Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30926

SIMULATION OF LIQUID FUEL ATOMIZATION IN AN INDUSTERIAL SPRAY NOZZLE OF A POWERPLANT BOILER

Iman Mirzaii
Graduate Student,
Ferdowsi University of Mashhad,
Mashhad, Iran

Hasan Sabahi
Graduate Student,
Ferdowsi University of Mashhad,
Mashhad, Iran

Mohammad Passandideh-Fard
Associate professor,
Ferdowsi University of Mashhad,
Mashhad, Iran

Nasser Shale
R&D Specialist,
Touss Powerplant of Mashhad,
Mashhad, Iran

ABSTRACT

In this study, the liquid fuel atomization in the injector nozzle of the combustion chamber of a powerplant boiler is numerically simulated. The atomization of a liquid fuel injector is characterized by drop size distribution of the nozzle. This phenomenon plays an important role in the performance of the combustion chamber such as the combustion efficiency, and the amount of soot and NOx formation inside the boiler. The injector nozzle, considered in this study, belongs to a powerplant boiler where the liquid fuel is atomized using a high pressure steam. First, the geometric characteristics of the injector are carefully analyzed using a wire-cut process and a CAD model of the nozzle is created. Next, one of the nozzle orifices and the atomization zone where the high pressure steam meets the liquid fuel is recognized. The computational domain is extended long enough to cover the whole atomization zone up to the end of the orifice. The flow governing equations are the continuity and Navier-Stokes equations. For tracking the liquid/gas interface, the Volume-of-Fluid (VOF) method along with Youngs' algorithm for geometric reconstruction of the free surface is used. The simulation results show the details of the liquid and steam flow inside the nozzle including velocity distribution and shape of the liquid/gas interface. It is found that the liquid breakup to ligaments and the atomization of liquid to droplets do not occur inside the nozzle orifice. A liquid jet with certain cross sectional shape leaves the orifice surrounded by a high speed steam. The numerical model provides the shape of the liquid jet, and the steam and fuel velocity distributions at the exit of the nozzle orifice. These parameters are then correlated to the final drop size distribution using analytical/experimental correlations available in literature.

INTRODUCTION

The fuel atomization process has a crucial effect on the combustion. Finer atomization of fuel by the atomizer nozzle leads to higher combustion efficiencies and affects amount of emission in exhaust gases. In this respect, type and performance parameters of atomizer nozzles, which directly affect the fuel particle size in combustion process, play an important role in power generation industry and atomization has been a field of interest for many years for scientists and engineers. There are many ways to generate a spray using rotary cups, twin fluids, pressure swirl, fan, ultrasonic atomizer, etc. [1]. Also the increasing need of controlling the distribution of droplet size, penetrating length of fuel jet and many other effective parameters still motivate researchers to invent new methods of atomization or modify existing methods [2-4].

For the industrial applications where viscous fuel oils have to be handled in large scale facilities such as boilers and furnaces, the number of methods giving necessary performance parameters is dramatically reduced. In these situations, one of the nozzles most commonly used is the steam assisted type with a "Y" configuration. The "Y" configuration orifices, which form a ring shaped layout on the outlet cross section of this kind of nozzle, causes a hollow conical shape of injected fuel jets in the boiler or industrial furnaces. In each orifice, fuel is injected with an angle into the exit port, where it mixes with the atomizing fluid (steam) [4]. But these nozzles when atomizing and burning heavy fuel oil or crude petroleum suffer from relatively large amount of steam, normally at high speeds. This high speed steam leads to local extinction of flame and also cooling the reaction zone of combustion. Also high speed steam will cause elongation of flame that frequently ends in a contact with the boiler walls. These effects will definitely affect all the parameters in boilers and furnaces and must be avoided. To overcome this drawback there have been wide spread attempts to substitute this kind of atomizer with more efficient ones.

The new policies set for the use of energy resources have initiated attempts to renew or modify the available powerplant facilities in order to increase their performance. This goal cannot be achieved without understanding the physical phenomenon occurring in these facilities. This study deals with the simulation of liquid fuel atomization in an industrial spray nozzle of a powerplant boiler. The considered injector nozzle is a Y-type where the liquid fuel is atomized using a high pressure steam. The main objective of this study is to investigate the atomization process that occurs in the atomizer. This study is the start of a project for modifying the operating parameters of the boiler.

GEOMETRIC CHARACTERISTICS OF THE NOZZLE

To investigate the atomization process, the geometric characteristics of the nozzle must be known first. As there was no clear information or drawing to show the exact 3D features of the injector, a sample nozzle was cut in different cross sections by a wire-cut process and the geometric characteristics of the injector were carefully analyzed. Figure 1 shows a longitudinal cross sectional view of the sample nozzle. Then a CAD model of the nozzle was created based on the measured dimensions of the sample nozzle. Figure 2 shows longitudinal cross section of the CAD model in which the high pressure steam channels are indentified with a red color. Next, one of the nozzle orifices and the atomization zone where the high pressure steam meets the liquid fuel was recognized. Figure 3 shows another cross section of the nozzle in which a nozzle orifice can be seen. As observed in the schematic figure of the orifice (Figure 4), the steam channel had a diameter of 2.216 mm with a divergency of 3.3° and the fuel channel had a diameter of 2.5 mm. The fuel channel was approaching the atomization zone with a 33° angle with respect to the center axis of the steam channel. Figure 5 shows the computational domain that was created exactly based on the values given in Figure 4. The computational domain considered for numerical simulation covered nearly 5 mm upstream of both the steam and fuel streams and the domain was also extended long enough to cover the whole atomization zone up to the end of the orifice. Due to the symmetry, only one half of the orifice was considered in the simulation to obtain the flow characteristic of the steam and the fuel. Two separate regions were considered in the entrance of the steam and the fuel channels in order to use lighter density of cells while meshing the computational domain to reduce the unnecessary increase of cells as much as possible.

Figure 1. Longitudinal cross section of the sample nozzel

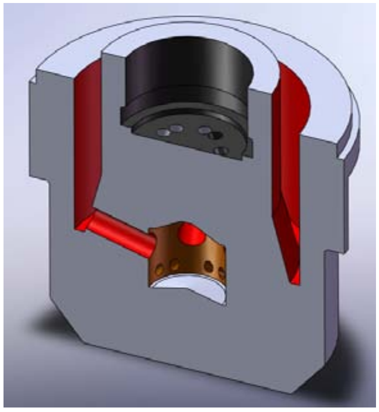

Figure 2. The 3D CAD model of the sample nozzle

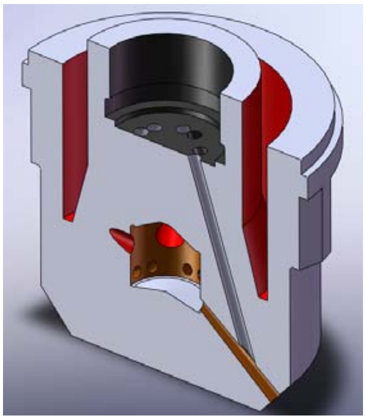

Figure 3. Longitudinal cross section of the nozzle to detect one orifice

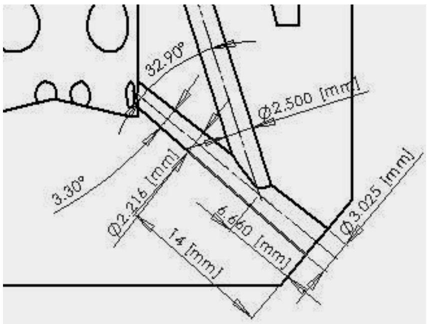

Figure 4. The atomization zone where the high pressure steam meets the fuel

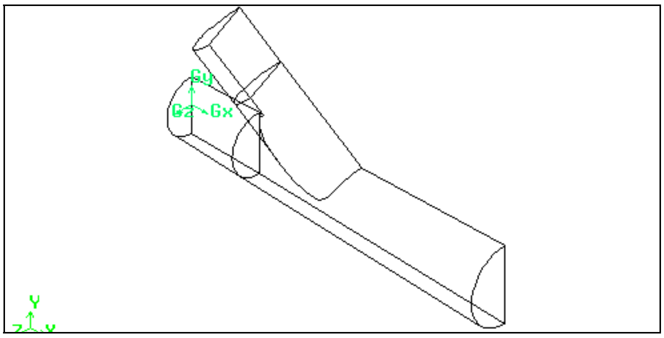

Figure 5. The 3D computational domain based on values given in Figure 4

NUMERICAL METHOD

To investigate the behaviour of the steam and fuel in numerical analysis it was inavitable to use a numerical method capable of tracking the free surface. The Volume-of-Fluid (VOF) method can model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain in the transient tracking of any liquid-gas interface and in this study is used to track the liquid/gas interface. The VOF formulation relies on a scalar indicator function between zero and unity to distinguish between two different fluids. For each additional phase that you add to your model, a variable is introduced that is the volume fraction of that phase in each cell of computational domain (f). Magnitude for the fraction of the phase is 0< f <1 in cells cut by the interface and $f=0$ or 1 away from it. A value of zero indicates the presence of one fluid and a value of unity indicates the second fluid. Since f is passively advected with the flow, it satisfies the advection equation

$$\frac{\partial f}{\partial t}+\left(\vec{V}.\nabla\right)f=0 \tag{1}$$

While tracking the interface, for all cells the summation of all volume fractions must satisfy following condition

$$\sum_{q=1}^{n} f=1 . \tag{2}$$

Youngs' proposed an algorithm for geometric reconstruction of the free surface and used it to solve the volume fraction equation [5].

A single momentum equation is solved throughout the domain, and the resulting velocity field is shared among the phases. The momentum equation, shown below, is dependent on the volume fractions of all phases through the properties ρ and μ :

$$\frac{\partial}{\partial t}(\rho\vec{v})+\nabla\cdot(\rho\vec{v}\vec{v})=-\nabla p+\nabla\cdot\left[\mu\left(\nabla\vec{v}+\nabla\vec{v}^{T}\right)\right]+\rho\vec{g}+\vec{F} \tag{3}$$

Depending upon the problem definition, additional scalar equations may be involved in solution process. In the case of turbulence quantities, a single set of transport equations is solved, and the turbulence variables (e.g., κ and ε) are shared by the phases throughout the field.

RESULTS AND DISCUSSION

Validation

In this section, a comparison is made between the results of simulations with those of the available data in the literature for a specific spray nozzle. To the best of our knowledge no experimental work is reported to investigate different parameters of a Y-type nozzle such as the average velocity of the steam and the fuel or the shape of fuel jet at the outlet of the nozzle orifice. Therefore, to validate the results of simulations, another atomizer is selected, a nozzle for which the experimental results or other numerical results are available in literature. The selected atomizer is a simplex swirl atomizer (or pressurized swirl atomizer) [6] which has a simple geometry with a multiphase condition similar to that of the Y-type atomizer. A schematic of the considered atomizer is shown in Figure 6. In this nozzle the liquid enters through the tangential inlet slots resulting in a strong swirling motion of the liquid in the swirl chamber. The liquid enters the exit orifice after flowing through a convergent section. The strong swirl velocity of the liquid is responsible for a thin liquid film close to the wall and a low pressure zone along the axis. This low pressure causes a back flow of air inside the atomizer which forms an air-cored vortex along the centerline of the chamber. The liquid leaves the orifice as a conical liquid sheet due to the centrifugal force. After leaving the atomizer, the annular liquid sheet becomes unstable and breaks up into a spray of droplets [6].

A 2D model of the simplex swirl atomizer [6] was created based on the dimensions given in Table 1. Important results for these kinds of atomizers, which are the thickness of the liquid film and the angle of the generated cone, are compared with those of Mandal [6] in Table ٢ . Close agreement between the two results validate the numerical model and its underlying assumptions. The configuration of the free surface of the fuel is displayed in Figure 7. As observed, a thin conical liquid film (red color) is formed at the exit of the nozzle. The path lines in the simplex nozzle atomizer is shown in Figure 8 where the fuel circulation before entering the nozzle throat, the air circulation at the nozzle exit behind the conical film, and the air back flow in the centerline of the nozzle are observed.

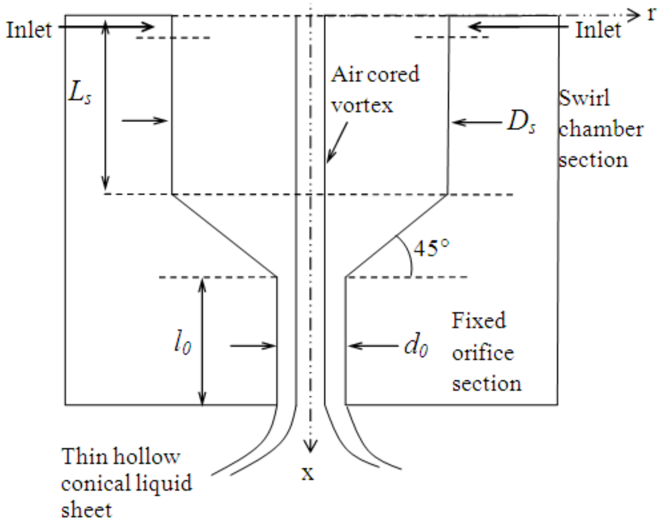

Figure 6. Schematic of a pressure swirl atomizer used for model validation [6]

Table ۱ . Geometric values of the 2D simplex atomizer of Figure 6.

Inlet pressure (MPa)	Inlet Area (mm^2)	d_o , l_o (mm)	D_s , L_s (mm)
0.5	0.054	0.3 , 0.3	0.9 , 0.45

Table ۲ . Comparison of liquid film thickness and cone angle from the model with those of Mandal [6]

Thickness of fuel	0.0405 (mm)
Thickness of fuel in the Ref. [6]	0.0435 (mm)
Cone angle	35.4°
Cone angle in the Ref. [6]	37°

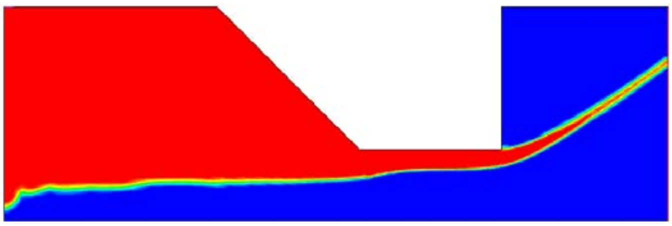

Figure 7. Configuration of the free surface of the liquid (red) and gas(blue) in the simplex swirl atomizer

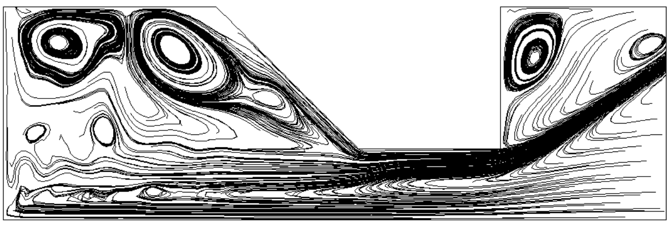

Figure 8. Calculated path lines in the simplex atomizer

Simulation results

The computational domain was discretized using an unstructured mesh which had approximately 250,000 cells. To reduce the required computational time of the simulation, the entrance region in both steam and fuel channels are discretized with a coarser mesh as shown in Figure 9. As seen in the figure, a fine mesh was selected for the atomization zone because of the expected higher velocities of steam in that region.

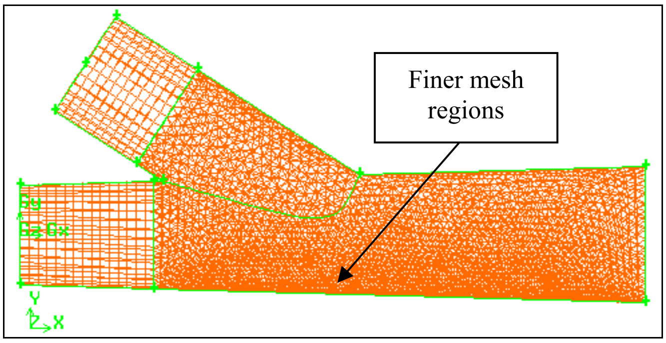

Figure 9. The computational grid for numerical simulation

For one atomization orifice of the nozzle in the nominal operating condition of the boiler (i.e., 100% of the boiler capacity), the necessary inlet condition and the physical properties of the steam and fuel are known by the powerplant documents and are as follows:

Mass flow rate of fuel	0.11423 kg/s
Mass flow rate of steam	2.28×10^{-3} kg/s
Inlet pressure of fuel	21 atm
Inlet pressure of steam	6 atm
Inlet temperature of fuel	120°C
Inlet temperature of steam	220°C
Density of fuel	0.007805 kg/m.s
Density of steam	1.68×10^{-5} kg/m.s
Viscosity of fuel	2.712 kg/m^3
Viscosity of steam	892 kg/m^3
Surface tension	0.02 N/m

All the simulations in this study are conducted by the FLUENT commercial program. Both the steam and fuel are considered to be incompressible and due to a high Reynolds number the flow is turbulent. The simulations of the Y-type atomizer are conducted by an explicit VOF and a geometric reconstruction interpolation (Geo-Reconstruct) scheme. The modeling results show the details of the fuel and steam flow inside the nozzle including velocity distribution (Figure 11), and the shape of the fuel/gas interface (Figure 10). It was found that the fuel breakup to ligaments and its atomization to droplets do not occur inside the nozzle. Instead, a liquid jet with certain cross sectional shape leaves the orifice while surrounded by a high velocity steam (Figure 10). These results were expected because of a short distance (6.6 mm) available for the interaction of the steam and fuel streams. The same behavior of flow interaction in Y-jet atomizers has been observed experimentally as well [7]. The fuel jet breaks into ligaments shortly after entering the combustion chamber of the boiler because of shear stresses and fluctuations caused by the high velocity steam. The ligaments are then broken into a spray of small droplets of the fuel.

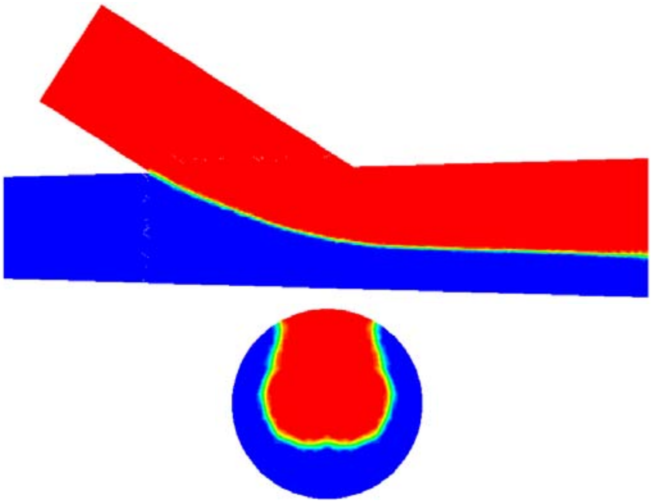

Figure 10. Longitudinal (up) and outlet cross sectional views (bottom) of the flow inside the nozzle. The red color shows the fuel and the blue color the steam.

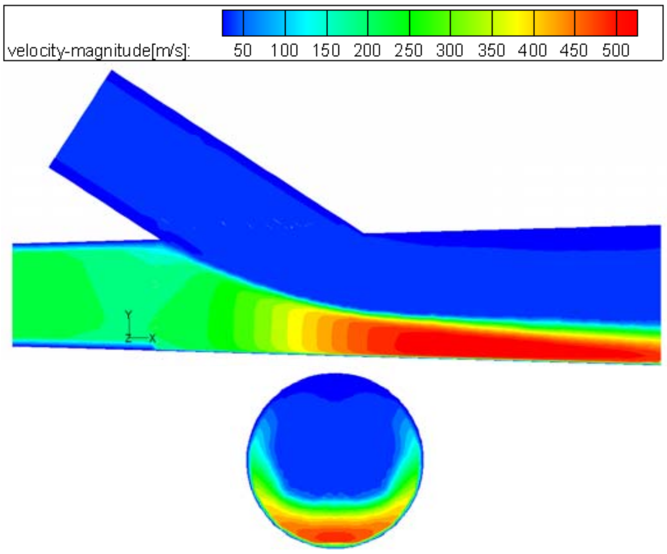

Figure 11. Velocity distribution along the nozzle orifice (up) and in the outlet cross sectioanl view (bottom) in m/s.

Numerically studying the drop size distribution outside the nozzle is a very time consuming and expensive process due to a large difference between the scale of the fuel jet and that of the spray drop size. Figure 12 shows an example of such a simulation for a simplified 2D atomizer where the primary breakup of the liquid jet to ligaments is displayed. For this 2D model, the number of cells was more than 50000 which reveals the fact that for a 3D simulation significantly more nodes need to be used. Therefore, continuing the simulation to get to the final drop size needs further costly computations not feasible in practical applications.

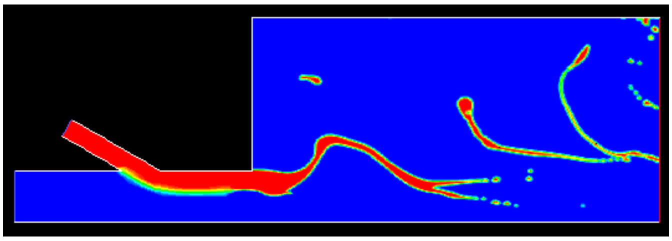

Figure 12. A 2D model of a simplified atomizer showing the atomization of fuel outside the nozzle

As an alternative approach, it is common to use experimental/analytical correlations to determine the fuel drop size [8-10]. These correlations are usually based on the liquid and gas flow configurations at the nozzle exit and their physical properties. In this study, the numerical model provides the shape of the liquid jet, and the steam and fuel velocity distributions at the nozzle exit. These parameters are then correlated to the final drop size distribution using analytical/experimental correlations available in literature. A recently introduced correlation for jet atomization [8] is employed in this paper. This correlation, which corresponds to a circular liquid jet surrounded by a high velocity gas flow, determines the Sauter Mean Diameter (SMD) of the fuel drop size distribution. In this study, therefore, the shape of the fuel at the nozzle exit was approximated by a circular jet of the same cross sectional area. The employed correlation is as follows

$$\frac{SMD}{D_l} = C_1\left(1+m_r\right)\left(\frac{b_g}{D_l}\right)^{1/2}\left(\frac{\rho_l/\rho_g}{\mathrm{Re}_{bg}}\right)^{1/4}\frac{1}{\sqrt{We_{D_l}}}\times \left\{1+C_2\left(\frac{D_l}{b_g}\right)^{1/6}\left(\frac{\mathrm{Re}_{bg}}{\rho_l/\rho_g}\right)^{1/12} We_{D_l}^{1/6}Oh^{2/3}\right\} \tag{5}$$

in which D_l, ρ_l and ρ_g are liquid jet diameter, liquid jet density and gas density, respectively, and $b_g = \left(D_g - D_l\right)/2$ where D_g is the gas stream diameter. SMD is the sauter mean droplet diameter of fuel droplets defined as $SMD = \sum N_i d_i^3 / \sum N_i d_i^2$ in which N_i is the number of droplets per unit volume in size class i and d_i is the droplet diameter. Non-dimensional terms used in equation (5) are

$$\text{Mass flux ratio}: m_r = \frac{\rho_l U_{Liquid} A_l}{\rho_g U_{Gas} A_g}, \tag{6}$$

where A_l and A_g are the occupied area of the liquid and gas at the nozzle exit, respectively, and U_{Liquid} and U_{Gas} are the average velocity of liquid and gas,

$$\text{Reynolds number}: \mathrm{Re}_{bg} = \frac{U_{Gas} b_g}{\upsilon_g}, \tag{7}$$

$$\text{Ohnesorge number}: Oh = \frac{\mu_l}{\sqrt{\rho_l \sigma D_l}}, \quad (8)$$

$$\text{Weber number}: We_{D_l} = \frac{\rho_g (U_{Gas} - U_c)^2 D_l}{\sigma} \quad (9)$$

where U_c is defined as

$$U_c = \frac{\sqrt{\rho_l} U_{Liquid} + \sqrt{\rho_g} U_{Gas}}{\sqrt{\rho_l} + \sqrt{\rho_g}} \quad (10)$$

The values for both coefficients C_1 and C_2 in equation (5) are determined by experimental results available for SMD of the nozzle in different operating conditions [8]. C_2 depends on the viscosity and surface tension of liquid jet and it has been shown [8] that a value of one for this constant for different liquids (Newtonian and non-Newtonian) results in a good prediction of the SMD. C_1 depends on the gas nozzle geometry in general, and on the contraction ratio of the nozzle in particular. This is because for a given nozzle size, the gas boundary layer thickness at the nozzle exit depends strongly on the contraction ratio [8]. To determine this constant, at least one experimental data for the SMD of the fuel must be available in an operational condition of the nozzle. In other studies available in literature, the magnitude of C_1 is considered to be one [8] and 0.58 [9] based on experimental data, respectively (in Ref. [9], a preliminary version of the presented correlation, Eq. 5, is used). As there was no experimental data for the nozzle considered in this research, we used the results of another study in which the nominal condition of the combustion process inside the boiler was modeled using CFD simulations. That numerical work showed that for a fuel spray size of 60 μm for SMD, the calculated amount of temperature and NOx at the boiler exit agreed well with the measured values available in the powerplant documents (the difference being less than 10%). Therefore, a value of 60 μm for the nominal SMD of the nozzle was used to obtain a value of 0.165 for the constant C1. Having obtained all terms of equation (5), the effect of different parameters such as fluids (steam and fuel) properties and velocities on the SMD can now be investigated. It should be mentioned that the numerical simulation provides the values required in equation (5) namely the average velocity of steam and fuel and the approximate diameter of fuel jet at the nozzle exit.

Increasing the steam to fuel momentum ratio will increase the shear stresses exerted to the fuel jet which leads to a smaller SMD. This reduction of momentum is correlated to either increasing the velocity of the steam or decreasing the fuel velocity. Figure 13 shows the effect of the mass flow rate of steam on the SMD. The mass flow rate of steam in this figure is for the entire nozzles in the boiler. The boiler has nine nozzles with ten orifices for each. Increasing the mass flow rate of steam will increase its velocity and consequently will increase the momentum ratio which leads to a decrease of the SMD. Figure 14 shows the effect of the total mass flow rate of fuel in boiler on the SMD. Reducing the fuel mass flow rate will decrease the fuel velocity resulting in a smaller SMD.

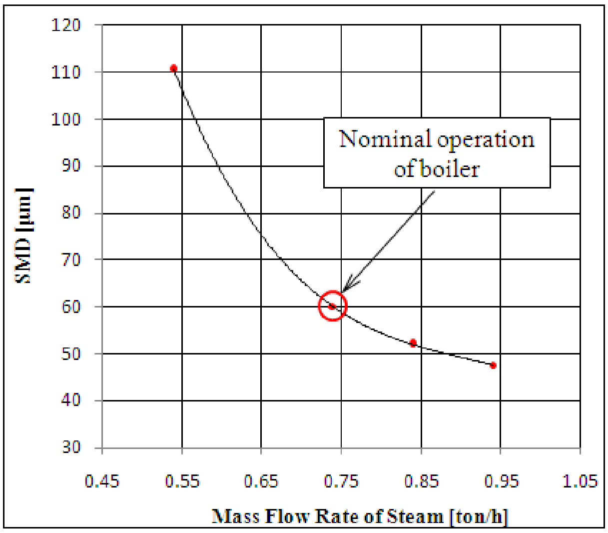

Figure 13. Variation of SMD vs. mass flow rate of steam

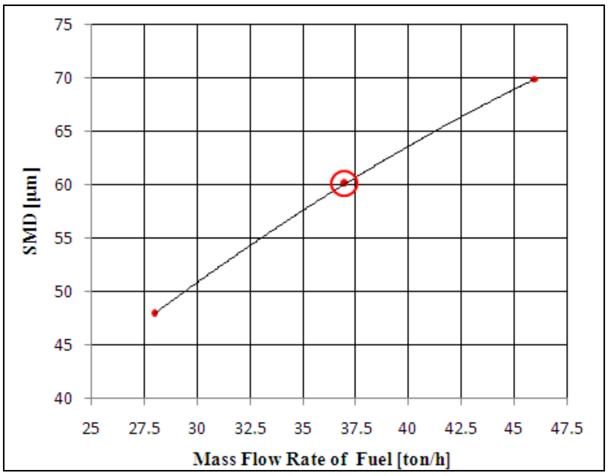

Figure 14. Variation of SMD vs. mass flow rate of fuel

The fuel physical properties namely viscosity and surface tension affect the fuel jet stability against the fluctuations and shear stresses exerted on the fuel jet by the surrounding turbulent flow. That in turn will change the magnitude of SMD. An important parameter that affects the fuel viscosity is the preheating temperature of the injected fuel to the nozzle. As the variation of fuel viscosity against temperature was available in the powerplant charts, the preheating influence on the SMD can also be investigated as shown in Figure 15. Raising the preheating temperature reduces the viscosity and, therefore, the SMD of the fuel spray at the nozzle exit is reduced. The variation of SMD versus surface tension of the injected fuel is shown in Figure 16. As seen from the figure, increasing the fuel surface tension leads to a bigger drop size in the resulting spray.

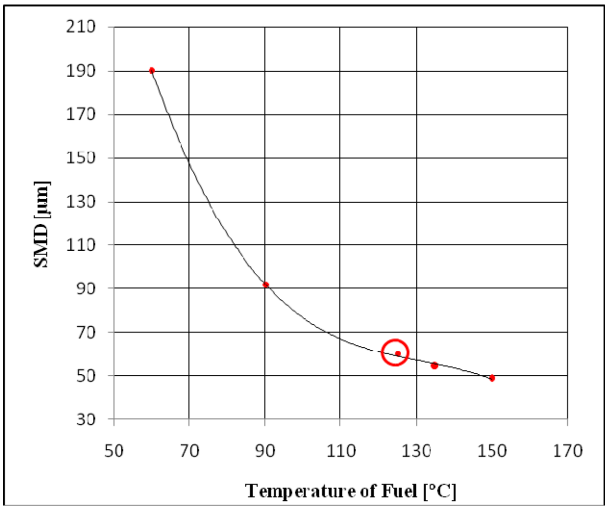

Figure 15. Variation of SMD vs. temperature of injected fuel

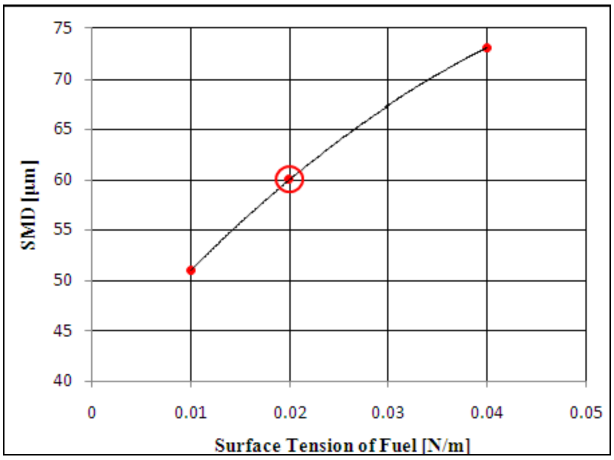

Figure 16. Variation of SMD vs. surface tension

CONCLUSION

In this study, the liquid fuel atomization in the injector nozzle of the combustion chamber of a powerplant boiler is numerically simulated by a commercial software (FLUENT). The simulation results show that the atomization of fuel does not occur inside the nozzle. Instead, the fuel leaves the orifice with a certain shape that can be approximated by a circular fuel jet surrounded by a high velocity gas resulting in the atomization of fuel shortly after exiting the atomizer. Because of a large difference in the scale of the fuel jet at the nozzle exit compared to that of the final drop size which leads to time consuming and expensive numerical modeling process, experimental/analytical correlations available in literature are used to obtain the final SMD of the spray. The numerical simulation provides the values required for the correlations namely the average velocity of steam and fuel and the approximate diameter of fuel jet at the nozzle exit. The results of this study show that the spray drop size can be reduced by employing one or combination of these techniques: increasing the amount fuel preheat before the nozzle inlet, increasing the mass flow rate of the atomizing steam, and reducing the fuel surface tension.

ACKNOWLEDGEMENTS

This research was supported by the Khorasan Regional Electric Company of Mashhad, Iran (www.krec.ir); their support is gratefully acknowledged. The authors also would like to thank Dr. B. Motakef of Ferdowsi University of Mashhad for generating the 3D CAD model of the considered nozzle.

REFERENCES

1. Lefebvre, A. H., 1989, Atomization and Sprays, Hemisphere.
2. US patent NO. 0128805, Twin fluid atomizer.
3. US patent NO. 3831843, Method of fuel atomization and a fuel atomizer nozzle therefore.
4. Atomization and Sprays, vol. 16, pp. 127-145, 2006, Experimental characterization of industrial twin fluid atomizers, Félix Barreras, Antonio Lozano, Jorge Barroso, and Eduardo Lincheta
5. In Numerical Methods for Fluid Dynamics, Academic Press, 1982, Time-Dependent Multi-Material Flow with Large Fluid Distortion, D.L., Youngs.
6. Master of Science Degree, 2007, University of Cincinnati, Computational Modeling of Non Newtonian Fluid Flow in Simplex Atomizers, Anirban Mandal.
7. Atomization and Sprays, vol. 6, pp. 193-209, 1996, Study of atomization mechanism of gas liquid mixtures flowing through Y-jet atomizers, Si Hong Song, Sang Yong Lee.
8. International Journal of Multiphase Flow, vol. 34, pp. 161–175, 2008, Atomization of viscous and non-newtonian liquids by a coaxial, high-speed gas jet. Experiments and droplet size modeling, A. Aliseda, E.J. Hopfinger, J.C. Lasheras, D.M. Kremer, A. Berchielli, E.K. Connolly
9. Journal of Fluid Mechanics, vol. 497, pp. 405–434, 2003, Initial breakup of a small-diameter liquid jet by a high-speed gas stream, C. M. Varga, J. C. Lasheras, E. J. Hopfinger
10. International Journal of Multiphase Flow, vol. 32, pp. 807–822, 2006, Breakup and atomization characteristics of mono-dispersed diesel droplets in a cross-flow air stream, Sung Wook Park, Sayop Kim, Chang Sik Lee