Numerical Investigation of Spray Characteristics of a Swirl Airblast Atomizer for Varying Geometry and Flow Conditions

Jyotichandra Jingade, S.N. Sridhara
1-Student, PG Studies, 2-Head of the Department Mechanical and AE Rotating Machinery Design
M.S.Ramaiah School of Advanced Studies, Bangalore

Abstract
The main function of the fuel atomizer in gas turbine combustors is to deliver finely atomized fuel to primary combustion zone in order to secure a high rate of heat release. In order to satisfy the legislation on pollution control, special attention must be paid to the aeriation of the fuel spray, since this can have a major effect on the rates of formation of NOx. The combustion characteristics and emission from gas turbine greatly depend on the spray characterization issued from the atomizer. The subsequent combustion process in a gas turbine engine is significantly affected by the initial droplet distribution. Therefore, understanding the underlying mechanisms that govern fuel atomization in airblast atomizer is critically important in developing next generation gas turbine engines.

In the present study the characteristics of spray produced by airblast atomizer have been reviewed numerically using the commercial code FLUENT. The numerical results are validated and benchmarked against experimental results. Further the spray characteristics in terms of Sauter Mean Diameter (SMD), mass flux distribution at varied axial distances, velocity ratio, air to liquid ratio, momentum ratio, swirl numbers, varied geometry and flow parameters has been discussed.

The finer SMD is demonstrated for each value of increase in swirl number. Smaller droplets start to appear further downstream in addition to significant increase in radial spread of droplets with shorter atomization axial distance for each value of increase in swirl number. Employing both inner and outer air swirl in combination with inner air co-rotating and outer air-counter-rotating has proved to have excellent enhancement in atomization by producing finer SMD when compared to single swirl atomizer. Inner air diameter variation plays significant improvement in the atomization, and it is noticed that reducing inner air diameter below 3mm for the fixed value of flow rate will not enhance the performance of atomization. Significant finer SMD is demonstrated as the flow rate of air is increased. Also an increasing trend of SMD is noticed when the flow rate of liquid is increased, which is mainly because, increasing mass flow rate of liquid in a atomizer would increase the momentum of the liquid jet making it more stable to the disturbances and hence leads to poorer atomization.

Key Words: Atomisation, characterisation of spray, Airblast atomiser

1. INTRODUCTION
Meeting emission requirements and constantly improving combustion efficiency remain to be two primary challenges for the development of next generation gas turbine engines. To achieve lower levels of NOx, significantly and for rapid and uniform fuel/air mixing, it is important to optimize the fuel atomization process and combustor aerodynamics. Because of its desirable attributes, such as lower fuel pressure requirement, larger flow turn-down ratio and lower pollutant emissions, the airblast atomizer has been considered as advanced fuel injection device and is widely used in gas turbine engines [1]. The subsequent combustion process in a gas turbine engine is significantly affected by the initial droplet distribution. Therefore, understanding the underlying mechanisms that govern fuel atomization in airblast atomizer is critically important in developing next generation gas turbine engines. The computational simulation of spray combustion in gas turbines engines use droplet distribution in a spray emanating from airblast atomizer as initial conditions, therefore establishment of an advanced computational fluid dynamics for design/simulation will be useful in improving atomizer design.

Atomization is the process of transformation of bulk liquid into spray. Because of random nature of the atomization process, the resultant spray contains a wide spectrum of droplet sizes. The factors that govern the atomization process include physical properties of the fluids involved, geometrical aspects of the injector design, and the pressure drop across the atomizer and the ambient conditions. In case of fuel combustion, effective atomization is expected in terms of very fine droplets that results in increase in specific surface area and achieve high rates of mixing and evaporation. Hence it is important to be able to control drop sizes in sprays, which would require better understanding of basic atomization process [2, 3].

Extensive research on airblast atomization has yielded considerable information on the effect of variation in liquid properties, air properties and atomizer design features on the drop size distributions produced in the spray [4]. Also theoretical, experimental and mathematical analyses on the mechanism of atomization have been carried out since Taylor (1933) and Rayleigh (1987) detailed reviews of the earlier work have also been published by Lefebvre (1989). The summery of such studies throws light on physics of atomization and also provides data and correlation needed for further computational approach.

SASTECH 57 Volume 7, No.1, April 2008
The detailed review and studies on plain jet breakup done by Sridhara et al., 2000, [2, 5], has given physical insight into the parameters influencing the quality of atomization. His studies showed that the performance of the airlift atomization can be significantly improved by varying flow parameters i.e., increasing the air-liquid ratio, relative velocity between air-liquid and air to liquid velocity ratio also shown the effect of atomizer geometry on the quality of atomization. However his studies depicted that how all above parameters are influencing the liquid jet breakup phenomena. The experimental and analytical study by May Y. Leong et al., 2000 [6] has focused on the process of liquid jet breakup and atomization. The atomization of a liquid jet occurs when the relative velocity between the injected liquid and the surrounding gas medium induces instabilities that lead to breakup. A subsequent transition in breakup modes (depleted in Fig.1) occurs when the influence of the dynamic pressure of the ambient gas is increased with an increase in the jet velocity. Lefebvre 1989 and May Y. Leong et al., 2000 [3, 6, 7, 8].

Fig 1. Different Regimes of jet disintegration in a quiescent environment [10]

The process of airlift atomization involves the use of two fluids (Fig. 2a). The liquid jet is forced to break up because of the shear forces caused by a high velocity stream of co-flowing air. The advantages of airlift atomization in gas turbine applications include such factors as requiring lower pressures to deliver the fuel, and accomplishing partial premixing of the air and liquid fuel prior to mixing with the bulk air flow for combustion (Lefebvre, 1989 [3]). On the other hand, the plain-jet airlift atomizer is easier to fabricate because of its simpler design. Figure 2.3(b) depicts representative schematics of both types of atomizers for comparison.

Fig 2. (a) Atomization of a liquid jet with the aid of a co-flowing air stream [8] (b) Prefilming airlift atomizer [3]

In an airlift atomizer the liquid fuel is first spread out into a thin annular, low velocity liquid sheet and then is exposed to high-speed airstreams on both sides. The liquid sheet breakup occurs due to kinetic energy of high-speed airstreams, and its breakup is responsible for spray formation and determines the resultant spray characteristics such as mean droplet diameters and droplet size distribution. [1, 9]. The performance of airlift atomizer is usually expressed in terms of variation of Sauter Mean Diameter (SMD) as a function of air-to-liquid mass ratio, relative velocity, air-to-liquid velocity ratio and air-to-liquid momentum ratio [10].

Computational fluid dynamics (CFD) analysis plays a valuable role in the fuel injector design process for both aerospace and industrial gas turbines. It is used to give insight into the design that cannot otherwise be obtained or that would be more costly and time-consuming to obtain by other means. Single phase CFD analyses are commonly used in the gas turbine fuel nozzle design process. These applications of CFD have proved to be very successful in identifying potential weaknesses in the designs. The situation is different when it comes to the actual liquid fuel atomization process. Where there is two phase interaction between the liquid fuel and air. In this case one must turn to experimental analyses and rely upon past experience, “cut-and-try” methodology and design of experiments [11]. Whereas these methods have yielded successful designs, it is preferable to have an analytical tool to gain insight into how and why geometry changes influence performance and give guidance to the design process.

The literature reveals the need for, Need for numerical study on droplet distribution, numerical study on effect of flow parameter variation on quality of atomization, numerical study on effect of geometry variation (inner and outer air core diameter) on quality of atomization and study of effect of swirl on the atomization In the current work, numerical simulations are carried out to provide an insight into physics of the flow field within a torque converter. An effort is also made to understand the flow details at different operating conditions and suggest methods to improve torque converter performance.

2. BASELINE CONFIGURATION AND VALIDATION.

The baseline configuration is a setup consists of fuel injector mounted at the end of an air box spraying downward into a tank. The experiments were conducted with MIL-C-7024-B fluid [2]. The density, surface tension and viscosity of this liquid at standard temperature of 298K are 765 kg/m3, 0.0025N/m and 0.00092 kg/m-s respectively. Inner cone of 10mm, an inter medium cone for the liquid flow with half cone angle of 45 deg. And 40mm long external cone with a half cone angle of 45 deg. The inner and outer Injector diameter is 15mm and 15.9 mm respectively (Fig.3). And the liquid sheet thickness is 0.45mm. The inputs for the inner and outer core airstreams are at velocity of 3m/s and liquid mass flow rate of 0.000405 kg/s is used for the experiment [11].

The same model (3D-CFD model) is used for the benchmarking validation configuration of airlift atomizer in FLUENT. The grids were generated for this case with structured hexahedral grid consisting of approximately of 600,000 cells. The grids used for turbulence (Realizable K-ε model) modeling approach with grids tightly spaced near walls to ensure the Y+
values for the first grid cell off the wall are less than 50. The spray simulation is carried out using discrete phase modeling technique in FLUENT.

The published experimental measurements made with airblast atomizer include instantaneous drop size, averaged over plane 51 mm downstream of the atomizer exit. Fig. 4 shows the comparison between measured and predicted SMD variation. The differences between the numerical and experimental results are 5% to 8%. Hence considering the experimental uncertainties of 5%, the agreement between the predicted and measured values is encouraging. Therefore the results of benchmarking validation with the available experimental measurement suggest that model used for the primary and secondary atomization is valid.

![Fig 3. Hybrid atomizer configuration used for experimental test.](image)

![Fig 4. Validation Against Calculated and Measure SMD](image)

It is also observed that simulation of airblast atomization with co-axial airstreams, in which inner air stream is co-rotating and outer air stream is counter rotating with fuel enhances the quality of atomization when compared to the two co-axial airstreams are co-rotating with the fuel.

3. DETAILS OF THE ATOMIZER MODEL CONSIDERED FOR NUMERICAL STUDY.

Fig. 5 shows the schematic diagram of the atomizer used throughout the work. The flow configuration consists of three coaxial cones, namely: i) inner cone; ii) an intermediate cone for liquid flow and iii) external cone. The air is fed through the inner and outer cones separately with swirl imparted to the inner as well as outer airflow by means of flat vanes at predetermined angle. The overall degree of swirl was characterized through a non-dimensional swirl number S. The input conditions are the liquid mass flow rate, liquid sheet thickness and corresponding resultant velocity.

3.1 Specification of the Injector

The four different models are used for the computational study is as show in the Figure 6.

![Fig 5. Schematic diagram of the nozzle [9]](image)

1. The first configuration (see Figure 6a) is the baseline configuration with no change in geometry which is used for computational simulation for following case studies.

   1) With no swirl for both inner and outer air core
   2) With swirl, on only outer air core

2. Second atomizer configuration (see Figure 6b) is the baseline configuration with no change in geometry which is used for computational simulation for following case studies.

   1) With swirl imparted on both inner and outer air core, where inner air core is co-rotating and outer air core is counter rotating with fuel.
   2) With swirl, on both inner and outer air core along with varying flow parameters i.e., variation of air and water mass flow rates.

3. Third atomizer configuration (see Figure 6c) is used to evaluate the effect of variation of inner air core exit diameter on the quality of atomization keeping outer air core diameter constant i.e., 40mm. While swirl imparted on both inner and outer air core where inner air core is co-rotating and outer air core in counter rotating with fuel.

4. Fourth atomizer configuration (see Figure 6d) is used to evaluate the effect of variation of outer air core exit diameter on the quality of atomization keeping inner air core diameter constant i.e., 5mm. While swirl imparted on both inner and outer air core where inner air core is co-rotating and outer air core in counter rotating with fuel.

3.2 Details of the Numerical Model

The governing system of equation (continuity, Helmholtz vorticity, Equation for Two-Phase and
transport equations) with implicit formulation is solved using the steady state analysis by segregated solver with pressure velocity coupling by the SIMPLE method. Also the standard pressure equation, second order upwind equations for momentum, turbulent kinetic energy and turbulent dissipation rate are used for the discretization solution controls. The two equation Realizable k-ε turbulence model is used. The grid for these analysis were generated using ANSYS 10 which is has interface between ANSYS to FLUENT. The grid considered was hexahedral cell type. The mesh size ranges from 650000 to 899153.

The air velocity range of outer air is varied between 20m/s to 30m/s with 2% turbulent intensity. And the mass flow rate of inner air was varied from 1.92m/s to 2.88g/s with turbulent intensity of 10%. The constant pressure boundary conditions are treated with specified zero shear. The realizable k-ε turbulence model gives a more accurate prediction of the spreading rate round jet than the standard k-ε turbulence model. The solution controls with default under relation factors are used, and the standard convergence criteria of residual being 1e-06 is used for all the calculations.

The effects of discrete phase trajectories on the continuous phase has been included in the calculation. Default values for all model constants and dimensionless parameters have been taken except inner and outer diameter of annular liquid passage, liquid flow rate, liquid injector half cone angle, relative velocity and liquid injection position. The number of particle streams considered was 60. This option controls how many parcels of droplets are introduced into the domain. For this problem, the injection should begin at t=0 and not stop until long after the time period of 40milliseconds. The large value of stop time (e.g., 100s) has considered ensuring the injection would essentially never stop.

The computations for all atomizer configurations were performed using realizable k-ε turbulence model, considering spray is done to the open room. In the model, constant pressure boundary conditions were imposed as show in Figure 7 to capture the ambient room effect on the atomization process. All solutions are independent of grid sizes.

5. RESULTS AND DISCUSSIONS

The computational fluid dynamic studies were conducted for four different model configurations which are explained in section 3. And the results of these configurations i.e. influence of the swirl parameters; influence of atomizer geometry variation and influence of flow parameters on the performance of atomization is discussed in detail in terms of dropsize distribution & dispersion, liquid mass distribution, velocity ratio, air to liquid ratio and momentum ratio.

- **Influence of Swirl on Spray Characterization**

The Sauter mean diameter (SMD) results were extracted 50mm, 75mm and 100mm downstream of the injector. The axial distribution of SMD and its variation with swirl number is shown in Fig 8.
Fig 8. Effect of swirl number on the SMD
(Swirl imparted to only outer air stream)

The CFD simulation shows the dispersion of water droplets when air is swirling as shown in fig. 9. This makes a qualitative comparison to the one shown in by M. Aziz, L.S. Carvalho 2001 [9]. It is also evident from the Figure 6.1 that the increase in swirl caused more radial spread of droplets. This is due to the increase in angular flux of air which increases the tangential component of the velocity [2].

Fig. 10 shows the features of the dispersion of droplets for the case where both inner and outer air swirl are present. Because of the contribution from both (inner and outer air stream) swirling airstreams growth rate of the secondary helical mode is much higher than that of the either axisymmetric or first helical mode. This demonstrates that a combination of the inner and outer air swirl which is much more effective in promoting the instability of the liquid than a single stream with air swirl and will lead to severe improvement in airblast atomization. However it is revealed from the experimental studies [1] that a combination of co-rotating inner airstreams and counter-rotating outer air stream with respect to the rotational direction of liquid sheet produces the finest spray.

Fig. 11 shows the distribution of droplet in terms of SMD along the radial direction extracted at 75mm axial distance from the nozzle exit. It can be noticed that using combination of inner and outer air swirls the radial spread of droplets are increased. Also lowest SMD is noticed when both inner and outer swirl is imparted. This result proved that the combination of swirs increases the instability of the liquid sheet and as reason finer SMD and higher radial spread of droplets are resulted.

Fig. 12 shows the distribution of liquid mass along the radial direction at 75mm axial distance from the nozzle exit. The liquid percentage of mass distribution is represented on half of the plane. It can be noticed from the Figure 12 that the liquid percentage of mass increases as the swirl number increases and when both the inner and outer air swirl are applied because of the more radial spread of droplets due to increase in swirl number.

Fig 9. Spread of droplet with increase in swirl number (S). With red corresponds to maximum droplet diameter.

Fig 10. Effect of swirl number on the SMD (Swirl imparted to both inner and outer air stream).

The mass flux in the center part of spray becomes lower because of less number of drops having smaller SMD. This corresponds to negligible mass flow at the center of the spray. A significant number of drops with larger droplet size are observed at the sheath of the spray, which increases the mass flow rate at the sheath region, due to which increase in mass flux towards the boundary surface and then decreases again after reaching the maximum value can be noticed. Hence it can be noted that, as the swirl increases the smaller drops are thrown to the periphery of the spray leaving larger drops at the sheath only.
The atomizer performance is very sensitive to the changes made in atomizer geometry. A small change made in the atomizer design would cause considerable changes in the dropsize and size distribution. Fine dropsize are obtained by using the atomizers designed to provide high contact between liquid and air. Hence the atomizer geometry is carefully designed to expose the liquid stream to the high velocity air stream in desired way so as to produce the spray of required characteristics. In pursuit of producing finer sprays, the inner and outer air stream diameters are modified, which enhanced the performance of the atomization and has given better understanding on effect of inner and outer air core diameter variation.

The computational fluid dynamic studies were conducted for configuration (Figure 6c) with outer air core diameter is varied from 36mm to 52mm keeping inner air core diameter constant i.e., 40mm. The airblast atomizer with these configurations were operated at fixed valued of air at outer air stream velocity of 20m/s, inner air stream flow rate of 0.00192 kg/s and water flow rate of 0.0108kg/s and the performance was evaluated. It is noticed from the flow simulation that (see Fig.13), for the fixed value of velocity and flow rate of air and liquid, as the inner air core diameters is decreased from 5mm to 2.5mm, the pressure drop across the atomizer nozzle exit decreases and the magnitude of axial velocity increases. As a result the relative velocity between the air and liquid increase with decrease in inner air diameter, this will create more instability to liquid sheet producing finer droplets. The increase in relative velocity will also create dominance in overcoming surface tension of the droplet and thereby creating finer secondary atomization droplet.

Fig.14 shows the effect of inner air diameter on velocity ratio and momentum ratio. As discussed above decreasing inner air diameter from 5mm to 2.5mm the magnitude of velocity will be increased. There is another parameter on which the performance of the atomization is measured is the velocity ratio (velocity of air to liquid), and momentum ratio. From the Figure 13 it can be notice that as the inner air diameter is decreased the velocity ratio and momentum ratio are increasing. This is evident that the increase in velocity ratio and momentum ratio will enhance the performance of atomization.

5.3.2 Effect of Outer Air Diameter Variation

The computational fluid dynamic studies were conducted for configuration (Figure 6d) with outer air core diameter is varied from 36mm to 52mm keeping inner air core diameter constant i.e., 5mm. The airblast atomizer with these configurations were operated at fixed valued of air at outer air stream velocity of 20m/s, inner air stream flow rate of 0.00192 kg/s and water flow rate of 0.0108kg/s and the performance was evaluated in terms of effect of swirl on both inner and outer air core where inner air core is co-rotating and outer air is counter rotating with fuel.

Fig. 12. Effect of swirl on percentage of mass distribution along radial direction

Fig. 13. Effect of Inner air diameter on SMD. (Inner Air Stream Co-rotating & Outer Air Counter-rotating with fuel, S=0.25)

It is noticed from the flow simulation that, for the fixed value of velocity and flow rate of air and liquid, as the outer air core diameters is increased from 36mm to 52mm, the pressure drop across the atomizer nozzle exit will increases and the magnitude of axial velocity decreases. As a result the relative velocity between the air and liquid decreases with increase in outer air diameter, this will create less instability to liquid sheet and hence producing bigger droplets. The Figure 14
shows the decrease in SMD as the outer air core diameter is increased. It can be noticed from the Figure 14 that as the outer air diameter is increased the poor atomization will results and hence the larger SMD is observed. This is due to the decrease in relative velocity between air and liquid.

Fig 14. Effect of inner air diameter on velocity ration and momentum ratio. (Inner Air Stream Co-rotating & Outer Air Counter-rotating with fuel, S=0.25)

Fig.16 shows the effect of outer air diameter on velocity ratio and momentum ratio. From the Figure 6.18 it can be notice that as the outer air diameter is increased the velocity ratio and momentum ratio are decreasing. This is evident that the decrease in velocity ratio and momentum ratio will affect the performance of atomization by producing poorer SMD. This is mainly due to increase in annular width of airflow at constant flow rate of air results in the loss of relative velocity and momentum ratio of air to liquid resulting in the loss of spray [2].

Fig 15. Effect of outer air diameter on SMD. (Inner Air Stream Co-rotating & Outer Air Counter-rotating with fuel, S=0.25)

5.3 Influence of Flow Parameters on Spray.

The computational fluid dynamic studies were conducted for baseline configuration (Fig.6b) with fixed geometry (i.e., no change in atomizer design inner diameter 5mm and outer diameter 40mm). The airblast atomizer with this configuration was operated at varied flow rate of inner air from 1.92g/s to 2.88g/s (i.e., 0 to 50% increase in inner air flow rate) and outer air velocity is varied from 20m/s to 30m/s (i.e., 0 to 50% increase outer air velocity) and water flow rate varied from 10.8kg/s to 14.04 kg/s (i.e., 0 to 30% increase in liquid flow rate) and the performance was evaluated in terms of effect of swirl on both inner and outer air core where inner air core is co-rotating and outer air is counter rotating with fuel. The minimum of 5 different flow parameter (flow rate of air and liquid) were varied and the computational fluid dynamic simulation was carried out.

Fig 16. Effect of outer air diameter on velocity ratio and momentum ratio. (Inner Air Stream Co-rotating & Outer Air Counter-rotating with fuel, S=0.25)

It is evident from the Fig.17 that as the flow rate of air at inner and outer air streams are increased from 0 to 50% there is significant improvement in the performance of the atomization producing the finer SMD. The main cause for this is that, increasing the mass flow rate of air at inner and outer air stream, increases the relative velocity and air momentum resulting in higher shear and aerodynamic forces on the liquid, leading to better atomization.

Fig 17. Effect of mass flow rate of air and fuel on velocity ratio (Inner Air Stream Co-rotating & Outer Air Counter-rotating with fuel, S=0.25)

Also it can be noticed from the Figure 19 that, as the mass flow rate of liquid increases there is significant increasing SMD. This is because, increasing mass flow rate of liquid in a atomizer would increase the momentum of the liquid jet making it more stable to the disturbances and hence leads to poorer atomization.
Fig 18. Effect of mass flow rate of air and fuel on liquid mass flow ratio (Inner Air Stream Co-rotating & Outer Air Counter-rotating with fuel, S=0.25)

The detailed studies on liquid jet breakup have shown dependence of the liquid jet breakup length and breakup frequency on the momentum ratio [2]. Since the spray characteristics depend much on the jet breakup length, which in turn depends on the momentum ratio, the relation between mean drop diameter (SMD) and momentum ratio is plotted as shown in Fig 18. At lower value of momentum ratio the atomization is poorer and becomes finer at higher values. At very high values of these quantities saturation is reached, beyond which no much further improvement in the atomization could be achieved.

6. CONCLUSIONS

- The numerical investigation is validated using two sets of experimental results.
- The difference between numerical and experimental results is 5% to 8% this is mainly due to numerical investigation is conducted assuming non evaporating droplets. However this level of prediction is in very good agreement with the experimental results.
- The results suggest that the model used for the primary and secondary atomization is valid.

Fig 19. Effect of mass flow rate of air and fuel on momentum ratio (Inner Air Stream Co-rotating & Outer Air Counter-rotating with fuel, S=0.25)

- The variation of predicted SMD with swirl number demonstrated a decrease in its value for each level of increase in the swirl number.
- Smaller droplets start to appear further downstream in addition to significant increase in radial spread of droplets with shorter atomization axial distance for each value of increase in swirl number.
- Employing both inner and outer air swirl in combination with inner air co-rotating and outer air-counter-rotating has proved to have excellent enhancement in atomization by producing finer SMD, when compared to single swirl atomizer.
- The mass flux in central part of the spray becomes lower and increases towards the boundary surface and decreases after reaching the maximum value.
- The variation of predicted SMD with decrease in inner air stream diameter demonstrated a decrease in the value as it is expected. Also it can be noticed that the variation of inner air stream diameter further below from 3mm will not show significant change in the SMD.
- The reduction in the outer air passage area at constant air and flue flow rate will minimize the SMD value; the reason for this is due to increase in the relative velocity.
- The increase in the outer air passage area at constant air and flue flow rate will maximize the SMD value, the reason for this is due to increase in the pressure drop and decrease in the relative velocity for each value of increase in outer air diameter.
- The atomization can be enhanced at higher air passage mass flow rate keeping constant fuel flow rate. i.e. the trend of minimum SMD is found as the air to liquid ratio is increased.
- The atomization can be further improved at lower fuel flow rate keeping constant air passage mass flow rate. i.e. the trend of minimum SMD is found as the air to liquid ratio is increased.
- Significant increase in velocity ratio and momentum ratio is noticed for the higher value of air flow rate.
- The maximum value of momentum ratio and velocity ratio enhances the atomization process.

REFERENCES

Velocities, Journal of Propulsion and Power


