International Journal of Modern Studies in Mechanical Engineering (IJMSME)
Volume 3, Issue 4, 2017, PP 27-32
ISSN 2454-9711 (Online)
DOI: http://dx.doi.org/10.20431/2454-9711.0304003
www.arcjournals.org

Numerical Investigation on Transversely Liquid Jet in Air Flow

Bahram Jalili[1], Fathollah ommi[1*], Salman Nourazar[2]

[1]*Department of Mechanical engineering, Tarbait Modares University, Tehran, Iran*
[2]*Department of Mechanical engineering, Amirkabir University of Technology, Tehran, Iran*

Corresponding Author:** ***Fathollah ommi, *Department of Mechanical engineering, Tarbait Modares University, Tehran, Iran*

Abstract: *Cross-flow injection of liquid jet into gaseous flowis considered as a practical method for fuel injection, because of its high evaporation rate and proper atomization. The suitable fuel-to-air ratio could be achieved by tolerating the momentum ratio, injection angle, and, also using kinds of swirl injector. By considering the characteristics mentioned above, one could consider this flow as a proper way to achieve the ideal mixture of air and fuel. Designing with respect to injection data, could result to decrease in pollutants, increased efficiency, and decreased fuel consumption. In this paper, the effects of different geometries on trajectory line and break-up point are studied numerically, using a high fidelity CFD method. Results show that despite the circular nuzzle, that the break-up point could be considered almost constant along the wind direction, by changing the nuzzle geometry, the break-up point moves along both cross-wind and along-wind directions. The results obtained by numerical method are in good agreement with previously published experimental and numerical data.*

Keywords: *crossflow, Numerical Investigation, elliptical nozzle, spray trajectory*

1. Introduction

Due to adequate atomization and high vaporizing rate, Cross Jet injection is one of the most advanced fuel injection systems. Furthermore, in order to achieve a desirable fuel to air ratio it is possible to use momentum ratio changes and adjustable injection angles or even swirl injector. All the aforementioned examples are a proof of high capability of this procedure in achieving desirable fuel to air ratio. In the end, all of these reduce the environmental pollution and fuel consumption and increase the ignition efficiency. So far, many analytical, experimental and numerical approaches were used to study microminiaturization characteristics of a jet liquid Perpendicular on gas flow. Each one of these approaches are focused on a specific segment of jet characteristics and cross-flow and multiple equation were developed to predict crossover flow characteristics. Crossover flows nature is reported to be extremely unsteady. This unsteadiness happen because of the boundary layers close to the wall and the turbulence in the flow. Most of the past studies were focused on jet path [6-2] and depth of penetration [10-7] and jet lateral spread [11]. Woo et al. [2] used a force balance in cross air direction between aero dynamical acceleration Force and after force on jet pillar to simulate jet movement path.

Marzbali et al. [12] theoretically studied the changes at the incoming liquid jet pillar path to the cross over gas flow. They analytically studied liquid jet path close to nozzle for momentum below 100 and weber number more than 100 and offered an analytical equation for jet path. Riyan et al. [13] offered a new numerical approach including 2 separate sub model. Gas Cross flow is simulated by commercial code CFX to predict local pressure and gas pillar around jet pillar and on the other hand, liquid jet was changed by VOF which is being linked with CFX using duplicate method. By comparing the mean path and empirical results, they showed that results are completely overlapped. In this study, with concentrate to numerical study and VOF method, we tried to analyze effective factors on jet liquid path and penetration depth of jet liquid in cross air flow and to find out the effect of nozzle geometry on liquid jet path and penetration in cross air flow.

2. VOF Method

This approach is also known as Volume of fluid or (VOF) and is more common than other approaches. In this approach, the color function (F) for one of the fluids is one and for the other is equal to zero. Using this function it is possible to define the physical characteristics of fluid at any point in the range of solution. If we consider ϕ as a physical characteristic like density, viscosity etc…, for this property, we can use the following equation:

$$\phi = \phi_2 + F(\phi_1 - \phi_2) \tag{1}$$

Where ϕ is intended fluid. Mathematical presentation of the function F is as below:

$$F = \begin{cases} fluid\,1 & 1 \\ fluid\,2 & 0 \\ at\ interface & 0 < F < 1 \end{cases} \tag{2}$$

In other words, F is ratio of volume of fluid 1 to the total cell volume. So in cells where only fluid 1 exist, this function is equal to 1 and in cells with only fluid 2, it is equal to 0. It is clear that in a cell with both fluids, F is between 0 and 1. Figure 1describe this in a more clear way.

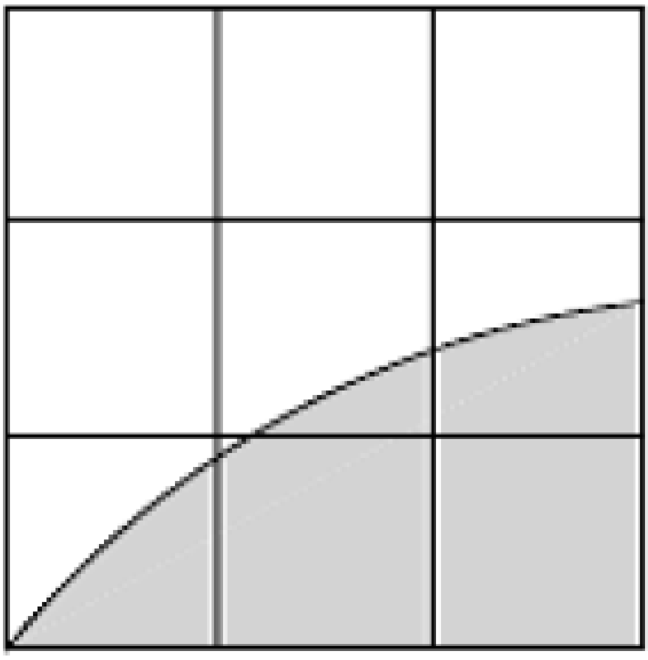

Figure1. *Common surface between two fluids*

In Figure 1, black space is fluid and white space is fluid 2. As you can see some cells have both white and black spaces which actually is the shared space. As you can see with shared space border in them, function F is something between0 and 1. As shown earlier the thinner this space, we will get more precise answers. In VOF approach some technics are used to get to this level of precision. In this approach, in cells at shared space, a geometrical approximation was selected for the space. Based on this, different approaches were made that we analyze their revolution.

3. Geometry and Jet Trajectory

Computational network structure is shown in Figure 2. This network was made by mapping the lower level with the upper level of channel (the level with water spray nozzle). The network has more than one million and two hundred thousand cells and each geometry has subtle differences. Network nodes have a higher density at spray space and around spray point which is because of simulation of border layer at spray point. At matching spaces, network cells are foursquare and unorganized and at Perpendicular direction to air flow they are organized hexagonal cells. Figure 3 shows flowing liquid jet. Liquid ligaments dimension increase is because of network cells dimension increase when getting further away from spray point (in sprays actual physics, getting away from spray point decreases droplet size).

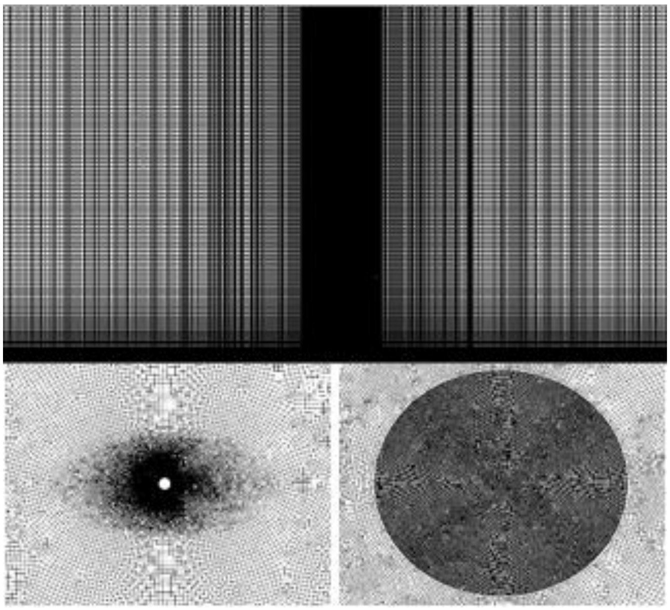

Figure2. *A view of generated grids*

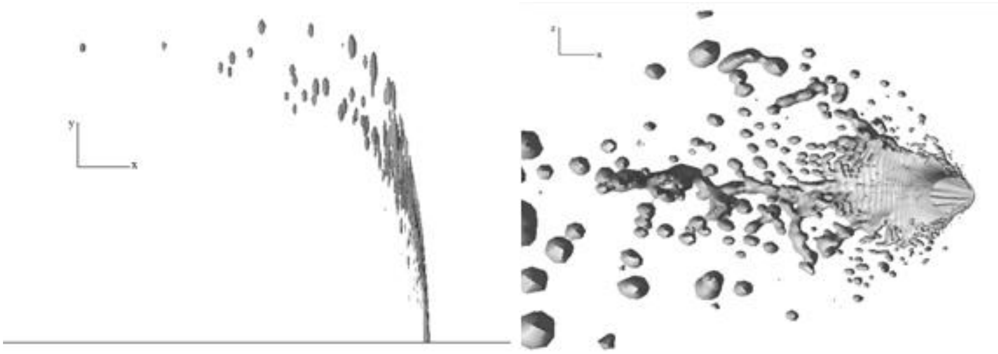

Figure3. *A view of spray trajectory*

4. Result And Discussion

The effect of liquid jet spray nozzle geometry on jet path and breaking point is show in Figure 4. As you can see there is a considerable correlation between numerical simulation results and empirical experiment. Using a nozzle with oval cross section and aspect ratio higher than 1, decrease the breaking length in flow direction and decreasing oval nozzle aspect ratio could increase break length. This increase and decrease in breaking length is because of drag force applied to fluid jet and cause the breaking length to increase. Compared to jet coming out of circular nozzle, Liquid fluid jet path is substantially different.

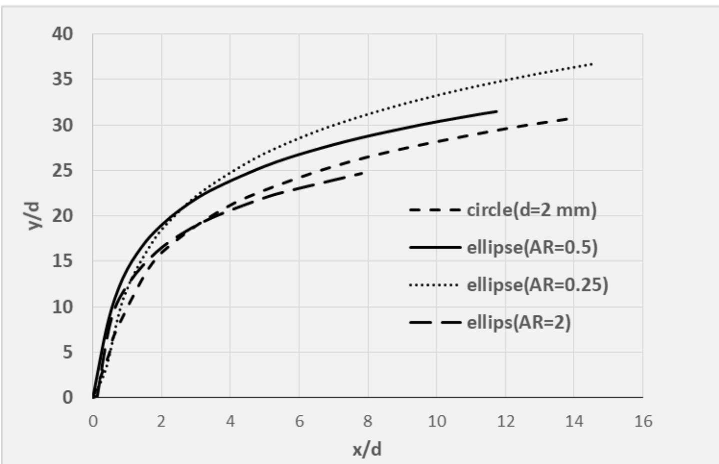

Figure4. *Effect of Nozzle geometry on spray trajectory*

As you can see in Figure 5, for momentum ratio 50 and Webber number 30, the results from oval nozzle geometry with aspect ratio of 0.5was compared with the results of Thawley et al. [14] work. As you can see, path and the breaking length is almost equal which shows the precision of this approach.

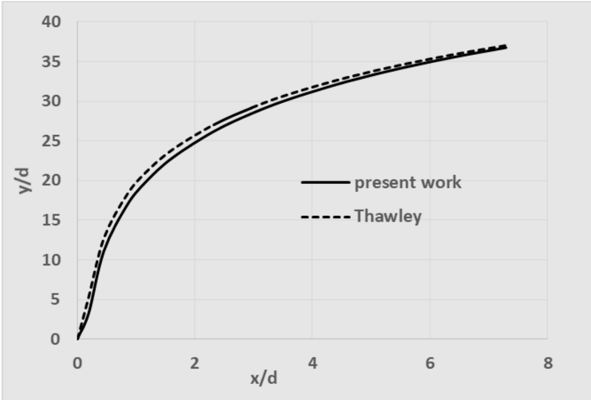

Figure5. *Comparison of obtained result and Thawley [14]*

Figure 6, shows the flow line around the liquid jet using of time average contours. In these lines decrease of liquid column thickness are shows that causing variation in jet trajectory. Jet trajectory with fitting curve shows in Figure 7. According to the Figure 7, we could obtain trajectory equation as a bellow equation:

$$\frac{y}{d} = 8.3\left(\frac{x}{d}\right)^{0.42} \tag{3}$$

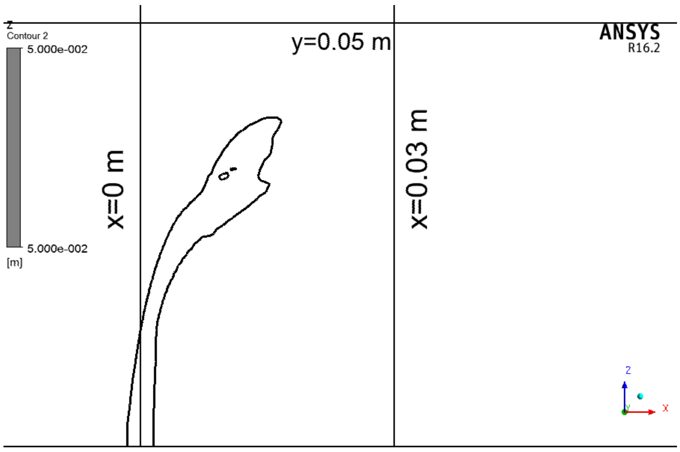

Figure6. *Boundary lines of liquid jet for obtaining the jet trajectory equation*

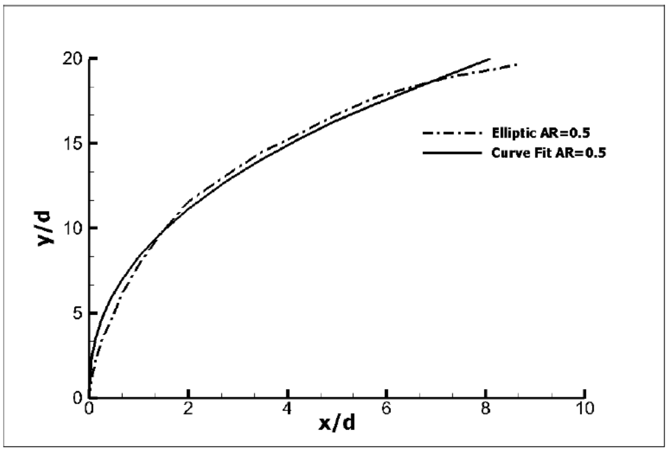

Figure7. *Liquid jet trajectory and curve fit*

5. CONCLUSION

In this study, we tried to analyze effective factors on fluid jet path and liquid jet penetration depth in cross airflow by focusing on numerical study and VOF. The nozzle geometry effect and liquid jet penetration in cross airflow were also analyzed. As it is apparent from the results, considering different geometry, the breaking length and breaking height changes but for circular nozzles the breaking length is constant. Also by comparing the resulting path with other works done by different researchers it is shown that there is a proper adaptation between them.

REFERENCES

[1] Ashgriz, Nasser. Handbook of Atomization and Sprays. New York: springer, 2011.

[2] Wu, P. K., K. A. Kirkendall, R. P. Fulle, and A. S. Nejad. "Breakup Processes of Liquid Jets in Subsonic Crossflows." Journal of Propulsion and Power 13, no. 1 (1997): 64-73.

[3] K. Mahesh, The interaction of jets with crossflow, *Annual Review of Fluid Mechanics*, Vol 45, pp. 379 407, 2013.

[4] Stenzler, J. N., Lee, J. G., Santavicca, D. A., & Lee, W. (2006). Penetration of liquid jets in a cross-flow. *Atomization and Sprays*, *16*(8).

[5] M. Guo, R. Kishi, B. Shi, Y. Ogata, K. Nishida, Effects of cross-flow on fuel spray injected by hole-type injector for direct injection gasoline engine, *Atomization and Sprays*, Vol. 25, pp. 81–98, 2015.

[6] Farvardin, E., et al. "Comparative study of biodiesel and diesel jets in gaseous crossflow." *Journal of Propulsion and Power* (2013).

[7] Song, Yoonho, Donghyun Hwang, and KyubokAhn. "Effect of Orifice Geometry on Spray Characteristics of Liquid Jet in Crossflow." In *55th AIAA Aerospace Sciences Meeting*, p. 1961. 2017.

[8] A. R. Karagozian, Transverse jets and their control, Progress in Energy and Combustion Science, Vol. 36, No. 5, pp. 531-553, 2010.

[9] Mashayek, A., A. Jafari, and N. Ashgriz. "Improved Model for the Penetration of Liquid Jets in Subsonic Cross flows." AIAA journal 46, no. 11 (2008): 2674-2686.

[10] Tian, Z. F., P. J. Witt, W. Yang, and M. P. Schwarz. "Numerical simulation and validation of gas-particle rectangular jets in crossflow." *Computers & Chemical Engineering* 35, no. 4 (2011): 595-605.

[11] Mashayek, A., and N. Ashgriz. "Model for deformation of drops and liquid jets in gaseous cross flows." *AIAA J* 47 (2009): 303-313.

[12] S. N. Marzbali., A. Dolatabadi., "Near-field Trajectory of Circular Liquid Jets Injected into Subsonic Gaseous Crossflow" 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition 4 – 7 January 2011, Orlando, Florida.

[13] CFD prediction of the trajectory of a liquid jet in a non-uniform air crossflow. Ryan, Matthew J. s.l.: Computers & fluids, 2006, Vol. 35, pp. 463-476.

[14] Thawley, S. M., et al. "Evaluation of column breakpoint and trajectory for a plain liquid jet injected into a cross flow." *Proceedings of the 21st Annual Conference on Liquid Atomization and Spray Systems*. 2008.

AUTHORS' BIOGRAPHY

BahramJalili, is a PhD of aerospace engineering from Tarbiat Modares University. He is a member of nonlinear Dynamic team at Noushirvany university of Babol. He is interested to nonlinear problems in mechanical and aerospace engineering. Also, he wor k on two-phase flow and cross flow trajectory and spray in ramjet and scramjet engine.

FathollahOmmi, is an associate professor at Tarbiat Modares University. He has been teaching in his field for more than 30 years. He is expert in aircraft engine and internal combustion engine. Supervised a number of graduate students to obtain a doctorate and master's degree and has a number of research published in the national and international journals.

Salman Nourazar, is an associate professor of Mechainical engineering at Amirkabir University of Techmology. He is an expert on nonlinear dynamic and flow instability. Supervised a number of graduate students to obtain a doctorate and master's degree and has a number of research published in the national and international journals.

Citation: *Fathollah ommi et al, (2017). Numerical Investigation on Transversely Liquid Jet in Air Flow, International Journal of Modern Studies in Mechanical Engineering (IJMSME), 3(4), pp.27-32, DOI: http://dx.doi.org/10.20431/2454-9711.0304003.*