VOL. 11, NO. 1, JANUARY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL SIMULATION OF DROPLET DISPERSION AND EVAPORATION AND OF AIR VELOCITY IN A JET BURNER

Imed Miraoui[1, 3] and Mouldi Chrigui[2]
[1]Department of Mechanical Engineering, Aljouf University, Engineering College, Skaka, KSA
[2]Department of Mechanical Engineering, Technical University of Darmstadt, Germany
[3]University of Gafsa, Sciences College, MEER Research Unit, Gafsa, Tunisia
E-Mail: aimed_mir@yahoo.fr

ABSTRACT

Many practical engineering and industrial applications involve droplet evaporation, turbulent flow, and spray combustion. In this work the droplet behaviour within a lean premix prevaporised (LPP) burner and the air velocity of turbulent flow in coaxial jet burner are studied numerically by using the method of Eulerian-Lagrangian approach and RANS-modeling approach, respectively. Three-dimensional computational fluid dynamics (CFD) code has been used and the gas phase equations are solved using the finite volume method. The droplet evaporation rate has been checked by the distribution of the droplet diameters. The numerical results of droplet mean axial velocity, droplet diameters, and mean swirl air velocity have been compared to the experimental measurements. Results showed good agreement between predicted and experimental data.

Keywords: simulation, droplet, evaporation, turbulent flow.

INTRODUCTION

Spray combustion is used extensively in gas turbine and internal combustion engines. In order to achieve the combustion and pollution control as efficiently as possible, deeper detailed analysis of involved phenomena is essential. Droplet evaporation may be a source of local extinction or variation of the gas temperature which causes pollutants formation and decrease device performances.

Several numerical and experimental studies have been proposed to investigate the problems of spray dispersion, evaporation and combustion [1-4].

Oefelein *et al*. [5] have studied the difference between high-pressure and classical low-pressure evaporation models. They have indicated that there is a difference between these two evaporation models (the classical low-pressure and the high-pressure evaporation models).

Miller *et al*. [6] have evaluated the existing evaporation models which are applicable to describe the droplets of different diameters at low pressure. As regards the numerical simulations, when we want to simulate unsteady flows in which we have a large Reynolds number, the method of Large Eddy Simulation (LES) may be used. The Reynolds Averaged Navier Stokes (RANS) model is recommended in the case of highly turbulent flow, it is the most used method for industrial flows. In general, we use LES for large scales and RANS simulation for small scales.

A numerical simulation of turbulent jet flow was presented by Zhou *et al*. [7]. They have calculated the steady free jet flow by applying the k–ε turbulent model. The evaporation and the reaction of fuel droplets in forced turbulent flows were investigated by Mashayek [8]. He has found that the rate of evaporation has a major effect on combustion process.

A simulation of spray combustion and vaporization was performed by Réveillon and Vervisch [9] by applying a 3 dimensional DNS simulation (Direct Numerical Simulation).

The aim of this work is to investigate the droplet behaviour and the air velocity of turbulent flow by using numerical simulation. The numerical results are compared to the experimental measurements.

MODELING METHODS

RANS-modeling approach is used to describe the turbulent fluid phase. The RANS (Reynolds Averaged Navier Stokes) equations are closed in standard k–ε model by assuming that (Boussinesq Hypothesis) the turbulent stresses are proportional to the mean velocity gradients and that the constant of proportionality is the turbulent viscosity. The turbulent viscosity(μ_t) is correlated with the turbulent kinetic energy (k) and the dissipation rate of turbulent kinetic energy (ε), as indicated in the following relation

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

The transport equations were solved for the components of velocity, mass conservation, turbulent kinetic energy k, and dissipation rate of turbulent kinetic energy ε [9]. The transport equations were modified for two phase flow description by including source terms for phase exchange and phase transition processes [10].

Characteristic parameter was introduced (Equation (1)) to characterize the deviation of phase transition on the droplet surface from the equilibrium [9]. For the case of a non-equilibrium evaporation model, the fraction of the molar mass χ_s is found by using the Equation (1).

$$\chi_{s,neq} = \chi_{s,eq} - \left(\frac{L_K}{d/2}\right)\beta_L \qquad (1)$$

Where $\beta_L = -\left(\frac{3\,\mathrm{Pr}_G\ \tau_d}{2}\right)\frac{\dot{m}}{m}$ is the half of the blowing Peclet number, d is the diameter of the droplet, Pr_G is the Prandtl number, L_K is the Knudsen length and τ_d is the relaxation time of the particle.

Abramson and Sirignano [11] have introduced modified Nusselt and Sherwood numbers for the acquisition of Stefan's flow.

We need a dispersion model to determine the fluctuations of the velocity of the gas phase at the droplets position. We have used the Markov-Sequence-Model [12] based on the determination of Lagrangian and Eulerian correlations.

NUMERICAL APPROACH

The simulation has been performed using three dimensional computational fluid dynamics (CFD) code and the gas phase equations are solved using the finite volume method. The diffusion terms are discretized with flux blending schemes on a non orthogonal block structured grid.

The flow is solved by CFD-FASTEST as numerical code using a block structured grid. As model we have used K-ε model with finite volume method. The Lagrangian equations for droplets are discretized using first order scheme and solved explicitly.

The droplet injection is based on a stochastic approach by considering the mass flux of the droplet and the distribution of the droplets size obtained from the experimental measurements at the inlet near the nozzle exit.

DESCRIPTION OF THE CONFIGURATIONS

Partially Prevaporized Spray Burner (PPSB) consists of ultrasonic nozzle has been studied using water droplets (case of only evaporation).

A set of sinter metal plates were used to heat the main air in a way that the carrier phase temperature is set to 90°C. The flow rate is 300*ln*/*min.* As shown in the Figure-1, the mixing with the spray occurs in the tube which has the distance "s" [13].

For the single annular combustor, the overall mesh is about 350 000 control volumes. The grid is composed by 18 multiply-connected domains: they consist of several separate flow-paths which interact with each other.

In order to generate the mesh, we have used Cartesian coordinates and hexahedral cells. A turbulence intensity of 10% of the resultant velocity through an inlet has used to determine the inlet conditions for the turbulent kinetic energy. The dissipation rate distribution is estimated using the following expression.

$$\varepsilon = C_\mu^{3/4}\,\frac{k^{3/2}}{0.41\cdot\Delta r}$$

The turbulent flow has been investigated by using the configuration based on the experimental study of Marchione *et al*. [13].

The burner used in this study consists of two concentric circular ducts of length 350 mm. The inner diameter of the outer duct is 35 mm. The burner exit has a conical bluff body with diameter of 25 mm.

A computational grid consists of 1 500 000 cells has been used. The 3D axisymmetric block-structured CFD grid is shown in Figure-2.

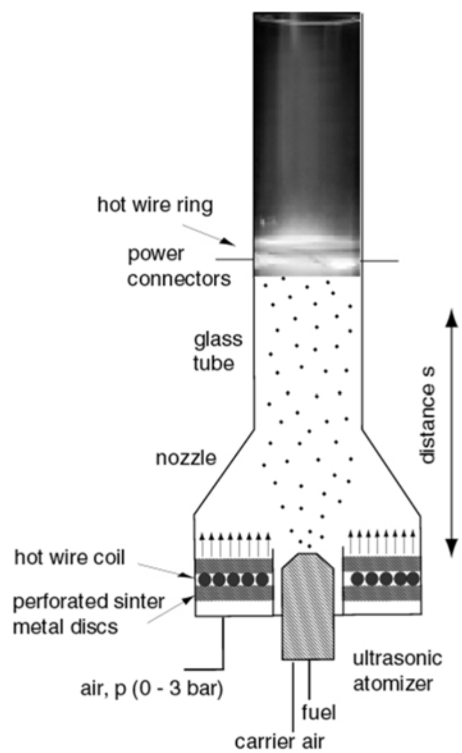

Figure-1. Partially Prevaporized Spray Burner: experiment [14].

VOL. 11, NO. 1, JANUARY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

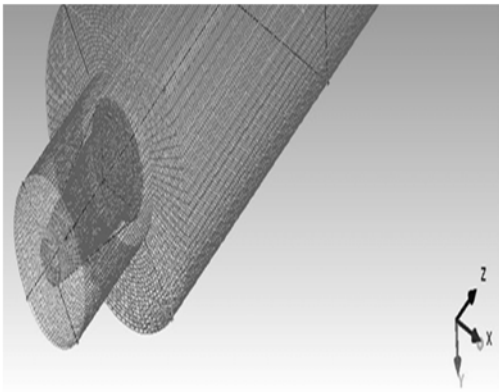

Figure-2. 3D Axisymmetric block-structured CFD Grid.

RESULTS AND DISCUSSIONS

The contours plots of the droplet mean diameter is shown in Figure-3 (case of heated main air flow of 90°C). It can be seen that a large number of droplets are able to come over the evaporation zone.

Turbulent kinetic energy of the mean axial air velocity and the swirl air velocity of the cold flow are shown in Figure-4 and Figure-5. The highest turbulent kinetic energy is observed at the zone near the bluff body. The recirculation zone created at the side zone causes a rapid decrease in turbulent kinetic energy.

A comparison between numerical and experimental results of the droplet mean axial velocity as a function of the radius is indicated in Figure-6 (case of preheated air flow at 90°C). The distance from the fuel nozzle outlet is s= 810 mm. It can be seen that there is a good agreement between numerical and experimental results. From Figure-6, it is clearly observed that at fixed distance from the inlet, the velocity varies due to the anisotropic nature of the dispersed phase flow.

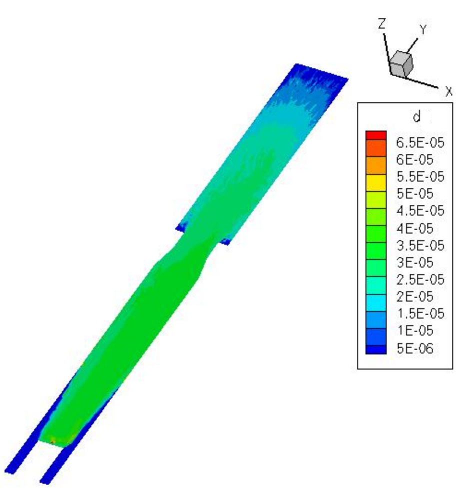

Figure-3. Contours plots of droplet mean diameter.

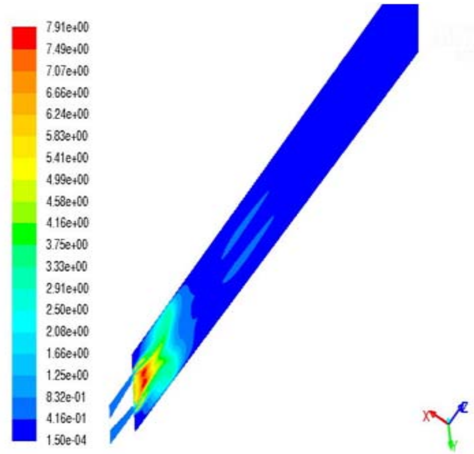

Figure-4. Turbulent Kinetic Energy k (m^2/s^2) of the mean axial air velocity of the cold flow in absence of spray.

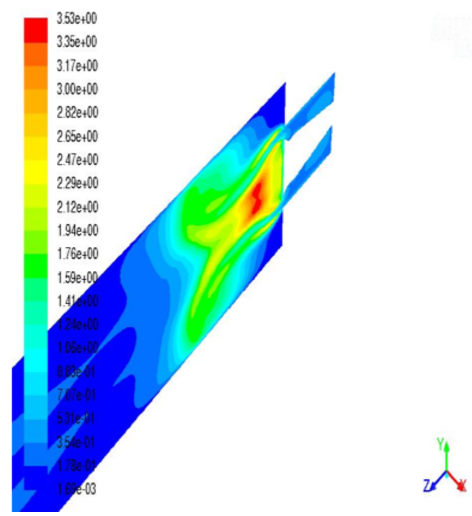

Figure-5. Turbulent Kinetic Energy k (m^2/s^2) of the swirl air velocity of the cold flow in absence of spray.

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

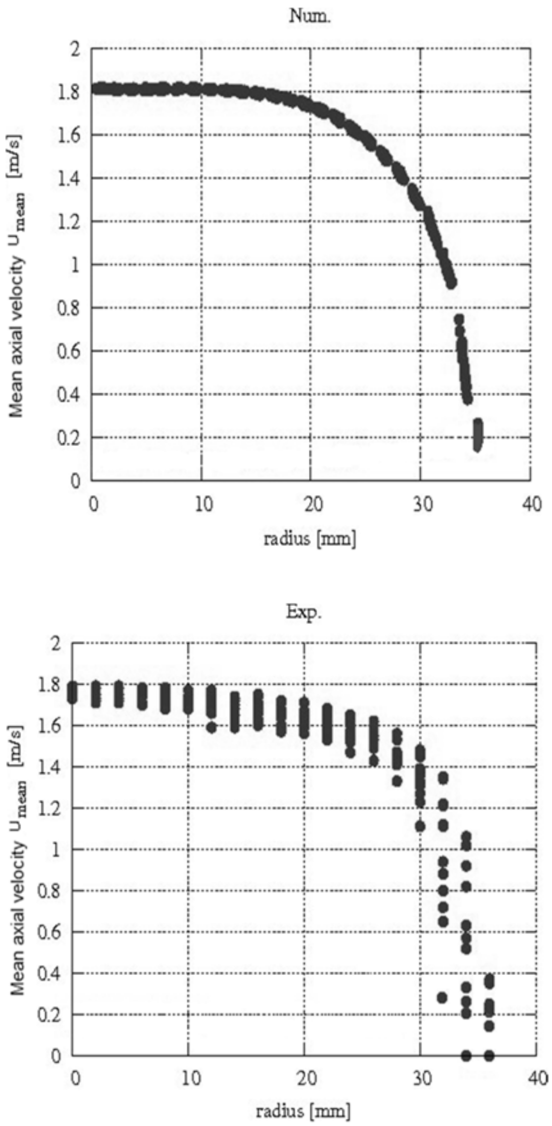

Figure-6. Comparison between numerical and experimental results of the droplet mean axial velocity at a distance of 810 mm from the inlet.

In order to analyze the droplet evaporation rate, the variation of the droplet diameters was studied. Figure-7 shows a comparison between numerical and experimental results of the variation of surface area mean diameter, volume mean diameter, and the Sauter mean diameter as a function of the radius at a distance of 610 mm from the inlet (case of preheated air flow at 90°C). It is clearly observed that there is a good agreement between numerical and experimental results. Only a small disagreement is observed for the volume mean diameter. Also, the results show that the boundary conditions have a major effect on the distribution of the droplet diameters.

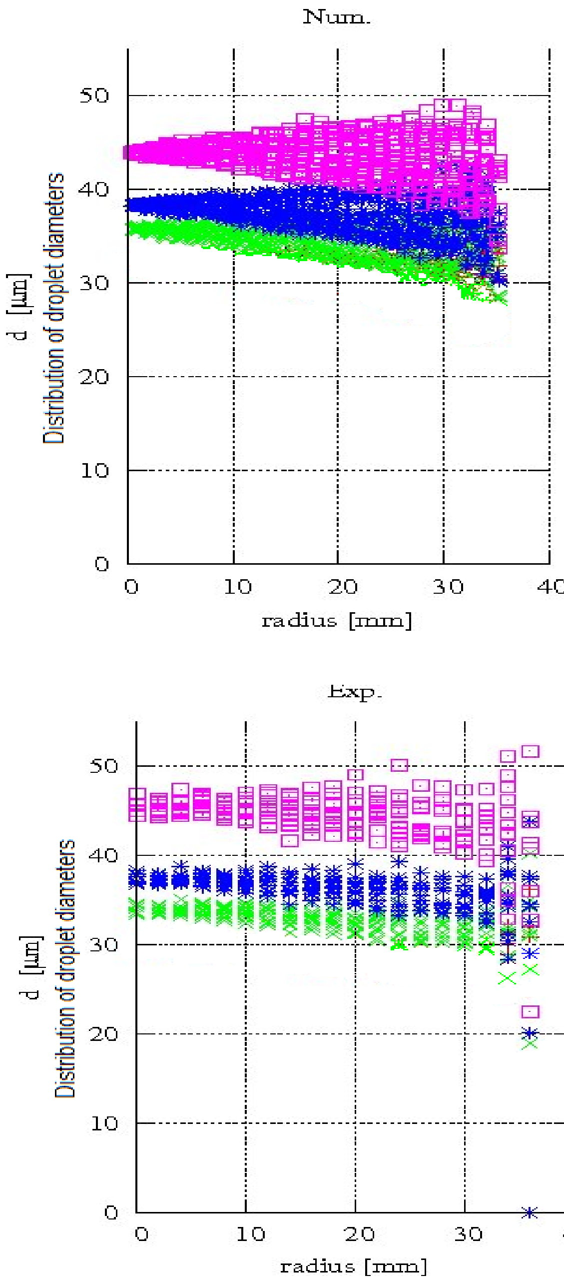

Figure-7. Comparison between numerical and experimental of the surface area mean diameter (blue color), volume mean diameter (green color), and the Sauter Mean Diameter (mauve color) at a distance of 610 mm from the inlet.

Figure-8 shows a comparison between predicted and measured of radial profile of the mean swirl velocity, for air flow in absence of spray. The experimental data are from the experimental results of Marchione *et al.* [13]. It was observed a reasonable agreement between numerical and experimental results. There is an under estimation of the air velocity when the radial position is more than 20 mm, which corresponding to the recirculation zone.

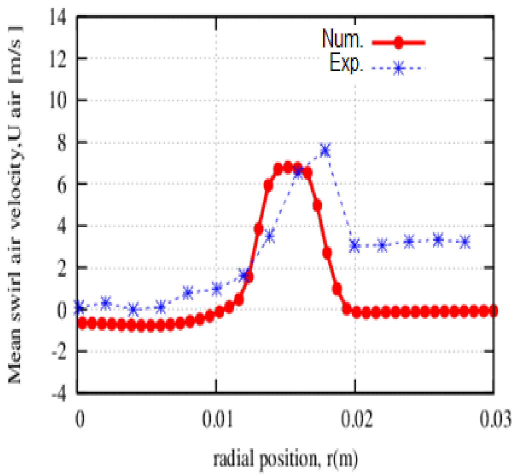

Figure-8. Comparison between predicted and measured radial profile of the mean swirl air velocity.

CONCLUSIONS

The droplet behaviour within a lean premix prevaporised burner and the air velocity of turbulent flow in coaxial jet were simulated. The method of Eulerian-Lagrangian approach was used to investigate the dispersion and the evaporation of water droplets. The air velocity and the kinetic energy of the axial and swirl air velocity of the turbulent flow were investigated using the k–ε turbulent model. A comparison between experimental and numerical results has been presented.

The results show that the highest turbulent kinetic energy are at the zone near the bluff body. The recirculation zone created at the side zone causes a rapid decrease in turbulent kinetic energy. The predicted axial droplet velocity and the predicted droplet diameters are in good agreement with the experimental results. The predicted mean air velocities show reasonable agreement with experimental results, especially at low radial position.

ACKNOWLEDGEMENTS

The authors acknowledge the financial support of this research project from Aljouf University, KSA. Grant no. 34/244.

REFERENCES

[1] Faeth G. M. 1996. Spray combustion phenomena. Prog. Energy Comb. Sci. 26 (1): 1593-1612.

[2] Borghi R. 1996. The links between turbulent combustion and spray combustion and their modelling. In: 8th International Symposium on Transport Phenomena in Combustion, pp. 1–18.

[3] Sommerfeld M. and QIU H. H. 1993. Characterization of particle-laden confined swirling flows by phase doppler anemometry and numerical calculation. Int. J. Multiphase Flow. 19 (6): 1093–1127.

[4] Li T., Nishida K. and Hiroyasu H. 2011. Droplet size distribution and evaporation characteristics of fuel spray by a swirl type atomizer. Fuel. 90, 2367–2376.

[5] Oefelein J.C., Aggarwal S.K. 2000. Toward a unified high pressure drop model for spray simulation. Center for turbulence research, Proceedings of summer program, pp. 193-205.

[6] Miller R.S., Harstad K., Bellan J. 1998. Evaluation of equilibrium and non-equilibrium evaporation models for many droplet gas liquid flow simulation. Int. J. of Multiphase flow. 24: 1025-1055.

[7] Zhou X., Sun Z., Durst F. and Brenner G. 1999. Numerical simulation of turbulent jet flow and combustion. Computers and Mathematics with Applications. 38: 179-19.

[8] Mashayek F. 1999. Simulations of reacting droplets dispersed in isotropic turbulence. AIAA J. 37: 1420-1425.

[9] Reveillon J. and Vervisch L. 1998. Accounting for spray vaporization in turbulent combustion modeling. Center for Turbulence Research, Proceedings of the Summer Program.

[10] Sadiki A., Chrigui M., Maneshkarimi and Janicka J. 2005. Modeling and simulation of effects of turbulence on vaporization, mixing and combustion of liquid fuel sprays. J. of turbulence flow and combustion.

[11] Abramson B. and Sirignano W.A. 1989. Droplet vaporization Model for Spray Combustion Calculations. Int. J. Heat Mass Transfer. 32: 1605-1618.

[12] Chrigui M., Ahmadi G. and Sadiki A. 2004. Study on Interaction in Spray between Evaporating Droplets and Turbulence Using Second Order Turbulence RANS Models and a Lagrangian Approach. Progress

in Computational Fluid Dynamics. Special issue, pp 62-174.

[13] Marchione T., Ahmed S.F., Mastorakos N. 2009. Ignition of turbulent swirling n-heptane spray flames using single and multiple sparks. Combustion and Flame. 156: 166-180.

[14] Baessler S., Moesl K. and Sattelmayer T. 2006. Nox emissions of premixed partially vaporized kerosene spray flame. Proceedings of ASME Turbo Expo 2006 Power for Land, Sea, and Air May 08-11, Barcelona, Spain.