IJE TRANSACTIONS B: Applications Vol. 26, No. 5, (May 2013) 553-562

International Journal of Engineering

Journal Homepage: www.ije.ir

The Effects of Pressure Difference in Nozzle's two Phase Flow on the Quality of Exhaust Mixture

M. Abbasalizadeh[a]*, S. Jafarmadar[a], H. Shirvani[b]

[a] *Department of Mechanical Engineering, Urmia University, Urmia, Iran*
[b] *Department of Computing Science, Anglia Ruskin University, Chelmsford, CM1 1SQ, UK*

PAPER INFO

Paper history:
Received 07 October 2012
Received in revised form 12 December 2012
Accepted 24 January 2013

Keywords:
Compressible Flow
Multi-phase Flows
RANS: Reynolds Averaged Navier-Stokes
Supersonic
Turbulence Models
Two-phase Flows Premixed
Un-premixed Nozzle Flow

ABSTRACT

In most applications of nozzles with gas-liquid two-phase flow, the quality of mixture inthe exhaust of nozzle as well as the flow velocity are the most important parameters. On the other hand, in some industrial applications, such as industrial painting or water injection in forced induction (turbocharged or supercharged) internal combustion engines, the quality of the spray is the main goal of design. In this case, and for improving the injection performance the air-water two phase nozzle injection flow is more remarkable subject. There are two options for this purpose, premixed or un-premixed air-water in entrance of the nozzle. In both cases, the nozzle not only has to accelerate gas and liquid to extra high velocity, but also it is supposed to have a high quality mixture in exhaust of nozzle. In this study, the turbulent gas-liquid two-phase premixed/un-premixed flow through the nozzle is simulated by the Eulerian-Lagrangian approach. The gas phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the liquid as a dispersed phase is solved by tracking a large number of droplets through the calculated flow field. The pressure, velocity and Mach number profiles as well as air flow-rates and particle residence time inside the nozzle for various back pressure values have been computed. This work has been validated by comparing pressure profiles and air flow-rates between simulated results and available experimental results for un-premixed nozzle flow

doi: *10.5829/idosi.ije.2013.26.05b.12*

1. INTRODUCTION

The converging-diverging nozzle as the most important and basic piece of engineering hardware associated with the high speed flow of gases, is used to increase speed of media, control the flow direction, atomization of fluid flow and even for cleaning the surfaces. Anderson and Terpstra [1] employed the developed high-pressure gas atomization nozzles with discrete jets resembling convergent–divergent nozzle. Lear and Sherif [2] presented an optimization analysis of a two-phase flow mixture of a gaseous phase and an incompressible condensed phase through a converging-diverging nozzle. The analysis was based on maximizing the condensed phase momentum flux for a set of mixture parameters that included the liquid mass injection ratio, liquid and gas properties, nozzle size, and nozzle stagnation-to-back pressure ratios. A good reference for un-premixed air-water two-phase flow through the nozzle which critical flow is experimentally and analytically investigated was presented by Lemonnier and Selmer-Olsen [3]. In this work, two different designs of axisymmetric converging-diverging nozzle investigated by injecting the liquid centrally or at the vicinity of the wall. Lemonnier and Selmer-Olsen observed that when the liquid was injected centrally, a liquid jet was formed which immediately breaks up and generates small droplets entrained in the gas stream. On the contrary, when the liquid was injected as a film close to the wall, the entrainment process was totally different. In the second inlet condition, Lemonnier and Selmer-Olsen found that the film entered the throat of the nozzle and the mixing of the two phases took place farther downstream. The acceleration of the liquid was delayed and this gave a higher level of mechanical non-

* Corresponding Author Email: mabbasalizadeh@yahoo.com (M. Abbasalizadeh)

equilibrium. According to the experimental data, critical flow phenomena related to the liquid fraction entrained at the inlet. Moreover, Lemonnier and Selmer-Olsen found that by progressively decreasing the outlet pressure that low gas quality flow might remain sub-critical in nature even if the upstream/downstream pressure ratio was as high as 6:1.

Air-water as a two-phase flow passes through converging, throat, and diverging part of nozzle; which start by subsonic flow in converging section and exit from the nozzle as a supersonic, sonic, or subsonic flow. Diverging part of the nozzle with supersonic flow usually includes the shock wave that produces a near-instantaneous deceleration of the flow to subsonic speed. Air-water two-phase flow through the nozzle also has other intricacy behaviour which should be considered in simulation, such as two or three-way turbulence coupling and droplet collision and break up [4]. Therefore, simulation of air- water two-phase flow through the nozzle needs to utilize appropriate turbulence model as well as proper multi-phase model.

In this study, finite volume method as a Computational Fluid Dynamics (CFD) method uses to solving two-phase turbulent flow equations. Since the two-phase flow inside the nozzle has high Reynolds number, using Reynolds Averaged turbulence models are the case throughout this research. Common "industrial standard" Reynolds average model is k-ε model, which employs two equations for turbulence kinetic energy and its dissipation rate. Standard, RNG (Re-Normalisation Group), and Realizable k-ε are available k-ε models in FLUENT package. Here, Realizable k-ε model [5] is utilized for modelling the turbulence effects of two-phase flow.

The other important part of numerically investigation of multi-phase flows through the nozzle is exertion of the appropriate model to prediction and computation of different behaviour of phases in various initial and boundary conditions. For this purpose, the first step is to determine of the best regime inside the nozzle which provides some broad guidelines for selecting the appropriate models. Generally, there are two approaches for modelling multi-phase flows, namely: Eulerian-Eulerian approach and Eulerian-Lagrangian approach. In the Eulerian-Eulerian approach, the different phases are considered mathematically as interpenetrating continua. Volume fractions for each phase are considered, and in this approach, conservation equations for each phase are applied. In the last two decades, the Eulerian–Eulerian method has been implemented for simulating some two-phase flows [6-12]. Three of the most popular and widely used models in Eulerian-Eulerian approach are the Volume of Fluid (VOF) model, the Mixture model and the Eulerian model. However, for numerical computation of two-phase flow through the nozzle, Mixture and Eulerian models are applicable. In the Mixture model, coupling between the phases should be strong so that it can model the n phases by solving the momentum equation for the mixture; it prescribes relative velocities to described phases by using the concept of slip velocities. However, Eulerian model solves a set of n momentum and continuity equations for each phase; hence, it can be more complex than Mixture model. The number of secondary phases in Eulerian model is limited only by memory requirements and convergence behaviour. The Eulerian model is a better choice compared to Mixture model whenever the accuracy is more important than computational effort. Otherwise, Mixture model, since it uses a smaller number of equations compared to the Eulerian model, is a good option to solve a simpler problem. Therefore, the Eulerian model is suitable for simulation of pneumatic transport of sand particles and water droplets through the nozzle. On the other hand, the Eulerian models are more appropriate for flows in which the volume fraction(s) of the secondary phase(s) exceed 10%. Flows in which the dispersed-phase volume fractions are less than or equal to 10% can be modelled using the Discrete Phase model.

In the Eulerian-Lagrangian approach, however, just the fluid phase is treated as a continuum. Time-averaged Navier-Stokes equations are solved for continuous phase, while the dispersed phase is solved by tracking a large number of particles and droplets through the nozzle. The Eulerian-Lagrangian approach or Discrete-Phase Model is utilized in dilute secondary phase(s). Set of Eulerian, Lagrangian and hybrid particle models implemented to simulation of coarse particle conveying through the curved and straight rectangular ducts [13]. Discrete-Phase Model was also utilized in simulation of two-phase flow through the nozzle and jet flow [14-18].

This research employs commercial CFD (Computational Fluid Dynamics) code, FLUENT, for supersonic turbulent multi-phase flow through the converging-diverging nozzle. Using FLUENT allows selection of the Reynolds averaged models such as RSM, two-equation k-ε models, as well as Eulerian model and Discrete Phase model [19]. However, as water volume fraction in available experimented data [3] is less than 10%, so this study puts emphasis on Discrete Phase Model rather than Eulerian Model.

2. NUMERICAL METHODOLOGY

2. 1. Geometry Two-phase flow through the converging-diverging nozzle by experimental application of the separated flows of air and water has been investigated by Lemonnier and Selmer-Olsen [3]. Figure 1 shows the schematic sketch and dimensions of mentioned nozzle with 5 mm throat diameter.

2. 2. Boundary Conditions and Experimental Results In this study the liquid injection flow rate is

93 kg/h, and the upstream pressure is supposed to be 6 bar. For constant inlet conditions, a gradual decrease in back pressure down to atmospheric has been exerted for exit of the Nozzle. The values for back pressure along with the experimentally measured and analytically calculated air mass flow rates are given in Table 1. More information of test equipments, experimental procedure, and physical modelling is given by Lemonnier and Selmer-Olsen [3].

Figure 2 shows pressure profiles measured for the nozzle sketched in Figure 1. The experimental data points in Figure 2 resemble the well-known topological behaviour of single-phase compressible nozzle flow under similar conditions.

In the following sections, the grid dependency test, numerical modelling and simulation of premixed and un-premixed two-phase flow is presented, and for validation of multi-phase simulation through the nozzle, the simulation results are compared with experimental data.

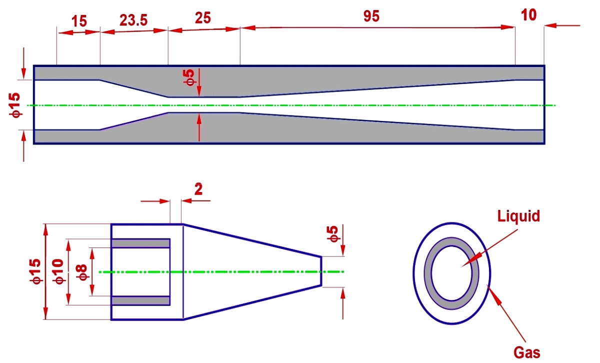

Figure 1. Converging-diverging nozzle with un-premixed air-water two-phase flow (All dimensions are in mm) [3]

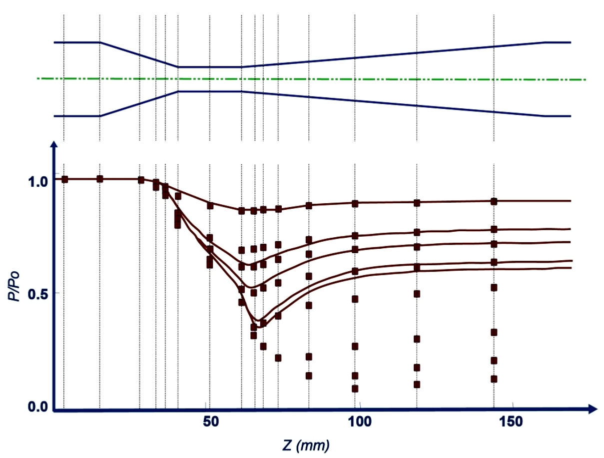

Figure 2. Pressure distribution measured vs. calculated through the Nozzle [3]

TABLE 1. The back pressure and air flow rate of nozzle [3]

P_{out} (bar)	Air Flow Rate (kg/h)	
	Experimented	Calculated
5.38	37.2	32.9
4.64	54.1	49.6
4.30	58.4	52.0
3.80	61.1	52.7
3.10	62.0	52.7
2.00	62.1	52.7
1.28	62.4	52.7
0.81	62.2	52.7

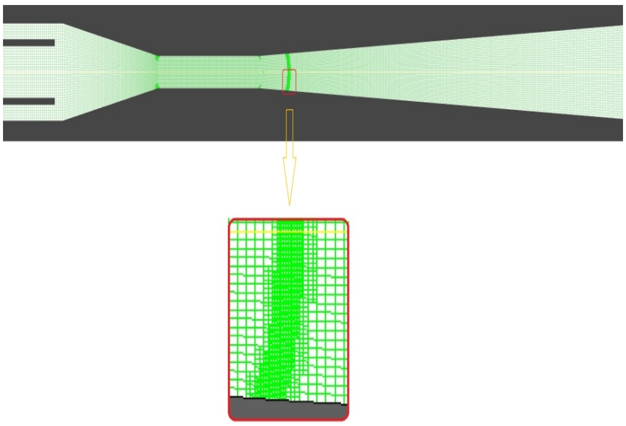

Figure 3. Mesh distribution on Nozzle and refined grid in divergent

2. 3. Computational Domain and Grid Dependency Test Numerical simulations are performed by using FLUENT 6.3 to predict the flow field of the propelling air and mixing with water flow which enters centrally to the nozzle, and finally the acceleration of water droplets. Two-dimensional plane symmetrical model is utilized to reduce the computation time. The computational domain is mapped with quadrilaterals 2D structured grids, and gradient adoption for refining grids is employed to ensure the calculation accuracy as a result of generation of the shockwaves in the supersonic flow. Figure 3 shows grid distribution entire nozzle and refined grid in divergent part. Standard no-slip condition is used at the nozzle wall for continuous phase and the reflect boundary condition with normal and tangent reflection coefficient of 0.1 is employed for dispersed phases. The heat transfer process between the gas and the wall is not considered, thus a fixed heat flux of zero is enforced at the wall.

A grid dependency test is also performed to ensure that the solution dependency on the grid size is <2%. For grid dependency test, the static and dynamic pressure distributions through the nozzle for various mesh sizes were computed. On the other hand, near-wall grid independency test was considered by plotting the y+ value distribution for each mesh size in nozzle. All these studies have been done for single and two-phase flow, separately. Finally, the grid number 13728 as the reliable grid size regarding the mentioned tests was preferred.

2. 4. Air and Water Droplets Two-phase Flow

The generic transport equation, that describes fluid dynamics and heat transfer of multi-phase flow, is a general partial differential equation which may be written as:

$$\frac{\partial(\alpha\rho\phi)}{\partial t}+\nabla\cdot\left(\alpha\rho\vec{V}\phi\right)=\nabla\cdot\bar{\bar{\tau}}+S_{\phi}$$

where in Discrete Phase model, ϕ is a continuum phase variable, α is unity, ρ is the continuum phase density, $\vec{V}$ is the continuum phase velocity, $\bar{\bar{\tau}}$ is the diffusion term, and $S\phi$ is the source term.

The continuity equation for dispersed multi-phase flow expresses that, in any steady state process, the mass flow of component N into the control volume is equal to the rate of mass leaving from that control volume subject to no phase change or chemical reaction for that component. The overall continuity equation for all phases is the combined phase continuity equation:

$$\frac{\partial\rho}{\partial t}+\nabla\cdot\left(\sum_{N}\rho_{N}\alpha_{N}\vec{V}_{N}\right)=0$$

where ρ is the mixture density and given by

$$\rho=\sum_{N}\rho_{N}\alpha_{N}$$

The ideal gas law is used to calculate the density in order to take the compressibility effects into consideration. The governing equations for gas flow include the physical laws of conservation of mass, momentum, and energy. For continuous phase, momentum equation is simplified as:

$$\frac{\partial}{\partial t}(\rho u)+\frac{1}{A}\frac{\partial}{\partial x}\left(A\rho u^{2}\right)=\rho g_{x}-\left(\frac{\partial p}{\partial x}+\frac{P\tau_{w}}{A}\right)+F_{N}$$

where A(x) and P(x) are the area and perimeter of the cross section of duct or pipe flow, respectively, τ_w is the wall shear stress, and FN is the force imposed from other components on the component N.

Similarly, the energy equation can be written in conservation form as:

$$\rho c_{p}\frac{DT}{Dt}=\nabla.(k\nabla T)+\dot{q}+\beta T\frac{Dp}{Dt}+\mu\phi$$

$$and\quad \phi=2\left[\left(\frac{\partial u}{\partial x}\right)^{2}+\left(\frac{\partial v}{\partial y}\right)^{2}+\left(\frac{\partial w}{\partial z}\right)^{2}\right]+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^{2}+\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^{2}+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^{2}-\frac{2}{3}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)^{2}$$

where c_p is the specific heat at constant pressure, k the thermal conductivity, $\dot{q}$ the internal heat generation, ϕ the viscous dissipation function, and β the volumetric coefficient of thermal expansion. In problems of turbulent flow with heat transfer, the turbulence form of energy transport Equation should be solved in addition to Reynolds averaged Navier-Stokes (RANS) equations. For this purpose the instantaneous quantities are replaced by the sum of their mean and fluctuating parts.

$$u=\bar{u}+u'\,,\quad v=\bar{v}+v'\,,\quad w=\bar{w}+w'\,,$$
$$p=\bar{p}+p'\,,\quad T=\bar{T}+T'$$

In most convection problems the simplified form of energy equation uses, where the fluid has negligible viscous dissipation ($\phi\approx 0$), zero internal heat generation ($\dot{q}=0$), and a negligible compressibility effect ($\beta T\,(Dp/Dt)\approx 0$). For this model energy equation can be written as:

$$\rho c_{p}\frac{DT}{Dt}=\nabla.(k\nabla T-\rho c_{p}\overline{u_{i}'T'})$$

The second term inside the parentheses is the turbulence fluctuations due to the heat transfer;hence, it is called “turbulent heat transfer”. To model the new stresses, energy transportation could be simulated to the momentum transferring. In other words, turbulent heat transfer might be written as a new term in diffusion form, which its coefficient comes from this assumption that the same eddies which are responsible for the transport of momentum are responsible for the transport of heat. By modifying turbulence term and using eddy viscosity as the coefficient of diffusion, the energy equation for the Standard and Realizable k-ε model is given by the following:

$$\rho c_{p}\frac{DT}{Dt}=\nabla.\left[\left(k+\frac{\rho c_{p}\varepsilon_{m}}{\Pr_{t}}\right)\nabla T\right]$$

where $\Pr_t$ is the turbulent Prandtl number and is equal to 0.85, and eddy viscosity is calculated by:

$$\varepsilon_{m}=c_{\mu}\frac{k^{2}}{\varepsilon}\quad,\quad c_{\mu}=0.09$$

where k and ε are obtained from two differential equations which represent the transport of turbulence energy, k, and its rate of dissipation, ε [20, 21].

In multi-phase flow, the motion of dispersed phase is controlled not only by the interaction of continuous and dispersed phases, but also by the inter-collisions of dispersed phase. In dilute dispersed phase the effect of inter-collisions of dispersed phase can be vanished. Fluid velocity imposes the lift and drag forces to the particles. On the other hand, in turbulent flow, the fluid velocity is the instantaneous velocity and is decomposed into a mean value and a fluctuating part. Here, there are two problems for calculating the particles motion. First, it needs a proper technique for simulating velocity fluctuations. For this purpose, the Reynolds stress model and k-ε models for single phase are applicable. The second, and more significant problem, is the fact that particles do not follow the fluid path. Therefore, evaluating the fluid velocity in the particle location needs to follow the particle trajectory.

There are two more popular, accurate and widely used approaches to modelling the dispersed phases. One approach that follows individual particles or sample particles, is the Lagrangian approach. The second approach is the Eulerian, which treats the particles as a cloud.

The lagrangian approach is applicable to both dilute and dense flows. In dilute flows, there is just particle-fluid interaction and the motion of the particles is the influence of the particle-fluid interaction, body forces, and particle-wall collisions. Trajectory method, that is a form of Lagrangian approach, is applicable for steady and dilute flows.

The dense and unsteady flows as well as dilute and steady flows can use the lagrangian approach. In dense flows, not only particle-fluid interaction, body forces, and particle-wall collision are important, but also particle-particle collision affects the motion of the particles. Discrete element method is applicable to unsteady and dense flows [22].

The dilute dispersed flow through the chamber at a steady rate is computed by trajectory method. The velocity of dispersed flow in the flow field for a given amount of mass and initial dispersed velocity is calculated from:

$$\frac{d\vec{v}}{dt} = \frac{\vec{F}_f}{m} + \vec{g}$$

where $\vec{g}$ is the gravitational acceleration, and $\vec{F}_f$ is the frictional force between the continuous phase and dispersed phase of mass m. By integrating this equation, the velocity vector of dispersed phase is computed. Hence, the trajectory of dispersed phase is obtained from:

$$\frac{d\vec{x}_q}{dt} = \vec{v}$$

where $\vec{x}_q$ is the droplet or particle (dispersed phase) position.

The dispersed phase temperature distribution, along the trajectory, can be calculated from:

$$\frac{dT_q}{dt} = \frac{1}{mc_q}\left(\dot{Q}_q + \dot{m}L\right)$$

where $\dot{Q}_q$ is the total heat transfer to the dispersed phase, and L is the latent heat of dispersed phase if the phase transition occurs in flow field.

The total number of dispersed phase, N, into the chamber and during a time interval (Δt) can be determined from:

$$N = nV = \sum_{traj} \dot{n}_j \Delta t_j$$

where n is the dispersed phase number concentration, V the volume of chamber, and Δtj the time required for the dispersed phase to pass through the chamber on trajectory j. The mean volume fraction of dispersed phase into the chamber can be described as:

$$\alpha_q = \frac{\sum_{traj} \dot{n}_j \overline{V}_q \Delta t_j}{V}$$

where $\overline{V}_q$ is the average volume of dispersed phase along trajectory j in the chamber. In the same way, the bulk density and temperature of dispersed phase in chamber can be obtained from:

$$d_q = \frac{\sum_{traj} \dot{n}_j \overline{d}_q \Delta t_j}{N}$$

and

$$T_q = \frac{\sum_{traj} \dot{n}_j \overline{c}_q \overline{T}_q \Delta t_j}{Nc_q}$$

where, $\overline{d}_q$, $\overline{c}_q$, and $\overline{T}_q$ are the average density, heat capacity and temperature of dispersed phase along trajectory j in the chamber [4,19,20].

Finally, turbulence in the dispersed phase and momentum transfer due to inter-phase turbulent are modelled by k-ε dispersed turbulence model and k-ε turbulence model for each phase [4].

3. RESULTS AND DISCUSSION

3. 1. Numerical Simulation of Un-premixed Air-water Two-phase Flow through the Nozzle

The Eulerian approach as well as Lagrangian approach might be applicable for simulation of air-water two-phase flow through the nozzle of Figure 1. Depending on the secondary phase volume fraction, as explained in the Section 1, Eulerian and Lagrangian models are selected. For the problem defined in Section 2.1, the calculated secondary phase volume fractions are presented in Table 2. In this table, secondary phase volume fractions have been calculated by assuming the water flow rate of 93 kg/h with constant density of 1000 kg/m3, and air flow rates from Table 1. The density of air has been calculated using the ideal gas law, $\rho = p/(RT)$.

Since secondary phase volume fraction values shown in Table 2 are less than 10%, so regarding the multi-phase limitation presented in Section 1, for numerical simulation of this problem, the Lagrangian approach or Discrete-Phase model is the best choice.

The simulation results for pressure distribution through the nozzle with similar conditions experimented by Lemonnier and Selmer-Olsen [3] are shown in Figure 4. Comparing Figures 2 and 4 shows that the numerical results have less error with respect to experimental results. Also, Figure 4 shows that the numerical results match experimental data quite well, especially in exhaust of nuzzle, which is very important in many nozzle applications.

For more accuracy check of the numerical simulation, the numerical and experimental values of gas flow rate through the nozzle have been compared in Table 3. Even though the pressure profiles do not give any indication of choking, the mass flow-rate is barely affected by the reduction in back pressure.

TABLE 2. The secondary phase volume fractions

P_{out} (bar)	Experimented air flow rate (kg/h) [3]	Estimated secondary phase volume fraction (%)
5.38	37.2	1.48
4.64	54.1	1.02
4.30	58.4	0.95
3.80	61.1	0.90
3.10	62.0	0.89
2.00	62.1	0.89
1.28	62.4	0.89
0.81	62.2	0.89

TABLE 3. The back pressure and air flow rate of nozzle

P_{out} (bar)	Air Flow Rate (kg/h)	
	Experimented [3]	Numerically Simulated
5.38	37.2	42.3
4.64	54.1	57.65
4.30	58.4	60.66
3.80	61.1	61.41
3.10	62.0	61.81
2.00	62.1	61.48
1.28	62.4	61.38
0.81	62.2	61.35

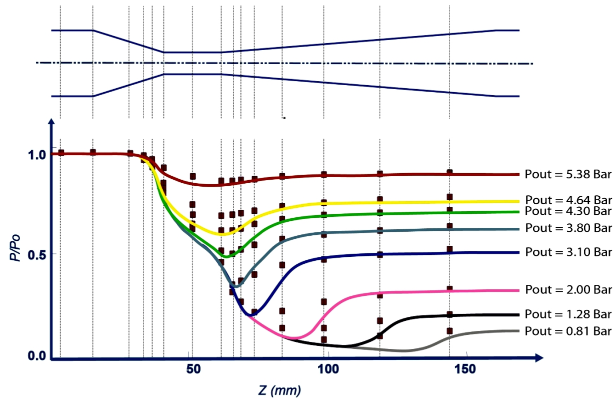

Figure 4. Comparison of simulated pressure distribution with experimental results

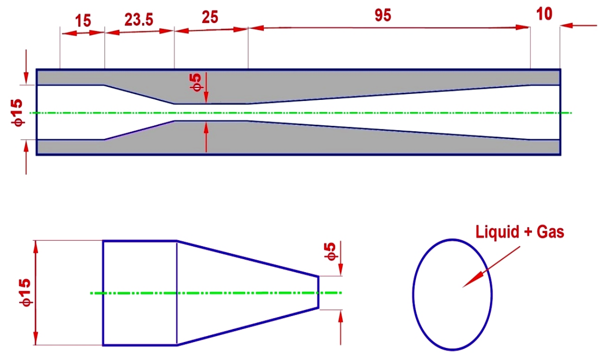

Figure 5. Converging-diverging nozzle with premixed air-water two-phase flow (All dimensions are in mm)

3. 2. Numerical Simulation of Premixed Air-Water Two-phase Flow through the Nozzle vs. Un-premixed Flow

The premixed air-water two-phase flow with similar inlet pressure and water flow rate has been simulated in the nozzle of Figure 5. All geometry and dimensions of this nozzle are similar to that of Figure 1 except for the flow entrance that unlike Figure 1, is un-separated. Therefore, in this study, the homogeneous two-phase flow has been assumed to enter the nozzle.

The validation of simulating of air-water two-phase un-premixed flow has been discussed in the Section 3.1. So, in the following section simulation of premixed flow is compared with un-premixed flow results. Figure 6 shows the comparison between pressure profiles of premixed and un-premixed air-water two-phase flow through the nozzle. As this figure shows, there is very good agreement between two different kinds of flows.

Air velocity contours for single and two-phase flow as well as premixed and un-premixed flows are shown in Figures 7(a-d).

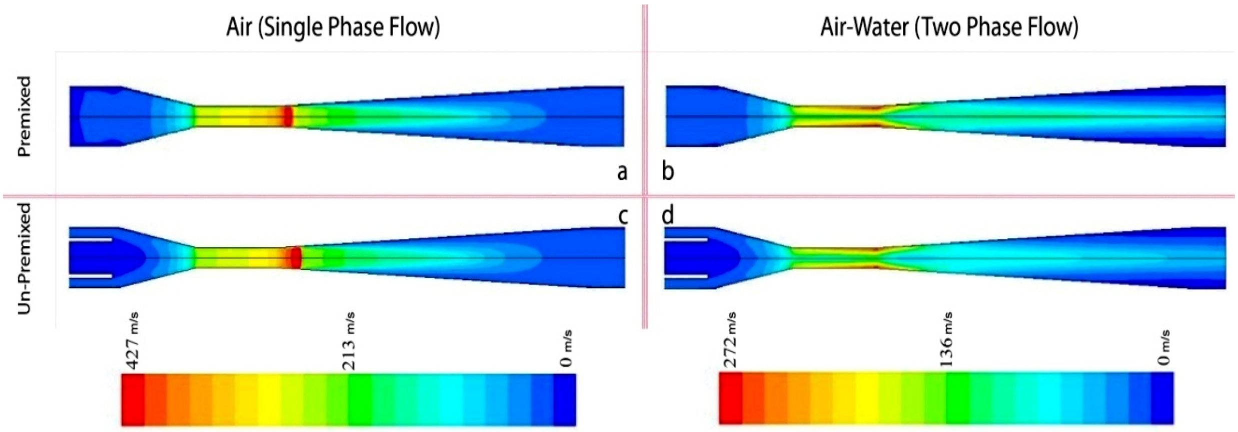

Figure 6. Air velocity contours; a- Single-phase, b- Two-phase un-premixed, c- Single-phase, d- Two-phase premixed

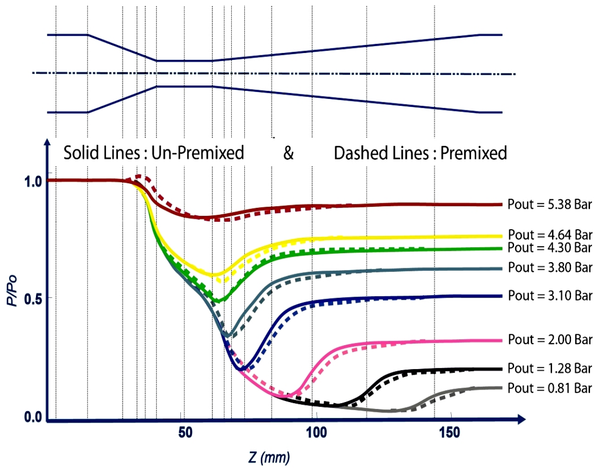

Figure 7. Pressure distribution through the Nozzle for premixed and un-premixed flow

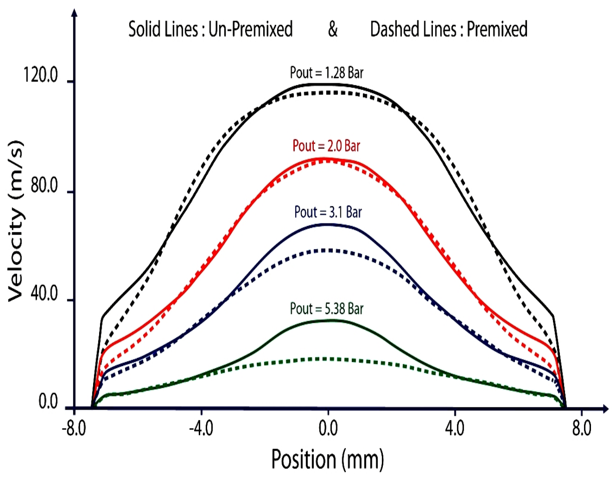

Figure 8. Comparison of exhaust air velocity from the nozzle for premixed and un-premixed air-water two-phase flow

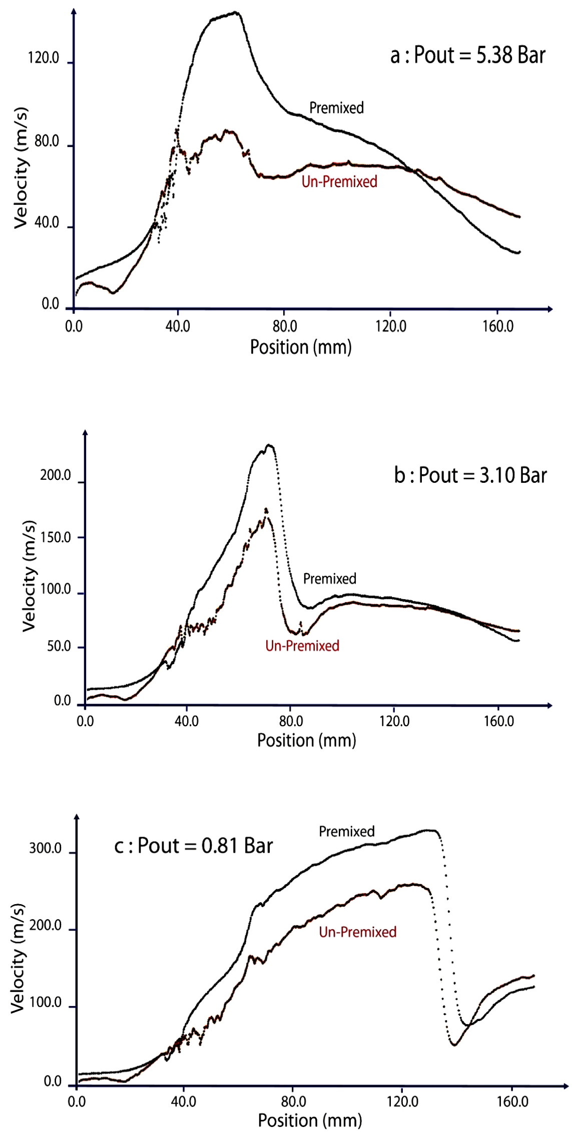

Figure 9. air velocity distribution inside the nozzle

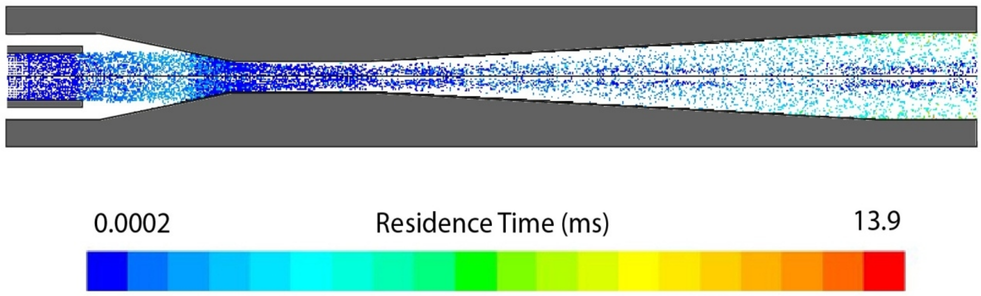

Figure 10. Water droplets traces colored by droplets residence time through the nozzle by exhaust back pressure of 5.38 bar

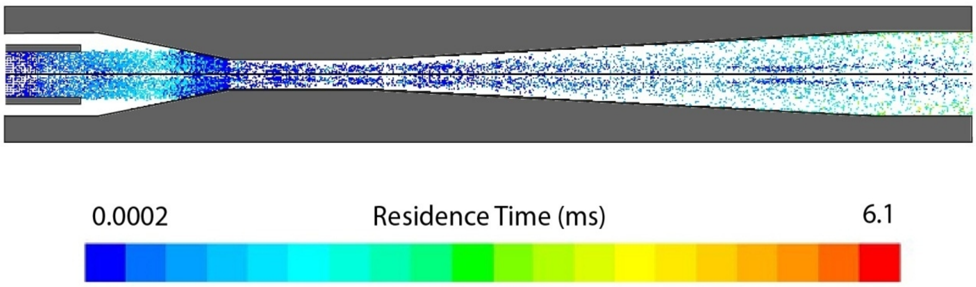

Figure 11. Water droplets traces colored by droplets residence time through the nozzle by exhaust back pressure of 3.1 bar

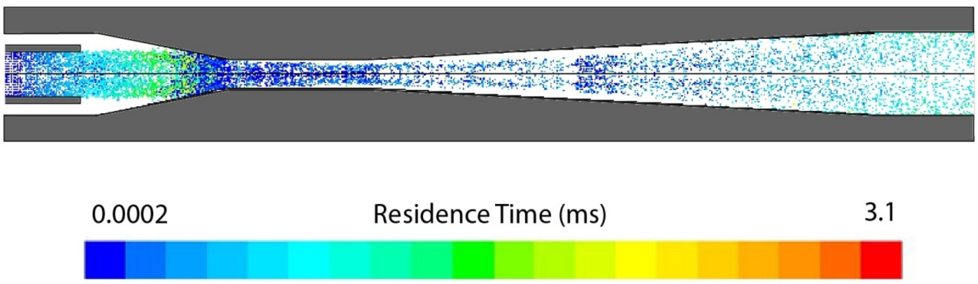

Figure 12. Water droplets traces colored by droplets residence time through the nozzle by exhaust back pressure of 0.81 bar

The exhaust air velocity from the nozzle for premixed and un-premixed air-water two-phase flow by various air pressure differences are shown in Figure 8.

Figure 8 shows that by decreasing air exhaust pressure (or increasing pressure difference of air flow) the air exhaust velocity for premixed and un-premixed two-phase flows follow almost the same trend.

For further study on the effects of premixed and un-premixed on two-phase flow through the nozzle, air velocity distribution inside and along the centre of nozzle has been compared for different back pressure in Figures 9(a-c).

Obviously, the flow of air and consequently water droplet are because of the pressure difference, and by increasing the pressure difference between inlet and back pressure, the momentum of flow makes augment. On the other hand, the raising of momentum helps to improve the air-water mixture quality. This phenomenon has been clearly shown in Figures 10 to 12.

By decreasing the back pressure from 5.38 bar in Figure 9(a) to 3.10 bar in Figure 9(b), the nozzle exhaust quality goes up and air-water two-phase flow is going to have good mixture when the back pressure is further decreased from 3.10 bar to 0.81 bar on Figure 9(c).

4. CONCLUSION

A numerical study has been conducted to investigate air-water two-phase flow through the nozzle. In this study the Realizable k-ε and Discrete Phase models were utilized for simulating multi-phase flow through the converging-diverging nozzle. The effects of various air exhaust pressure, and premixed or un-premixed flow on nozzle exhaust air velocity, and velocity and pressure distribution through the nozzle were presented. The following results are concluded by this research:

1. Adding/increasing of water droplets to flowing air through the nozzle eliminates the shock wave.
2. The flow quality in Nozzle outlet for Premixed and Un-Premixed flow will be the same if the pressure difference increases up to 3.8 bar, and consequently:
3. There is no need for use of mixer chamber if the pressure difference flow through the Nozzle goes up to 3.8 bar.

5. ACKNOWLEDGEMENT

The authors would like to acknowledge the financial, academic and technical support of the Anglia Ruskin University.

6. REFERENCES

1. Anderson, I. E. and Terpstra, R. L., "Progress toward gas atomization processing with increased uniformity and control", ***Materials Science and Engineering: A***, Vol. 326, No. 1, (2002), 101-109.
2. Lear, W. and Sherif, S., "Two-phase non-dissipative supersonic nozzle flow analysis for maximum condensed phase momentum flux", ***Acta Astronautica***, Vol. 40, No. 10, (1997), 707-712.
3. Lemonnier, H. and Selmer-Olsen, S., "Experimental investigation and physical modelling of two-phase two-component flow in a converging-diverging nozzle", ***International Journal of Multiphase Flow***, Vol. 18, No. 1, (1992), 1-20.
4. M., A., "Investigation of three-phase nozzle flow (water- sand - air) in an innovative sand-blasting system", in Anglia Ruskin University, UK, (2011).
5. Chien, K.-Y., "Predictions of channel and boundary-layer flows with a low-reynolds-number turbulence model", ***AIAA Journal***, Vol. 20, No. 1, (2012).
6. Benyahia, S., Syamlal, M. and OBrien, T. J., "Evaluation of boundary conditions used to model dilute, turbulent gas/solids flows in a pipe", ***Powder Technology***, Vol. 156, No. 2, (2005), 62-72.
7. Chalermsinsuwan, B., Piumsomboon, P. and Gidaspow, D., "Kinetic theory based computation of psri riser: Part i—estimate of mass transfer coefficient", ***Chemical Engineering Science***, Vol. 64, No. 6, (2009), 1195-1211.
8. Das, A., De Wilde, J., Heynderickx, G. and Marin, G., "CFD simulation of dilute phase gas–solid riser reactors: Part ii—simultaneous adsorption of SO_2–NO_x from flue gases", ***Chemical Engineering Science***, Vol. 59, No. 1, (2004), 187-200.
9. De Wilde, J., Van Engelandt, G., Heynderickx, G. J. and Marin, G. B., "Gas–solids mixing in the inlet zone of a dilute circulating fluidized bed", ***Powder Technology***, Vol. 151, No. 1, (2005), 96-116.
10. Gerber, A. and Kermani, M., "A pressure based eulerian–eulerian multi-phase model for non-equilibrium condensation in transonic steam flow", ***International Journal of Heat and Mass Transfer***, Vol. 47, No. 10, (2004), 2217-2231.
11. Gidaspow, D., "Multiphase flow and fluidization: Continuum and kinetic theory descriptions", Academic Press, (1994).
12. Jiradilok, V., Gidaspow, D., Damronglerd, S., Koves, W. J. and Mostofi, R., "Kinetic theory based CFD simulation of turbulent fluidization of FCC particles in a riser", ***Chemical Engineering Science***, Vol. 61, No. 17, (2006), 5544-5559.
13. Pirker, S., Kahrimanovic, D., Kloss, C., Popoff, B. and Braun, M., "Simulating coarse particle conveying by a set of eulerian, lagrangian and hybrid particle models", ***Powder Technology***, Vol. 204, No. 2, (2010), 203-213.
14. Kheloufi, K. and Amara, E.-H., "Numerical modelling of gas/particles diphasic jet in laser cladding by coaxial nozzle", ***Physics Procedia***, Vol. 5, (2010), 347-352.
15. Moshfegh, A., Shams, M., Ebrahimi, R. and Farnia, M. A., "Two-way coupled simulation of a flow laden with metallic particulates in overexpanded tic nozzle", ***International Journal of Heat and Fluid Flow***, Vol. 30, No. 6, (2009), 1142-1156.
16. Yin, S., Wang, X.-f. and Li, W.-y., "Computational analysis of the effect of nozzle cross-section shape on gas flow and particle acceleration in cold spraying", ***Surface and Coatings Technology***, Vol. 205, No. 8, (2011), 2970-2977.
17. Zekovic, S., Dwivedi, R. and Kovacevic, R., "Numerical simulation and experimental investigation of gas–powder flow from radially symmetrical nozzles in laser-based direct metal deposition", ***International Journal of Machine Tools and Manufacture***, Vol. 47, No. 1, (2007), 112-123.
18. Zhu, G., Li, D., Zhang, A. and Tang, Y., "Numerical simulation of metallic powder flow in a coaxial nozzle in laser direct metal deposition", ***Optics & Laser Technology***, Vol. 43, No. 1, (2011), 106-113.
19. Fluent-Inc, "Fluent 6.3 User's guide", Fluent Inc, (2006).
20. Bejan, A. and Kraus, A. D., "Heat Transfer Handbook", John Wiley, (2003).
21. White, F. M., "Fluid Mechanics", McGraw-Hill, (2003).
22. Crowe, C. T., "Multiphase Flow Handbook", CRC, (2005).

The Effects of Pressure Difference in Nozzle's two Phase Flow on the Quality of Exhaust Mixture

M. Abbasalizadeh[a], S. Jafarmadar[a], H. Shirvani[b]

[a] *Department of Mechanical Engineering, Urmia University, Urmia, Iran*
[b] *Department of Computing Science, Anglia Ruskin University, Chelmsford, CM1 1SQ, UK*

PAPER INFO

Paper history:
Received 07 October 2012
Received in revised form 12 December 2012
Accepted 24 January 2013

Keywords:
Compressible Flow
Multi-phase Flows
RANS: Reynolds Averaged Navier-Stokes
Supersonic
Turbulence Models
Two-phase Flows Premixed
Un-premixed Nozzle Flow

چکیده

در اکثر کاربردهای نازل با دو فاز مایع–گاز، کیفیت مخلوط در خروجی نازل و همچنین سرعت از پارامترهای مهم هستند. از طرف دیگر در برخی کاربردهای صنعتی همچون رنگ‌کاری و یا تزریق آب درسیستم مکش اجباری (سوپرشارژ و توربوشارژ) در موتورهای احتراق داخلی کیفیت جت سوخت از اهداف مهم طراحی هستند. در این حالت‌ها و برای بهبود راندمان پاشش نازل‌های تزریق دو فاری هوا و آب بیشتر برجسته می باشد. در هر دو حالت نه تنها نازل مخلوط را به سرعت‌های بالا شتاب می‌دهد، بلکه همچنین باید یک مخلوط با کیفیت بالا در خروجی داشته باشد. در این مطالعه جریان توربولانت دو فازی پیش‌امیخته و غیرپیش‌آمیخته در داخل نازل با استفاده از روش اویلری–لاگرانژی شبیه‌سازی شده است. فاز گازی به صورت فاز پیوسته بوده و از طریق معادلات نویر استوکس بررسی می‌شود، در حالی که فاز مایع بهصورت فاز پراکنده بوده و با در نظر گرفتن تعداد زیادی قطرات در داخل میدان بررسی می‌شود. پروفیل‌های فشار، سرعت و عدد ماخ و همچنین دبی هوای جاری و زمان اقامت قطرات در داخل نازل برای فشارهای بالادست متفاوت محاسبه شده است. نتایج کار حاضر با استفاده از داده های تجربی پروفیل فشار و دبی هوای در حال جریان برای مخلوط غیرپیش امیخته اعتباردهی شدند.

doi: *10.5829/idosi.ije.2013.26.05b.12*