

NUMERICAL SIMULATION OF PRESSURE SWIRL ATOMIZER FOR SMALL SCALE GAS TURBINE COMBUSTION CHAMBER

Dikshit S. B.* Kulshreshtha D. B. and Channiwala S. A.

*Author for correspondence

Mechanical Engineering Department,
Dr. S. & S. S. Ghandhy Government Engineering College,
Surat 395001,
India,

E-mail: saurabhdikshit@outlook.in

ABSTRACT

The use of Small Gas Turbine to meet on-site small-scale energy demand offers a great opportunity for preliminary energy saving and reduction of pollutant and greenhouse emissions. The properly designed atomizer can reduce emission and impart flame stability. The numerical simulation of designed pressure swirl atomizer for small scale gas turbine combustion chamber is carried out using ANSYS Fluent. The fuel injection pressure at inlet is kept 6 bar, 9 bar, 12 bar, 15 bar and 18 bar and ambient pressure and temperature is kept constant at atmospheric condition. LISA (Linearized Instability Sheet Atomization) model is used to understand the primary atomization process while the TAB (Taylor Analogy Breakup) model is used to understand secondary breakup of droplets. The numerical results of variation of non-dimensional Sauter mean diameter (SMD) at different injection pressure are compared with theoretical results. It has been found that the non-dimensional SMD decreases with increases in injection pressure and TAB model give satisfactory values compared to theoretical one.

INTRODUCTION

The transformation of bulk of liquid into spray of small droplets is important process for combustion system such as rocket engine, diesel engine, gas turbine engine and industrial furnaces. The combustion process and combustion efficiency depends on the spray characteristics and how droplets are generated. The physical phenomena of liquid sheet breakup helps to improve design of atomizer and spray characteristics. The atomization process of liquid disintegration is affected by liquid property, atmospheric condition and geometric parameter of atomizer [1].

NOMENCLATURE

B	[-]	User specified breakup constant
d	[mm]	diameter
h_0	[mm]	Liquid film thickness
K_y	[-]	Discharge coefficient
\dot{m}	[Kg/s]	Mass flow rate
Oh	[-]	<i>Ohnesorge Number</i>
ΔP	[bar]	Injection pressure
Re	[-]	Reynolds number
R	[mm]	Drop radius
t	[s]	Time

U	[m/s]	Total velocity
u	[m/s]	Axial velocity
V	[m/s]	Particle velocity
w	[m/s]	Tangential velocity
We	[-]	Weber number
x	[-]	Deviation of droplet equator
y	[-]	Dimensionless deformation

Special characters		
ρ	[Kg/m ³]	Density
σ	[N/m]	Surface tension
μ	[Ns/m ²]	Dynamic viscosity
ω	[rad/s]	Angular velocity
θ		Spray angle

Subscripts	
a	Air
c	Critical
ch	child
g	Gas
N	normal
o	Orifice
p	Particle
pr	parent
sl	Slip
0	Initial value

The atomization phenomena is very complex. CFD analysis becomes very useful tool to solve complex problem. There are number of models developed to solve atomization process. Lefebvre [1] provides a detailed review of the experimental and theoretical studies on the flow phenomena in atomizers and spray formation. Brickman et.al. [2] studied the different physical phenomena of atomization process. Recent CFD models of spray flows with and without combustors introduced for gas turbine and diesel engine analyses indicate the importance of atomization assumptions for spray flow dynamics. Such models are highly sensitive to spray origin parameters such as initial droplet size distribution, initial velocity and droplet number density [3].

In the present work the pressure swirl atomizer, designed earlier for small scale gas turbine combustion chamber, is analysed using CFD tool – ANSYS Fluent [3]. The variation of Sauter Mean Diameter (SMD) and normal droplet distribution are plotted using Taylor Analogy Breakup (TAB) model. The numerical results are compared with theoretical correlation.

NUMERICAL SIMULATION

The fluid flow through atomizer and after injection of fluid are totally different. The experimental analysis of atomization characteristics are time consuming and expensive while Computational fluid dynamics (CFD) analysis is the present state-of-art technique for fluid flow analysis. The continuous and dispersed (particle) flow can easily modelled and analysed with CFD. For atomization process, there is an interaction between two phase, gas/liquid, occurs. The gas is considered as continuous phase and liquid is as discrete phase. For the present study Euler – Lagrange approach with Lagrange Discrete phase model is used. The gas phase is treated as a continuum by solving the Navier–Stokes equations, while the dispersed phase is solved by tracking a large number of particles or droplets through the calculated flow field.

The atomization process can be divided in to two stages: Primary breakup and Secondary breakup. The liquid sheet get disintegrated into first ligament and then in to droplets by aerodynamic action between air and liquid sheet, is called primary breakup. Once the liquid droplets are formed, the spray evolution is determined by drag, collision, coalescence, and secondary breakup.

Primary Breakup Model

The primary breakup model provides starting conditions for the droplets. It can provide initial particle diameter, initial particle velocity components, and initial spray angle of the liquid discharging from atomizer. The primary breakup model in current study is Linearized Instability Sheet Atomization (LISA) Model. The LISA model is able to simulate primary atomization process in pressure swirl atomizer.

Figure 1 shows the primary atomization process. The swirling liquid comes out of exit orifice of diameter d_0 and makes a liquid film of thickness h_0 at an angle θ . Liquid sheet get disintegrated in to ligaments and further in to droplets. The process can be expressed by equation 1.

$$\dot{m} = \pi \rho_p u h_0 (d_0 - h_0) \quad (1)$$

It is assumed that the total velocity, U , is related to the injector pressure by the following relation [4]:

$$U = k_v \sqrt{\frac{2\Delta p}{\rho_p}} \quad (2)$$

and

$$k_v = \max \left[0.7, \frac{4\dot{m}}{\pi d_0^2 \rho_p \cos \theta} \sqrt{\frac{\rho_p}{2\Delta p}} \right] \quad (3)$$

The computed film thickness, h_0 , will be equal to half the injector nozzle diameter if the discharge coefficient, k_v , is larger than 0.7 [3].

Assuming that ΔP (injection pressure difference) is known, the total injection velocity can be computed from equation of U . The axial film velocity component, u , is then derived from:

$$u = U \cos \theta \quad (4)$$

Where θ is the spray angle, which is assumed to be known. At this point, the thickness, h_0 , and axial velocity component of the liquid film are known at the injector exit. The tangential component of velocity ($w = U \sin \theta$) is assumed to be equal to the radial velocity component of the liquid sheet downstream of the nozzle exit. The axial component of velocity is assumed to remain constant.

After the liquid film has left the injector nozzle, it is subject to aerodynamic instabilities that cause it to break up into ligaments. The theoretical development of the model is given in detail by Senecal et al. [5]. The model assumes that a two-dimensional, viscous, incompressible liquid sheet of thickness $2h$ moves with velocity U through a quiescent, inviscid, incompressible gas medium. A spectrum of infinitesimal disturbances is imposed on the initially steady motion, in terms of wave amplitude. The most probable droplet diameter that is formed from the ligaments is determined from:

$$d_p = 1.88 d_l (1 + 3Oh)^{1/6} \quad (5)$$

Where d_l is diameter of ligament, which is formed from liquid sheet and depends on type of wave – short or long. Using the droplet diameter size factor 1.88 and Oh is the particle Ohnesorge number that is defined as:

$$Oh = \frac{\sqrt{We_p}}{Re_p} \quad (6)$$

Where We_p is the Weber Number based on half the film thickness and the gas density. Re_p is the Reynolds Number based on the slip velocity.

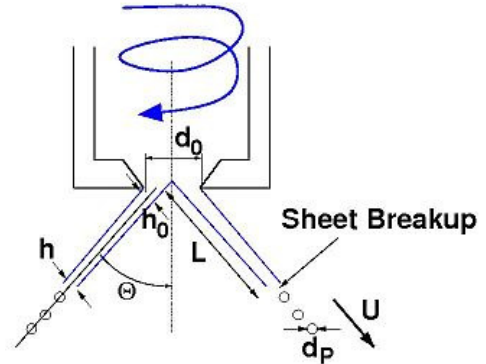


Figure 1. Primary Atomization Process [3]

Secondary Breakup Model

The secondary breakup of the liquid sheet is caused by turbulence within the liquid phase, implosion of cavitation bubbles and external aerodynamic forces acting on the liquid sheet. Depending on the injection parameters such as the relative velocity between liquid and gas, the liquid and gas densities and the liquid viscosity and surface tension the contribution of each of the above mechanisms to the spray breakup may vary.

Breakup regime typically may classified by Weber number of liquid phase.

$$We = \rho V^2 D / \sigma \quad (7)$$

If a droplet is exposed to a gas flow, significant deformation starts at a Weber number of unity. Above a certain value of the Weber number, the droplet deformation leads to breakup. Typically, the following breakup regimes are observed and shown in figure 2 [1]:

Vibrational breakup: $We < 12$

Bag breakup: $12 < We < 50$

Bag-and-stamen breakup: $50 < We < 100$

Sheet stripping: $100 < We < 350$

Catastrophic breakup: $350 < We$

For the numerical simulation of droplet breakup, a statistical breakup approach is used. It is assumed that if a droplet breaks up into child droplets, the particle diameter is decreased accordingly to the predictions of the used breakup model. Taylor Analogy Breakup (TAB) model is used for present study.

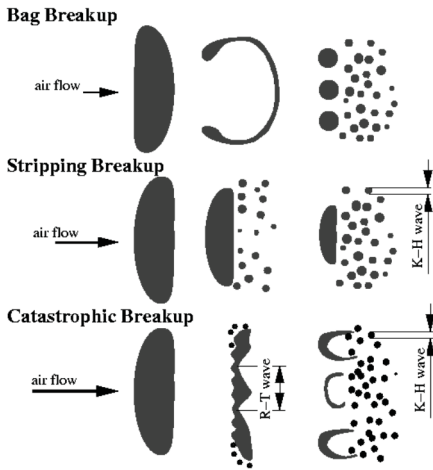


Figure 2. Types of Secondary Breakup [1]

Taylor Analogy Breakup (TAB) Model

O'Rourke and Amsden [7] proposed the TAB model that is based on the Taylor analogy which consider the droplet distortion can be described as a one-dimensional, forced, damped, harmonic oscillation similar to a spring-mass system. In the TAB model, the droplet deformation is expressed by the dimensionless deformation $y = 2(x/r)$, where x describes the deviation of the droplet equator from its undeformed position. Assuming that the droplet viscosity acts as a damping force and the surface tension as a restoring force, it is possible to write the equation of deformation motion as:

$$\frac{d^2 y}{dx^2} = \frac{5\mu_p}{\rho_p r^2} \frac{dy}{dx} + \frac{8\sigma}{\rho_p r^3} y = \frac{2\rho_g V_{slip}^2}{3\rho_p r^2} \quad (8)$$

Integration of this equation leads to the following time-dependent particle distortion equation:

$$y(t) = We_c + e^{-t/t_d} \left[(y_0 - We_c) \cos \omega t + \left(\frac{y_0}{\omega} + \frac{y_0 - We_c}{\omega t_d} \sin \omega t \right) \right] \quad (9)$$

Where following parameters are used:

$$\text{Droplet breakup time } t_d = \frac{2\rho_p r^2}{C_d \mu_p} \quad (10)$$

$$\text{Droplet oscillation frequency } \omega^2 = \frac{C_k \sigma}{\rho_p r^3} - \frac{1}{t_d^2} \quad (11)$$

$$\text{Critical Weber Number } We_c = We \frac{C_f}{C_k C_b} \quad (12)$$

C_b , C_d , C_f , and C_k are constants and their values are chosen to match experiments and theory.

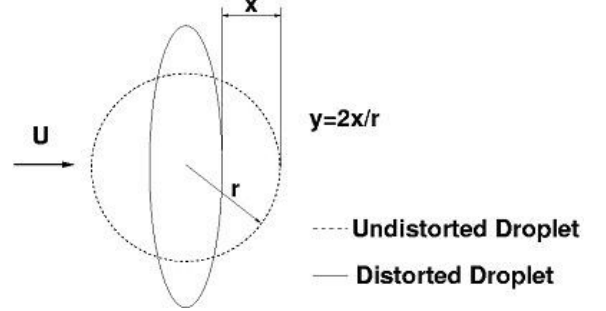


Figure 3 Particle Distortion for the TAB model [3]

Breakup only occurs if the particle distortion y exceeds unity, which means that the deviation of the particle equator from its equilibrium position has become larger than half the droplet radius. The Sauter mean radius of the child droplets after breakup is calculated from the following expression:

$$\frac{r_{p,parent}}{r_{p,child}} = \left[1 + 0.4K + \frac{\rho_p r_{p,parent}^3}{\sigma} dy_0^2 \left(\frac{6K-5}{120} \right) \right] \quad (13)$$

Which is based on the conservation of surface energy and energy bound in the distortion and oscillation of the parent droplet and surface energy and kinetic energy of the child droplets.

The TAB model has been used to determine the normal velocity of the child droplets after breakup. At the time of breakup, the equator of the parent droplet moves at a velocity of V_N in a direction normal to the parent droplet path. This velocity is taken as the normal velocity component of the child droplets and the spray angle θ can be determined from:

$$\tan \frac{\theta}{2} = \frac{V_N}{V_{slip}} \quad (14)$$

After breakup of the parent droplet, the deformation parameters of the child droplet are set to $y(0) = \dot{y}(0) = 0$.

NUMERICAL APPROACH

Figure 4 shows the dimensional drawing of the pressure swirl atomizer under study. In the present work commercially available CFD tool ANSYS – Fluent is used. The Primary breakup is carried out using LISA model which is best suited for pressure swirl atomizer. The secondary breakup is analysed by TAB secondary breakup model. The spray characteristics like spray penetration length and Sauter Mean Diameter (SMD) are analyze. The geometrical fluid model is developed based on

designed dimensions. Figure 5 shows the geometry under analysis and showing boundaries.

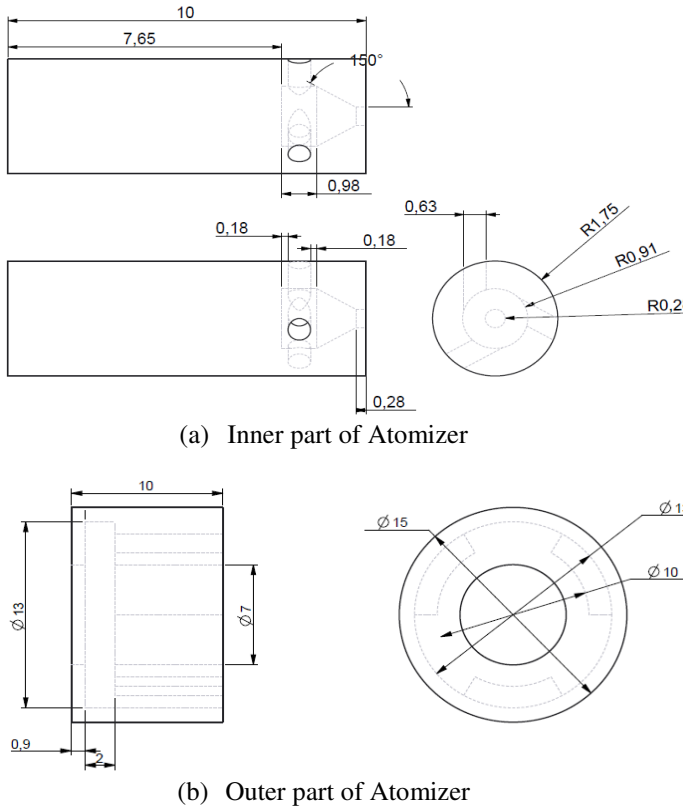


Figure 4 Dimensional Drawing of Pressure Swirl Atomizer (All dimensions are in mm)

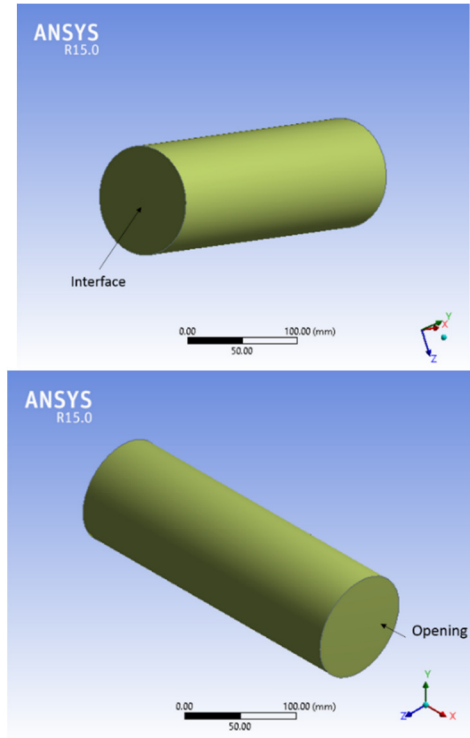
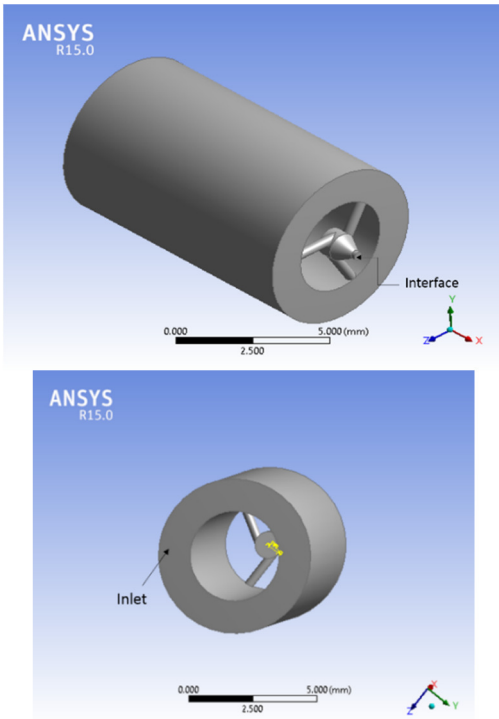


Figure 5 Nozzle and spray chamber geometry

The grid independent study has been carried out for the present geometry of the pressure swirl atomizer. The unstructured mesh with tetrahedral elements are used. The meshing has been started with coarse mesh having 167515 number of tetrahedral elements. The Sauter Mean Diameter is measured for 6 bar injection pressure. The number of tetrahedral elements are increased by changing minimum size of the elements in mesh size setting. It has been found that after 232318 elements, decreasing mesh size do not change SMD and it is shown in figure 6.

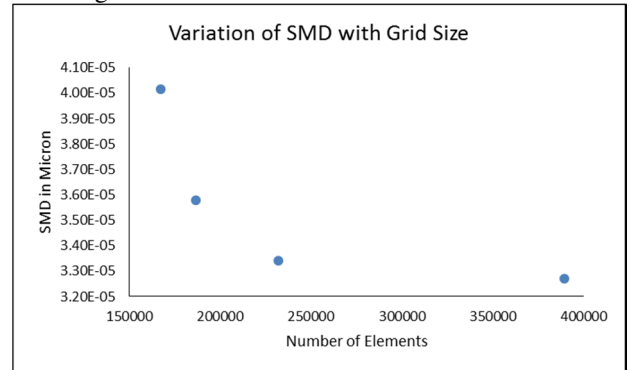


Figure 6 Grid Independence Study

Figure 7 shows the meshing of the fluid domain under analysis. The 3D analysis the object is carried out. In general setting of fluent Pressure Based, Steady State Solver is selected. In Model option, energy equation is kept on. The realizable k-ε turbulence model with standard wall function is selected to

define turbulence for the problem. The species transport and dispersed phase model is kept on.

Ethyl alcohol - air mixture is selected for vapor phase material and ethyl alcohol – liquid is selected as droplet material. In dispersed phase unsteady particle tracking is selected with particle time step size of 0.0001 s. The injection parameters are set for pressure swirl atomizer which uses LISA model for primary breakup. For secondary breakup TAB model is selected with dynamic drag law. The injection pressure is set to 6 bar, 9 bar, 12 bar, 15 bar and 18 bar.

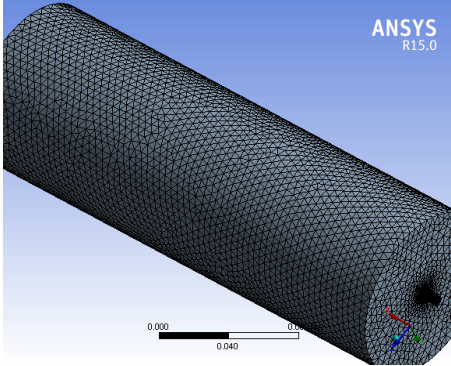


Figure 7 Generated Mesh

THEORETICAL APPROACH

In pressure swirl atomizer, the drop size relations are determined by empirical methods because of complexity of various physical phenomena are involved in atomization process from pressure swirl atomizer. The correlations for mean drop size of the form

$$SMD = \sigma^a v^b \dot{m}_L^c \Delta P_L^d \tag{15}$$

From the analysis of the flow process in the exit orifice of pressure swirl atomizer Lefebvre [9], derived following equation of SMD;

$$SMD = A \sigma^{0.25} \mu_L^{0.25} \rho_L^{0.125} d_0^{0.5} \rho_A^{-0.25} \Delta P_L^{-0.375} \tag{16}$$

The exponent value of surface tension and viscosity agree fairly with all reported value [10], [11], and [12]. No reliable experimental data exist to test the exponent for liquid density but all the evidence suggests that the effect of liquid density on SMD is quite small [1]. The value of exponents of injection pressure differential and air density are identical to the experimental value obtained by Abou-Ellail et.al [13]. The exponent of d_0 confirms the theoretical value. Substituting $d_0 \propto \dot{m}_L^{0.5} / (\Delta P_L / \rho_L)^{0.25}$ in the equation 16 and determining value of A from data given by Jasuja [14], equation of SMD is

$$SMD = 2.25 \left(\frac{\sigma \mu_L \dot{m}_L}{\rho_A \Delta P_L^2} \right)^{0.25} \tag{17}$$

RESULTS

From the above theoretical correlation, SMD is calculated by fixing the mass flow rate of 1.7259E-4 kg/s and pressure

differential of 6, 9, 12, 15 and 18 bar. The exit orifice diameter is 0.56 mm. Liquid ethanol is selected as fluid flowing through an atomizer and air is as continuous medium. The numerical simulation has also been carried out for the same atomizer dimensions and operating parameters using ANSYS – Fluent. LISA model is selected as primary breakup model and TAB model is selected as secondary breakup model.

From the theoretical and numerical results it has been found that SMD of the spray decreases with increase in injection pressure, which is an implication of the increase in atomization quality. The increase in the liquid pressure differential causes the liquid to be discharged from the nozzle at a high velocity, which promotes a finer spray. At high flow velocities, the droplet diameter becomes smaller due to better atomization. Figure 7 shows the variation of SMD with injection pressure. The numerical and theoretical results are matching with each other and it is clearly indicated in figure 7.

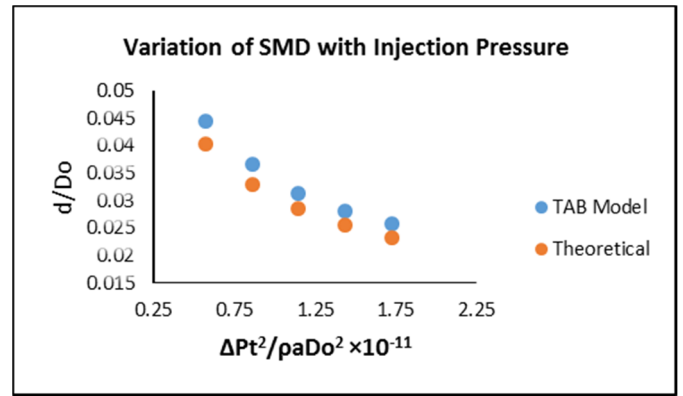


Figure 8 Variation of SMD with Injection Pressure

SMD does not present any information about the distribution of the droplet size. In other words, two sprays with the same SMD may have different droplet distribution. The droplet distribution can be represented in terms of normal distribution function $f(d)$ that gives the number of particle of given diameter d . The normal distribution curve is plotted for numerical results in figure 9. It is clear from the figure that number of smaller drops increases with increase in injection pressure.

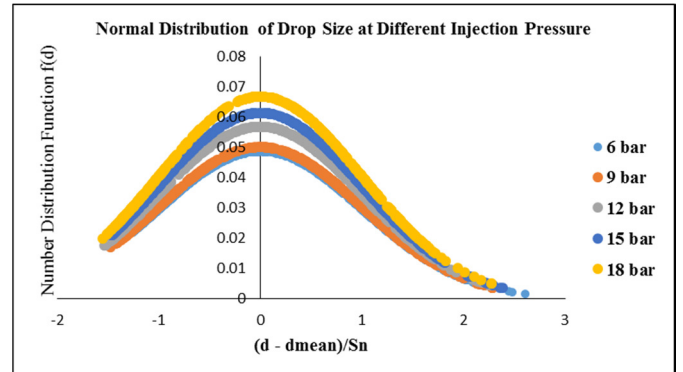


Figure 9 Number Distribution of Drop Size at Different Injection Pressure – A Normal Curve

Conclusions

The CFD analysis of small scale pressure swirl atomizer has been carried out using ANSYS – Fluent. The ethyl alcohol is used as fuel to atomize. The fuel is injected at 6, 9, 12, 15 and 18 bar in spray chamber at atmospheric condition. LISA model is used as primary breakup model and TAB model is used as secondary breakup model. The results are compared with theoretical analysis.

It is observed that as the injection pressure increases the SMD decreases. The same trend has been found for both numerical and theoretical results. The results of theoretical and numerical models are comparable and in good agreement with each other. It is clear from the results also indicate that as the injection pressure increases the number of smaller droplet increases.

REFERENCES

- [1.] Lefebvre A. H. Atomization and Sprays. *Hemisphere Publishing Corporation*, New York, 1989.
- [2.] Brinckman, Hosangadi, Ahuja, Dash and Feldman, A CFD Methodology for Liquid Jet Breakup and Vaporization Predictions in Compressible Flows. *AIAA Paper* No. 2008-1023.
- [3.] Theory Guide ANSYS 2015.
- [4.] Han Z., Perrish S., Farrell P. V. and Reitz R. D., Modeling Atomization Processes of Pressure-Swirl Hollow-Cone Fuel Sprays, *Atomization and Sprays*, Vol. 7, Nov.-Dec. 1997, pp. 663-684.
- [5.] Senecal P. K., Schmidt D. P., Nouar I., Rutland C. J., Reitz R. D. and Corradin M. L., Modeling High-Speed Viscous Liquid Sheet Atomization, *International Journal of Multiphase Flow*, 25, 1999, pp. 1073-1097.
- [6.] Pilch M. and Erdman C.A., Use of Breakup Time Data and Velocity History Data to Predict the Maximum Size of Stable Fragments for Acceleration-Induced Breakup of a Liquid Drop, *International Journal of Multiphase Flow*, Vol. 13, No. 6, 1987, pp. 741-757.
- [7.] O'Rourke P. J. and Amsden A. A., The TAB method for numerical calculation of spray droplet breakup, *SAE paper* 872089, 1987.
- [8.] Kolmogorov, A.N., On the log-normal distribution of particles sizes during breakup process. *Dokl. Akad. Nauk. SSSR XXXI* (2), 1941, 99–101.
- [9.] Lefebvre A. H., Gas Turbine Combustion, *Hemisphere, Washington DC*. 1983.
- [10.] Simmons H. C. and Harding C. F., Some Effects on Using Water as A Test Fluid in Fuel Nozzle Spray Analysis, *ASME Paper* 80-GT-90, 1980.
- [11.] Jones A. R., Design Optimization of A Large Pressure Jet Atomizer for Power Plant, *Proceedings of the 2nd International Conference of Liquid Atomization and Sprays*, Madison Wis, 1982, pp 181-185.
- [12.] Wang X. F. and Lefebvre A. H., Mean Drop Size from Pressure Swirl Nozzles, *AIAA J. Propulsion and Power* Vol. 3 No.1 1987 pp.11-18.
- [13.] Abou-Ellail, M.M.M., Elkotb M. M. and Rafat N. M., Effect of Fuel Pressure Air Pressure and Air Temperature on Droplet Size Distribution in Hollow Cone Kerosene Sprays, *Proceedings of the 1st International Conference on Liquid Atomization and Spray System* Tokyo, 1978, pp 85-92.
- [14.] Jasuja A. K., Atomization of Crude and Residual Fuel Oils, *ASME Journal of Engine Power*, Vol 101, No. 2, 1979 pp 250-258.

Justifications/Corrections

Thank you sir for your comments. I tried to justify and correct all the comments suggested by both reviewer.

Review 1

- **Comment:** There are numbers of grammatical mistakes in the draft paper.
 - **Justification:** All corrections are made.
- **Comment:** Each Figure should be explained clearly in the text, such as “Figure 1 shows ...”.
 - **Justification:** All figures are explained.
- **Comment:** Labels in Figures 7 and 8 should be better presented with explanations in the text.
 - **Justification:** Figure 7 and 8 are explained in detail
- **Comment:** The sentence “As the TAB over predict the SMD value ...” should be “... under predict ...”.
 - **Justification:** Correction is made.
- **Comment:** The difference between the TAB model and theoretical correlations seems to be very large.
 - **Justification:** The theoretical analysis is revised and the new results are plotted and it shows the results are matching with that of numerical analysis.
- **Comment:** In Ref [5], “1991” should be changed to “1999”.
 - **Justification:** Correction is made in reference list.

Review 2

- **Comment:** There are some grammatical mistakes should be avoided.
 - **Justification:** All corrections are made.
- **Comment:** What is the reference of equation (5) and equation (6) or how they have been developed?
 - **Justification:** Equation 5 and 6 are adopted form reference [5] Senecal P. K. et.al.
- **Comment:** Undefined characters have been used in the equations.
 - **Justification:** Nomenclatures are added for undefined characters
- **Comment:** The study of grid independence is missed.
 - **Justification:** The grid independent study is included and shown in figure 6.
- **Comment:** The steady state solution - unsteady particle tracking was considered.
 - **Justification:** If whole field is considered as unsteady, it requires more time to complete the solution and hence only particle flow is taken as unsteady and remaining domain is under steady state condition.
- **Comment:** SMD results for TAB model was approximately more than twice of that from the theoretical results; however the authors stated that TAB model gave satisfactory model.
 - **Justification:** The theoretical analysis is revised and the new results are plotted and it shows the results are matching with that of numerical analysis.
- **Comment:** The atmospheric temperature assumption.
 - **Justification:** Atomizers are generally tested at cold conditions and hence spray characteristics are derived at atmospheric temperature.