulence intensity of air jet on SMD of droplet.

2. METHODOLOGY

To obtain the complexities of the combustion process and following chemical reactions, water was selected as the fuel. Analysis of air blast atomizer is a multiphase problem as it involves liquid & gaseous phases so Eulerian Multiphase Model coupled with Population Balance Model was used in ANSYS FLUENT. Population Balance equation states that the number of droplets is conserved in a particular domain. The balance equation is:

$$\partial/\partial t(n(v,t)) + \nabla.[un(v,t)] = vg(v')\beta(v/v')n(v't)dv' - g(v)n(v,t)$$

Breakage frequency is represented by g(v'), that represents fraction of droplet with volume (v') breaking per unit time. Droplet breakage kernel is represented by β(v/v') and indicates probability that a droplet with volume V forms from fragmentation of droplet with volume V'. For the purpose of validation of CFD code, SMD was taken as the parameter to compare the results. The solver used for analysis is ANSYS Fluent 15.0. Three turbulence models i.e. Standard k-ε, RNG k-ε and Realizable k- ε were tested therefore it was found that the realizable k-ε model comes very close to the SMD values found in Zafar et al. [8] for the same grid and the boundary conditions. There are three methods for solving the population balance equation - Discrete method, Quadrature method of moments and Standard method of moments but in present study we use the Discrete method and other methods will be considered in future work. The discrete method is based on representing the continuous particle distribution in terms of discrete classes or bins. A large number of size bins may be required in order to obtain an accurate solution which may lead to computational complexity therefore in present study number of bins is 7 used. In present study atomization process is studied in two dimensions numerically. The geometry consists of two air jet of diameter of 6mm and one liquid jet of 4mm Whole solution domain is of 40×100 mm. A schematic geometry of co-axial two fluid air blast atomizers is shown in Fig.1

Fig-1: Schematic geometry of air blast atomizer

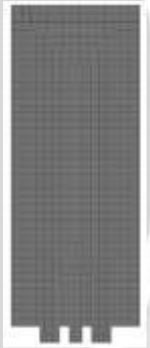


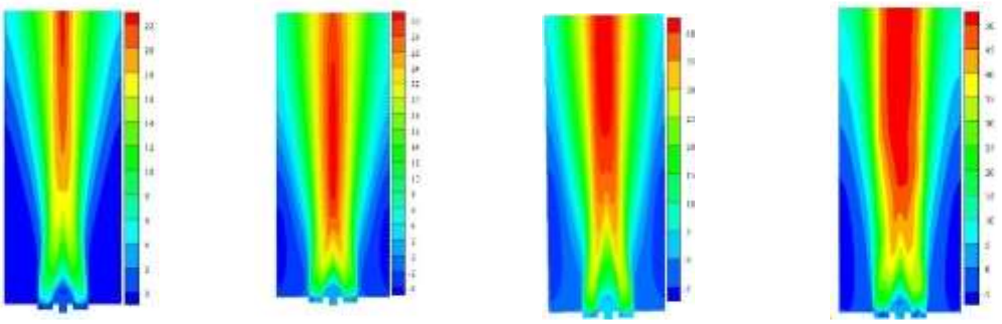
Fig-2: Mesh geometry

Breakage problem which is a time dependent phenomenon so unsteady solver is used for numerical analysis. Time step size is 0.00001 s and convergence criteria for each step is 0.000001. Pressure is atmospheric at the outlet of the pipe. Unstructured grid generated for present study analysis. Second order upwind differencing scheme is used for numerical analysis. Meshing of the geometry is shown in Fig.2. Simulations were carried out for two fluid co-axial air blast atomizer. The results obtained for two different flow conditions of air, one includes variation of turbulence intensity of air jet and other includes variation of air jet velocity on droplet size of water. SMD value has been determined at various points which are lying along the axis of water jet for different velocities in both flow conditions. In present study five air jet velocities were taken, these are 60 m/sec, 80 m/sec, 100 m/sec, 120 m/sec and 140 m/sec. In both flow conditions water inlet velocity is 1m/s and there is no change in population balance model data. For each condition simulation were run for 2000 time interval.

3. RESULTS AND DISCUSSION

The results have been obtained for two parametric study -

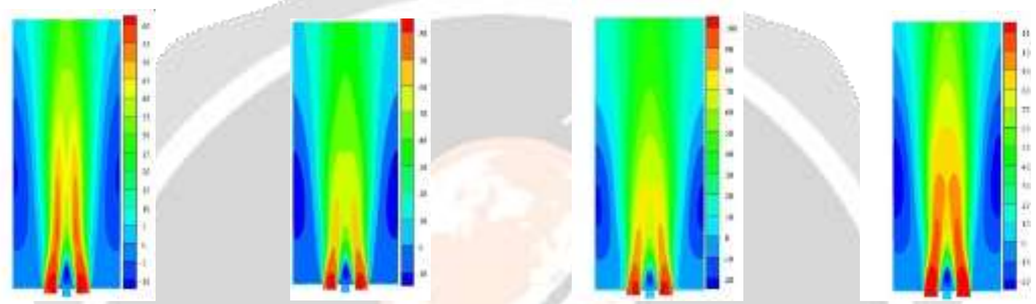
(a) **Effect of Air Jet velocity-** Effect of air jet velocity is first parametric study. Outcome of air jet velocity on water jet velocity contour is shown in Fig.3.



(a) at 60m/s **(b) at 80m/s** **(c) at 100m/s** **(d) at 120m/s**

Fig.3 Contours of variation of water jet velocity

In following contours, the water flow velocity increases with the increase in the velocity of air. At 60m/s maximum water jet velocity is 22m/s, at 80m/s maximum water jet velocity is 30m/s, at 100m/s maximum water jet velocity is 40m/s and at 120m/s maximum water jet velocity is 50m/s. Therefore, momentum transfer is increased from air to water. Fig.4 shows the variation of air jet velocity inflow domain.



(a) at 60m/s **(b) at 80m/s** **(c) at 100m/s** **(d) at 120m/s**

Fig.4 Contour of variation Air Jet Velocity in flow domain

In fig.4 (a) velocity of air flow in downstream direction is the lowest because of low air jet velocity. But from fig.4 (b) to fig.4 (d) velocity of air flow in downstream direction goes on increasing. So momentum transfer to water in fig.4 (d) is highest which results more drag force on water flow so breakage of water droplet of larger size into smaller size is highest and this fragmentation process occurs quickly. The effect of air jet velocity on SMD of water droplets as shown in Fig.5.

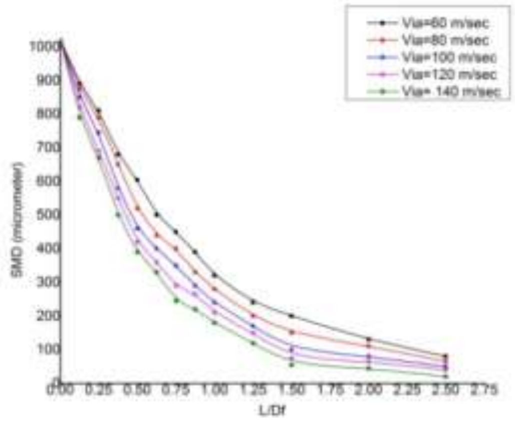


Fig.5 Graph between SMD and $\frac{L}{D_f}$for different air jet velocity.

Fig.5 shows the Graph between the value of SMD and L/D_f for different air jet velocity where L/D_f represents ratio of length of flow domain to diameter of flow domain and L/D_f has been taken along the axial line of water jet. SMD values are decreasing as L/D_f increases as shown in Fig.5, due to drag force, droplets of this size class break into lower size class. These lower size class droplets move into downstream direction, there is again drag force of air on these lower size class droplets, which further break into even lower size class, this process happens up to outlet. Thus at 60m/s droplet size is largest and for at 140m/s droplet size is smallest. At same L/D_f , SMD values are decreasing as velocity of air jet increases as shown in Fig.5.

(b) Effect of Air Jet Turbulent Intensity

The effect of air jet turbulence intensity on droplet size of water is another parametric study. As turbulence intensity of air jet increases, interaction of air flow with water increases in flow domain which results more momentum transfer from air to water so drag force on water flow in flow domain increases which results breakage of water sheet into smaller droplet size and this breakage occurs quickly. For analysing the effect of turbulence intensity of air on SMD of water droplet, five turbulence intensities of air were taken. These are 5%, 10%, 15%, 20%, 25%. Fig.6 shows the Graph between the value of SMD and L/D_f for different turbulent intensity where SMD values are decreasing as L/D_f increases. Due to drag force droplets of this size class break into lower size class therefore at outlet very low size droplets are present. At same L/D_f , SMD values are decreasing as turbulent intensity of air jet increases as shown in Fig.6.

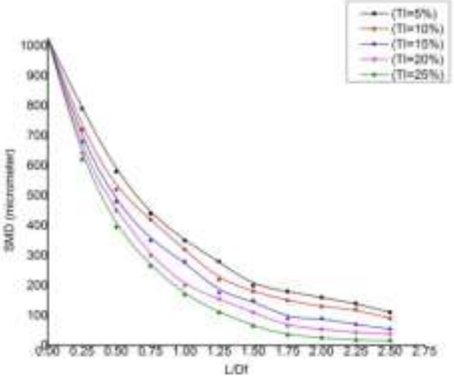


Fig.6 Graph between SMD and $\frac{L}{D_f}$ for different turbulent intensity.

Results show that the SMD of droplets is higher near the inlet of injector and goes on decreasing in axial direction giving minimum droplet size at outlet.

4. CONCLUSIONS

Efficiency of the gas turbine is depends upon the quality of atomization. If the SMD of the droplet at the outlet is large thereby quality of atomization is very poor. Large droplets reduced the surface area of liquid fuel which deceases the contact of fuel with air therefore burning of fuel is poor which results in decreased heat release. Atomization quality will be good when the droplets are fine. Fine droplets are increased the surface area of fuel thus the result is more contact area with the air which increases heat released therefore the efficiency of the combustor is increases. Hence it is concluded that for fine atomization, velocity of jet and turbulence intensity of air jet will be increased as per results which is confirmed from this study.

5. REFERENCES

[1]. S. Nukiyama, Y. Tanasawa, Experiments on the atomization of liquidsin an airstream, Trans. Soc. Mech. Eng. Jpn. 5 (1939) 68–75, Atomization and Sprays, Hemisphere Publishing Corporation, New York, 1989.

[2]. C. Engelbert, Y. Hardalupas, J.H. Whitelaw, Breakup phenomena in coaxial airblast atomizers, Proc. R. Soc. Lond. A 451 (1995) 189–229

[3]. N.K. Rizk, A.H. Lefebvre, Spray characteristics of plain-jet airblast atomizers, Trans. ASME J. Eng. Gas Turbines Power 106 (1984) 639–644.

[4]. Liu, Hai-Feng, Wei-Feng Li, Xin Gong, Xian-Kui Cao, Jian-Liang Xu, Xue-Li Chen, Yi-Fei Wang, Guang-Suo Yu, Fu-Chen Wang, and Zun-Hong Yu. "Effect of liquid jet diameter on performance of coaxial two-fluid airblast atomizers." *Chemical Engineering and Processing:Process Intensification*45,no.4 (2006): 240-245.

[5] Fathollah Ommi, Movahednejad, Ehsan, and S. Mostafa Hosseinalipour. "Prediction of droplet size and velocity distribution in droplet formation region of liquid spray." *Entropy* 12, no. 6 (2010): 1484-1498.

[6]. C.M. Varga, J.C. Lasheras, E.J. Hopfinger, Initial breakup of a small diameter liquid jet by a high-speed gas stream, J. Fluid Mech. 497 (2003) 405–434.

[7]. Aly, H.S., Lazin, T.M., Eldrainy, Y.A., Jaafar, M.N.M., 2009. Mathematical modelling of droplet atomisation using the population balance equation. International Conference on Signal Processing Systems, ICSPS 2009. Institute of Electrical and Electronics Engineers, New 681 York, 955-959.

[8]. S. Smith, N.P. Rayapati, M.V. Panchaghula, J.Peddieson, J.Short, international Journal of spray and combustion dynamics, vol.3, 19-44.