

Numerical predictions of internal flow through pressure swirl atomizer

Saurabh B. Dikshit¹, Digvijay Kulshreshtha² and S. A. Chaniwala³

^{1,2}Mechanical Engineering Department, C. K. Pithawalla College of Engineering and Technology, Surat-395 007, India

³Mechanical Engineering Department, S. V. National Institute of Technology, Surat-395 007, India

saurabh_dikshit@rediffmail.com

Abstract

Pressure swirl atomizers are used in a wide range of applications. The three dimensional model of atomizer is numerically investigated with the primary focus of the analysis on the internal flow characteristics in the pressure swirl atomizer. The variation of the pressure x, y and z-component of velocity along the length of the pressure swirl atomizer has been studied at different injection pressure of 6, 9, 12, 15 and 18 bar using commercial CFD code. The two-fluid Euler/Euler method is considered for the flow inside the atomizer.

Keywords: Numerical simulations, atomization, pressure swirl atomizer, velocity distributions, pressure distributions.

Introduction

Swirling injectors have been widely used in combustion systems such as gas turbine engines, boilers, internal combustion engines and other industrial applications to successfully mix fuel and oxidants with relatively lower injection energy. To ensure a good combustion, it is critical to design atomizers that can produce sprays with a predetermined droplet size distribution at the desired combustor locations. Therefore, prediction of the atomizer performance for design and analysis is critical. Currently, semi-empirical correlations are used to provide guidance in designing pressure-swirl atomizers. Lefebvre (1989) provided a detailed review of the experimental and theoretical studies on the flow phenomena in atomizers and spray formation. One way to study internal flows in atomizers is to use large-scale models in experimental investigations. Studies reported by Yule and Chinn (2000) and Cooper and Yule (2001) used large-scale pressure-swirl atomizers to enable measurements of air-core size and velocity fields inside the swirl chambers and orifices of the atomizers. Such studies are difficult to perform for production-scale atomizers.

As a compliment to the limitations of the experimental techniques, computational fluid dynamics (CFD) provides additional insight into the dynamics of the internal flow of pressure-swirl atomizers. The main difficulty in the numerical simulation of the flow in a pressure-swirl atomizer is the accurate tracking of the liquid/air interface. Jeng *et al.* (1998) and Sakman *et al.* (2000) performed 2D turbulent numerical simulations using moving grids for representing the free surface. Yule and Chinn (2000) used fluent to solve for the liquid flow field. They determined the position of the interface with an approximate method and restricted their simulation to laminar flow. Steinthorsson and Lee (2000) used the volume of fluid (VOF) method in fluent 5 to simulate 3D flow, while a Reynolds stress model was used to model turbulence. Lavante *et al.* (2002) performed numerical simulations of internal flow in both 2D and 3D models using the VOF method and Buelow *et al.* (2003) used a

two-phase VOF technique to simulate the flow through a small-scale pressure-swirl atomizer. Hansen and Madsen (2001, 2002) performed both experimental and computational studies of a large-scale pressure-swirl atomizer.

The atomizer used in this study has been designed for micro gas turbine engine (Kulshreshtha *et al.*, 2009) under joint development at National institute of technology, Surat and C. K. Pithawalla College of Engineering and Technology, Surat. The 3D atomizer is numerically investigated with primary focus of the analysis on the internal flow characteristics in the swirl chamber. The variation of the pressure x, y and z-component of velocity along the length of the pressure swirl atomizer has been studied at different injection pressure of 6, 9, 12, 15 and 18 bar.

Mathematical model

A two-fluid Euler/Euler method is considered for the flow inside the atomizer. The Eulerian multiphase approach treats the liquid and air as separate interpenetrating phases. For each phase mass and momentum equations are solved as well as an equation for volume fraction. In this study, momentum exchange between the gas and liquid is modeled through a symmetrical drag model. The gas-liquid momentum exchange coefficient, C_{gh} is written in terms of mixture density, ρ_{gh} and mixture interfacial area density, A_{gi}

$$C_{gl} = \frac{1}{8} C_D \rho_{gl} A_{gl} |\overline{u_g} - \overline{u_l}|$$

Where

$$\rho_{gl} = \alpha_g \rho_g + \alpha_l \rho_l \quad \text{and} \quad A_{gl} = \frac{6\alpha_g \alpha_l}{d_{gl}}$$

Where, the drag coefficient, C_D , is based on the Schiller-Naumann and Newton particle drag correlations:

$$C_D = \max \left[\frac{24}{Re_{gl}} \left(1 + 0.15 Re_{gl}^{0.687} \right), 0.44 \right]$$

Where Re_{gl} is the relative mixture Reynolds number obtained from:

$$Re_{gl} = \frac{\rho_{gl} |\overline{u}_g - \overline{u}_l| d_{gl}}{\mu_{gl}}$$

Where, $\mu_{gl} = \alpha_g \mu_g + \alpha_l \mu_l$ is the mixture viscosity of gas and liquid. This symmetrical drag model gives correct behavior for particle flow with vanishing volume fractions.

and tangential velocity distributions are studied along the length of the atomizer. The different locations are defined in Fig. 2.

Grid independency

The 3D grid independent study was carried out with number of nodes varying from 24796 to 34820 nodes. The simulation results do not vary comprehensively between nodes of 25821 to 30241 and hence a grid with 26792 nodes was selected for CFD simulations. The grid spacing selected is similar to the Eulerian grid and is

finest at the nozzle exit and becomes gradually coarser away from the nozzle exit. Due to unstructured grid high resolution advection scheme is selected for additional accuracy. The steady state simulations were carried out with the convergence criteria of rms residuals of $1e-05$. The total simulation time for converged results is about 9 CPU hours.

Results and discussion

The flow simulation is carried out for the pressure swirl atomizer to study the variation of pressure, axial, radial and tangential velocity components along the length of the atomizer.

The boundary conditions for the inlet and outlet are taken to be pressure boundary. The different inlet total pressure is 6, 9, 12, 15 and 18 bar for simulation and the relative outlet pressure is taken as zero. The flow that takes place in the atomizer is assumed to be fully turbulent and therefore the simulation is performed using the shear stress transport k-omega model. The k-omega based shear stress transport (SST) model gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy-viscosity. These results in a major improvement in terms of flow separation predictions; important criteria in fluid atomization. The outer casing of the atomizer is considered as wall with no slip boundary. For all cases fluid is taken as kerosene at 27°C.

Fig. 3 shows the variation of x-component of velocity (axial velocity component) along the length of atomizer. It is clear from the figure that the x-component of velocity remains constant until flow reaches to the tangential inlet ports. The locations slot 1, slot 1_1 and slot 1_2 are the points inside the tangential inlet port where velocity of the flow increases because of the decrease in area. When flow enters into the swirl chamber its velocity decreases and then it increases towards the exit of the atomizer. The x-component of the velocity decrease gets converted into z-component

Fig. 1. Dimensional drawing of pressure swirl atomizer.

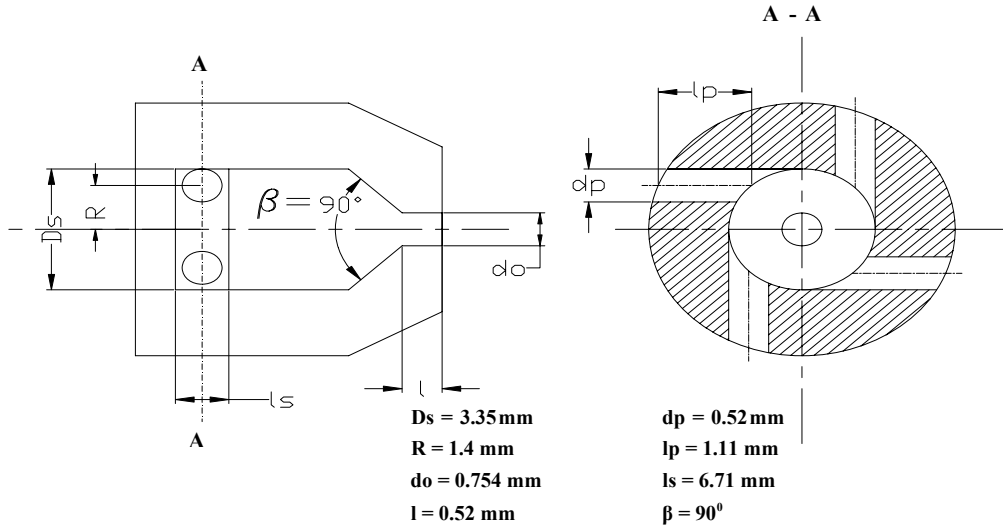
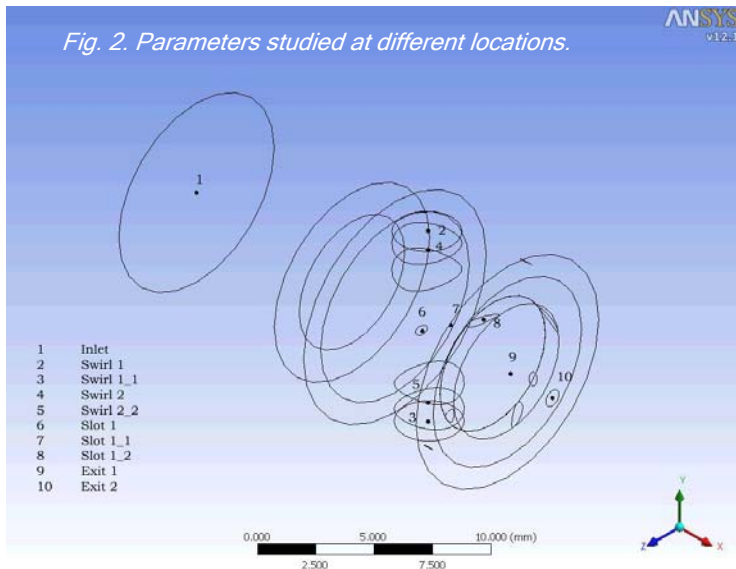


Fig. 1 shows the dimensional drawing of the pressure swirl atomizer used for this study. Fluid enters the atomizer through the common inlet. There are 4 tangential ports connected to the swirl chamber, which lead to the outlet via a cylindrical orifice. The diameter of tangential port is 0.52 mm. The larger diameter of the swirl chamber is 3.35 mm and the length was 6.71 mm. The orifice had a diameter of 0.75 mm. The axial, radial

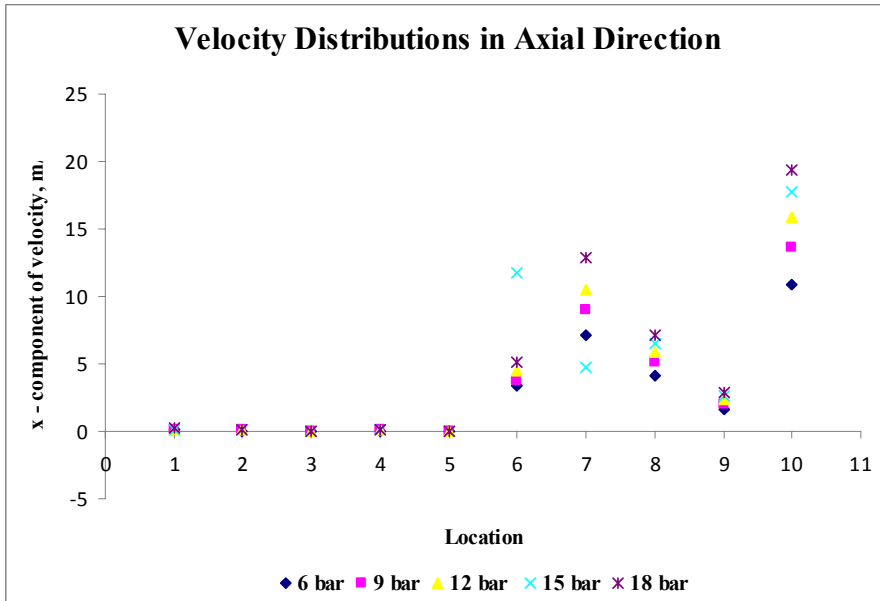
Fig. 2. Parameters studied at different locations.



(tangential velocity component) of velocity inside the swirl chamber and then increases because of the convergent

component is converted into axial velocity component of flow. As flow enters the swirl chamber, tangential velocity component increases because the x-component is converted to the z-component of velocity. The increase in tangential velocity component causes the flow to circulate in the swirl chamber and produce the swirling of the liquid desirable for the process of atomization. As flow passes through convergent section of the atomizer its x-component of velocity increases as other two components of the velocities get converted into axial component.

Fig. 3. Variation of x - component of velocity in pressure swirl atomizer.



section of the atomizer.

Fig. 4 shows the variation of the y-component (radial velocity component) of velocity along the length of atomizer. The figure indicates that the y-component of velocity increases as the flow enters the tangential inlet port because the x and z-components of the velocity gets converted into y-component of the flow and as the flow enters the swirl chamber, the y-component of the velocity once again gets converted into z-component of the velocity.

Fig. 5 shows the variation of z-component of a velocity along the atomizer length. The figure shows that tangential velocity component of flow decreases inside the tangential inlet ports because tangential velocity

Fig. 6 shows the pressure variation inside the atomizer. It is obvious that pressure of the liquid remains constant till flow reaches to the tangential inlet port. Thereafter the pressure decreases and then starts decreasing. The change in direction of the flow causes pressure energy conversion into kinetic energy leading to pressure drop across the atomizer. The higher pressure drop leads to better atomization of the liquid and therefore pressure drop across the atomizer increases as injection pressure increases, leading to better atomization at higher injection pressures.

Conclusion

The numerical simulation has been carried out to understand the flow phenomena inside the pressure swirl atomizer. The main parameters of the study were the variation of x, y and z-component of the velocity and pressure distribution along the length of the pressure swirl atomizer. The three dimensional production scale model has been analyzed numerically using CFD code. It is concluded that the axial velocity component at the exit of the atomizer is more and it increases as the injection pressure increases while the radial and tangential velocities at the exit of the atomizer decrease with increase in injection pressure. The pressure along the length of the atomizer decreases. With the increase in injection pressure, the pressure drop across the atomizer increases, this leads to better atomization. The preliminary results helps in understanding the velocity and pressure distribution of flow through the pressure swirl atomizer. These results do not include the modeling of liquid film cone exiting from the atomizer and hence fails to predict spray breakup and dispersion of spray. Eulerian Multiphase model needs to be incorporated to understand the

Fig. 4. Variation of y - component of velocity in pressure swirl atomizer.

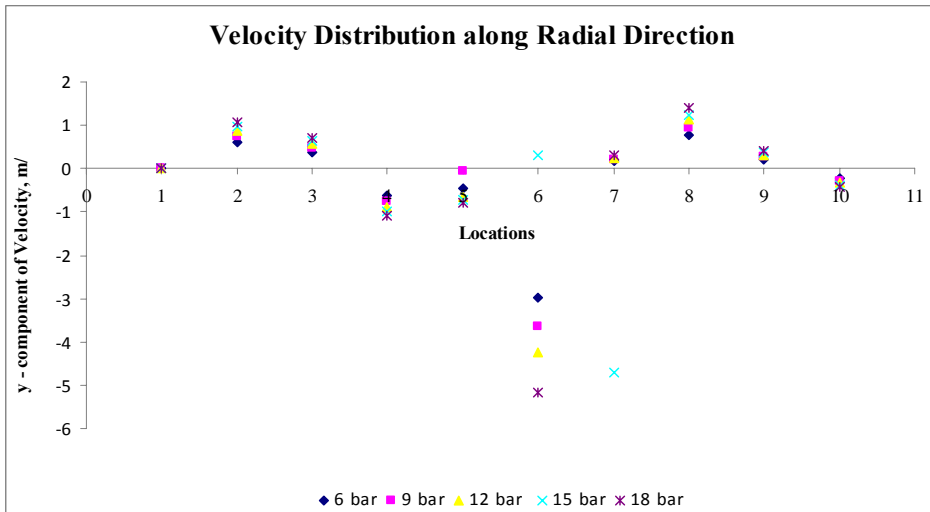


Fig. 5. Variation of z - component of velocity in pressure swirl atomizer.

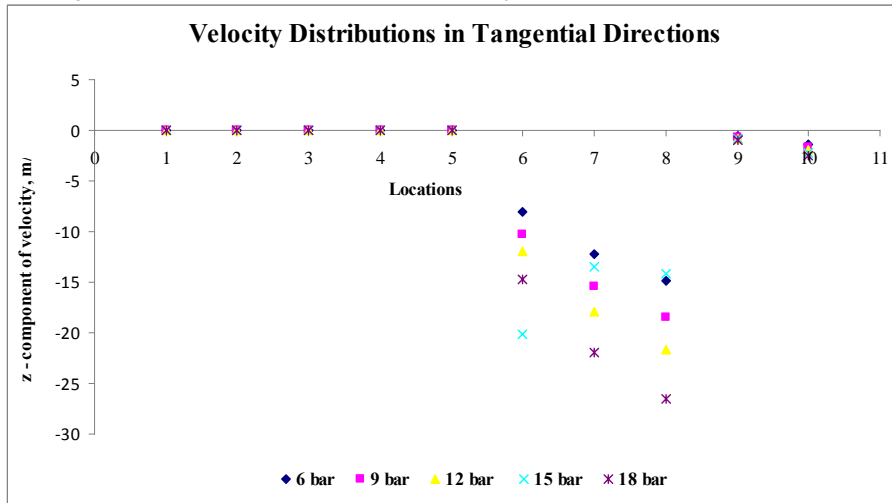
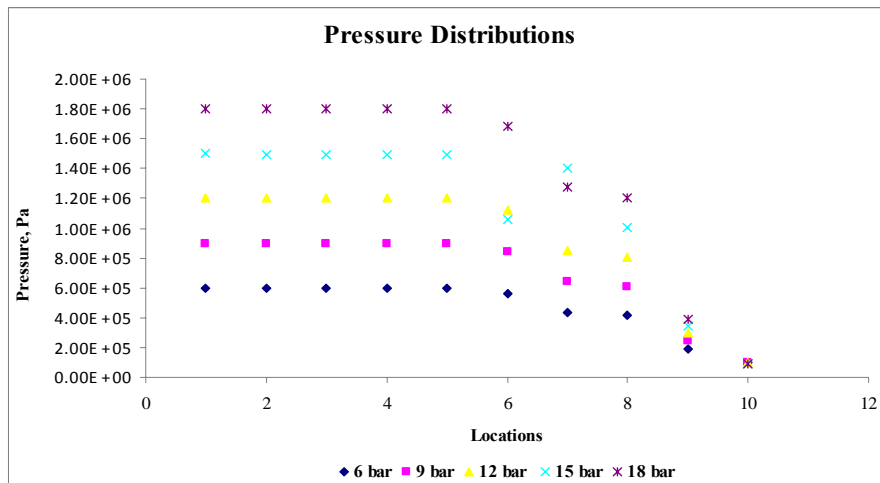


Fig. 6. Pressure variations inside the pressure swirl atomizer.



spray breakup phenomena at the atomizer exit. Following qualitative conclusions related to axial, radial and tangential velocity components can be derived.

1. The x-velocity component of the flow increases as flow enters the tangential inlet ports as area of the port is small, therefore y and z-velocity components are converted into x-velocity component. With increase in injection pressure the axial velocity component tends to increase; reaching maximum at 18 bar injection pressure.
2. As the flow enters the swirl chamber the area suddenly increases and x and y-velocity components decreases. When flow passes through the convergent section of the flow its x-velocity component increases again.
3. The y-velocity component decreases at the entry of the tangential inlet port. Then it recovers and increases as it flows through the inlet port.
4. As the flow enters the swirl chamber the z-velocity component of the flow increases. The x and y-velocity components of the flow are converted into z-component of the velocity therefore only z-velocity component causes the swirling of the flow.
5. When the flow passes towards the exit, the z-velocity component of the flow decreases and gets converted into

the x-velocity component of the flow. That helps in liquid sheet to move in axial direction. Very high velocities of the order of magnitude of around 30 m/s are achieved.

Acknowledgement

The authors are thankful to the staff of C. K. Pithawalla College of Engineering & Technology and S. V. National Institute of Technol., Surat.

References

1. Buelow PEO, Mao CP and Smith S (2003) Two-phase CFD modeling of a simplex atomizer. *Proc. ILASS-Americas*.
2. Cooper D and Yule AJ (2001) Waves on the air core/liquid interface of a pressure swirl atomizer. *Proc. ILASS-Europe*.
3. Cooper D, Yule AJ and Chinn JJ (1999) Experimental measurements and computational predictions of the internal flow field in a pressure swirl atomizer. *Proc. ILASS-Europe '99*.
4. Hansen KG and Madsen J (2001) A computational and experimental study of the internal flow in a scaled pressure-swirl atomizer. M.Sc. thesis, Aalborg University Esbjerg, Denmark.
5. Hansen KG, Madsen J, Trinh CM, Ibsen CH, Solberg T and Hjertager BH (2002) A computational and experimental study of the internal flow in a scaled pressure-swirl atomizer. *Proc. ILASS-Europe 2002*.
6. Jeng SM, Jog MA and Benjamin MA (1998) Computational and experimental study of liquid sheet emanating from simplex fuel nozzle. *AIAA J.* 36, 201-207.
7. Kulshreshtha DB, Channiwala SA and Dikshit SB (2009) Design methodology and development of pressure swirl atomizer for micro gas turbine combustion chamber. *Int. J. Appl. Engg. Res.* 4(10), 1921-1929.
8. Kulshreshtha DB, Dikshit SB and Channiwala SA (2009) Variations of spray cone angle and penetration length of pressure swirl atomizer designed for micro gas turbine engine. *Int. J. Dynamics Fluids.* 5(2), 165-172.
9. Lefebvre AH (1989) Atomization and sprays. Hemisphere publications, New York.
10. Ma Z, Wang D, Jeng SM and Benjamin MA (2000) On the internal flow of pressure-swirl atomizer at two different density ratios. *Proc. ICLASS2000*. p1206.
11. Sakman AT, Jog MA, Jeng SM and Benjamin MA (2000) Parametric study of simplex fuel nozzle internal flow and performance. *AIAA J.* 38, 1214-1218.
12. Steinthorsson E and Lee DM (2000) Numerical simulations of internal flow in a simplex atomizer. *Proc. ICLASS 2000*. p 324.
13. Lavante EV, Maatje U and Albina FO (2002) Investigation of unsteady effects in pressure swirl atomizers. *Proc. ICLASS-Europe*.
14. Yule AJ and Chinn JJ (2000) The internal flow and exit conditions of pressure swirl atomizers. *Atom. Sprays.* 10, 121-146.