

On the Performance of the Spray Fans used in HARAM for Improving Climate Conditions

A. Balabel, M. S. Youssef, M. Faizan and T. K. Kassem

Abstract— The present paper introduces the numerical simulations of two important jet atomization techniques for liquid jets either in co-flow or coaxial air stream. This work is motivated by the desire to improve the performance of the large spray fans used in HARAM piazzas for improving the climate conditions. Such large spray fans are fitted with a number of small nozzles distributed all over the rim of each fan. The optimum performance of the atomization process for the liquid jets issuing from the distributed nozzles is highly desired to obtain the so called fog cooling and to reduce the suspended air pollutants. The atomization process is numerically simulated using a numerical code developed previously by the authors. A new design for the implemented nozzles is introduced in order to increase the atomization efficiency in forms of fast breakup process of the liquid jet and the fine droplets obtained. In general, the numerical method applied showed a remarkable capability to predict the atomization process encountered in such large spray fans and it gave the possibility to extend the numerical simulation to include the nested effects of the distributed nozzles on the whole atomization process and the new atomization regimes encountered.

Index Terms— Atomization process, Co-flow/Coaxial liquid jet, droplet dynamics, Spray fans.

1 INTRODUCTION

ACCORDING to the general presidency for the affairs of the Al Masjid Al Haram in MAKKAH and Al Masjid Al Haram in MADINAH, there are about 500 large spray fans were distributed in the open piazzas of the both Harams for humidifying the air outside and to reduce the suspended air pollutants. This can lead to improve the climate conditions during the Hajj and Umrah seasons and to reduce the respiratory diseases during and after such seasons. By successive seasons, an improvement of the air quality levels and a reduction of the effect of the suspended air pollutants are mandatory. Due to the arid and hot atmospheric conditions and large exposed sandy deserts, the levels of particulate matter in MAKKAH are high and often exceed the air quality limits. Moreover, the particulate matter resulting from the expansion projects and the construction activities in both Harams can affect the air quality level.


According to the recommendations published in the previous symposiums for Hajj and Umrah researches, the number of the large spray fans should be increased to overcome such weather and pollution problems. This may require huge preparations and could impede the flexible movement of visitors. Alternative, as we suggest in the present research, is to increase the atomization efficiency of the existing spray fan system through either altering its operating conditions or by changing its geometrical design.

The general specifications of the large spray fans used in HARAM piazzas are shown in Table 1, however, no operating conditions could be obtained during performing our research. Consequently, our results will be presented in form of dimensionless numbers that can be applied for any group of operating parameters.

The physical domain of the large spray fan, shown above, can be illustrated in Fig. 1, where the axisymmetric liquid jet (internal flow) issuing from the nozzle is subjected to a co-flow air stream (outer flow) of infinite extent. The dimensions of the computational domain are given in [1], and it is used as a test case for verifying the numerical method adopted.

TABLE 1
SPECIFICATIONS OF THE LARGE SPRAY FANS USED IN HARAM PIAZZAS

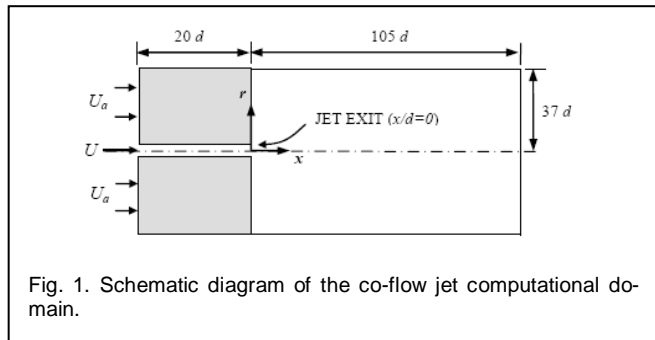
Parametr	Value
Diameter	80 cm
Width	38 cm
Weight	120 kg
No. of blades	7 blades
No. of nozzles	16 nozzles
Height of the fan from ground	3.5 m
Rotation angle of the fan	180°



The general specifications of the large spray fans used in HARAM piazzas are shown in Table 1, however, no operating conditions could be obtained during performing our research. Consequently, our results will be presented in form of dimensionless numbers that can be applied for any group of operating parameters.

The physical domain of the large spray fan, shown above, can be illustrated in Fig. 1, where the axisymmetric liquid jet (internal flow) issuing from the nozzle is subjected to a co-flow air stream (outer flow) of infinite extent. The dimensions of the computational domain are given in [1], and it is used as a test case for verifying the numerical method adopted.

- A. Balabel is currently a Professor in Mechanical Engineering Dept., Faculty of Engineering, Taif University, E-mail: ashrafbalabel@yahoo.com
- M.S.Youssef is currently an Assoc. prof. in Mechanical Engineering Dept., Faculty of Engineering, Taif University.
- M. Faizan is currently an Assist. prof. in Mechanical Engineering Dept., Faculty of Engineering, Taif University.
- T. Kassem is currently an Assoc. prof. in Mechanical Engineering Dept., Faculty of Engineering, Taif University.



Recently, the increased attention turned to studying the atomization of liquid jet in co-flow is motivated by its effect in different industrial and engineering applications, e.g., combustion stability [2], such as turbulent diffusion flames of combustion chambers [3], mixing mechanisms of turbulent jet [4]. Although of the importance of the co-flow in application of liquid atomization, the experimental and numerical investigations of such type of flow are relatively scarce [5].

Although several papers have recently reported experimental efforts to understand the physics of the liquid jet breakup, ligaments and droplets formation and its related dynamics in turbulent flow, however, experimental measurements and the observation of dense and small region with high spatial-temporal resolution in such applications have been difficult [6].

More recently, the carefully executed numerical simulations in such context can virtually replace experiments. In general, the numerical predictions of turbulent jet dynamics have been limited in accuracy partly by the development and the performance of three key elements, viz.: development of the computational algorithm, interface tracking methods, and turbulence models. In more particular, recently the numerical simulation of the coaxial liquid jet is of important interest due to its wide range applications and challenges. Phenomenological studies of coaxial liquid oxygen atomized by a fast, coaxial gaseous hydrogen jet under a broad variation of influencing parameters including injector design, inflow, and fluid conditions are performed in [7].

Recently, Spray formation was simulated in ANSYS CFX under a Lagrangian model. The primary breakup Blob model is used to handle atomization of the liquid while the secondary breakup TAB and ETAB models are evaluated for the subsequent breakup of the atomized droplets [8].

More recently, a Large Eddy Simulation is applied to predict the mixing and the intermittency of coaxial turbulent jet discharging into an unconfined domain [9]. In a coaxial shear injector element relevant to liquid propellant rocket engines, a numerical simulation for liquid water issued into nitrogen gas at elevated pressures is performed [10]. It can be concluded from such literature review that, the numerical simulation of coaxial liquid jets is also few and required further improvement and continuation.

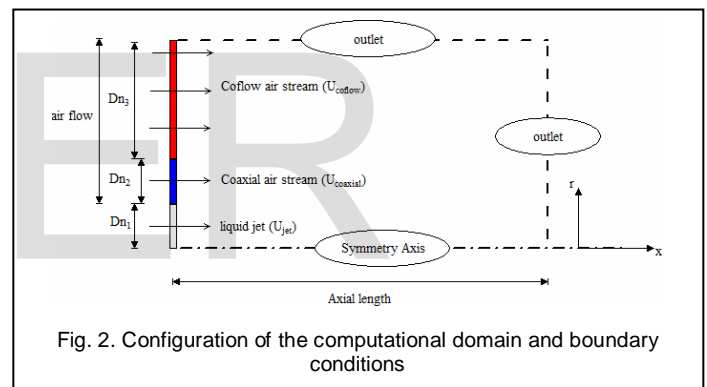
Consequently, in the present paper, the developed numerical method on the basis of the control volume approach is applied to predict the co-flow/coaxial jet dynamics. The complete system of the governing equations is solved using a de-

veloped computer program [11]. A comparison is made between the two jet orientation in terms of the atomization efficiency and characteristics.

2 MATHEMATICAL FORMULATION

The governing equations for 2D unsteady, axisymmetric, isothermal and incompressible turbulent two-phase flow are described in the present section. The system of the governing equations is based on the known RANS equations which stand for the Reynolds-averaged Navier Stokes equations. The standard two-equation STD $k-\epsilon$ turbulence model is applied for predicting the turbulence characteristics. The volume of fluid method is adopted for describing the topological changes of the liquid jet. Consequently, the associated boundary conditions and the numerical algorithms and models applied for solving the appropriate governing equations are also discussed.

The physical domain of the liquid jet to be solved numerically is shown in Fig. 2. The governing equations are solved numerically using the control volume approach.



2.1 Reynolds-Averaged Navier Stokes Equations

The Reynolds form of the continuity and momentum equations for turbulent two-phase flow, called here RANS equations, at each point of the flow field can be represented by the following equations after neglecting the body force:

$$\nabla \cdot (\rho \bar{u})|_{\alpha} = 0 \quad (1)$$

$$\frac{\partial(\rho \bar{u})}{\partial t} + \nabla \cdot (\rho \bar{u} \bar{u}) + \nabla p = \nabla \cdot (2\mu \hat{S} + \hat{\mathfrak{R}}_t)|_{\alpha} \quad (2)$$

where the subscript α takes the values 1 and 2 and denotes the properties corresponding to the liquid and gas phases, respectively. In the above system of equations, \bar{u} is the velocity vector, p is the pressure, ρ is the density, μ is the molecular viscosity, \hat{S} is the strain rate tensor and $\hat{\mathfrak{R}}_t$ is the turbulent stress tensor which are given as:

$$S_{ij} = 0.5 \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

$$\mathfrak{R}_{ij} = -\overline{\rho u'_i u'_j} = -\frac{2}{3} \rho k \delta_{ij} + 2\mu_t S_{ij} \quad (4)$$

where δ_{ij} is the Kronecker delta and $\overline{u'_i u'_j}$ are the average of the velocity fluctuations. The turbulent viscosity is defined as:

$$\mu_t = \rho C_\mu k^2 / \varepsilon \quad (5)$$

The turbulent kinetic energy k and its dissipation rate ε can be estimated by solving the following equations:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k \bar{u}) = \nabla \cdot (\mu + \mu_t / \Sigma_k) \nabla k + 2\mu_t \hat{S} \hat{S} - \rho \varepsilon \quad (6)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \varepsilon \bar{u}) = \nabla \cdot (\mu + \mu_t / \Sigma_\varepsilon) \nabla \varepsilon + \quad (7)$$

$$(2C_{1\varepsilon} \mu_t \hat{S} \hat{S} - C_{2\varepsilon} \rho \varepsilon) \varepsilon / k$$

The coefficients for the so-called STD $k-\varepsilon$ turbulence model are given as follows [Launder and Spalding (1974)]:

$$C_\mu = 0.09, \Sigma_k = 1, \Sigma_\varepsilon = 1.3, C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92$$

2.2 Level Set Function

The level set method is a class of capturing method where a smooth phase function ϕ , is defined over the complete computational domain. The level set function at any given point is taken as the signed normal distance from the interface separates the two fluids with positive on one side (i.e. $\phi > 0$), and negative on the other (i.e. $\phi < 0$). Consequently, the interface is implicitly captured as the zero level set of the level set function, as shown in figure 1. This level set function is updated with the computed velocity field and thus propagating the interface.

The update of the level set function with time can be determined by solving the following transport equation:

$$\frac{\partial \phi}{\partial t} + \bar{u} \cdot \nabla \phi = 0 \quad (8)$$

where \bar{u} is the velocity vector. Since the interface is captured implicitly, the level set algorithm is capable of capturing the intrinsic geometrical properties of highly complicated interfaces in a quite natural way. Consequently, the normal vector and the curvature of the interface can be defined as:

$$\bar{n} = \frac{\nabla \phi}{|\nabla \phi|}, \quad \kappa = \nabla \cdot \bar{n} \quad (9)$$

The time-stepping procedure for the level set equation is based on the second-order Runge-Kutta method. An important step in the solution algorithm of the level set function

is to maintain the level set function as a distance function within the two fluids at all times, especially near the interface region, i.e. the Eikonal equation; $|\nabla \phi| = 1$ should be satisfied in the computational domain. This can be achieved each time step by applying the re-initialization algorithm described in [Sussman, Smereka and Osher (1994)] for a specified small number of iterations.

Since the development of the level set method for incompressible two-phase viscous flow [Sussman, Smereka and Osher (1994)], a large number of articles on the subject have been published and several types of problems have been tackled with this method; see for instance the cited review [Sethian and Smereka (2003)]. However, the implementation of the level set method in predicting the moving interfaces under turbulent characteristics is indeed very scarce.

The present algorithm adopted here for solving the above system of the governing equations is based on the one extensively described in [11-15]. For more details about the numerical method and the assigned boundary conditions between the two phases one can consider the previous publications [11- 15].

3 VALIDATION (TURBULENT ROUND JET)

In this section, the accuracy and efficiency of flow computations with two-equation statistical turbulence models of $k-\varepsilon$ type, which are widely used in engineering applications, are investigated. The investigations involve the self-similar flow structures. The following investigations concern the accuracy, numerical efficiency and robustness of the numerical method described previously as well as the evaluation of the quality of the turbulence model adopted. That can be assessed by comparing the obtained numerical results with the previous numerical or experimental data.

Turbulent jets which constitute a subclass of free shear layer are of great practical interest. Many relevant technical applications are based upon jets, e.g. rocket engines or gas turbine. Moreover, jets do provide an interesting flow field for basic research because jet's flow field exhibits a self-similar structure. However, jets pose a difficult problem for $k-\varepsilon$ models employing a single fixed set of constant, because, for example, the velocity decreases and the spreading rate of plane and round jets are completely different. Therefore, jets are widely used as a standard test case for turbulence modeling. The geometry and boundary conditions of the test case is shown in Fig. 3. The set of used data are given in [11].

In order to further validate the implementation of $k-\varepsilon$ model against the turbulent round jet problem, the numerical results are compared with the standard reference experimental research of [16], for different quantities. The comparison includes (a) the distribution of the axial velocity along the jet axis u_c , made dimensionless with the inlet velocity, (b) the axial velocity distribution in the radial direction at $x/D_n=75$, (c) the distribution of the dimensionless radial velocity in the radial direction, and (d) the turbulent kinetic energy distribution in the radial direction, made dimensionless with its value at the jet axis at $x/D_n=75$.

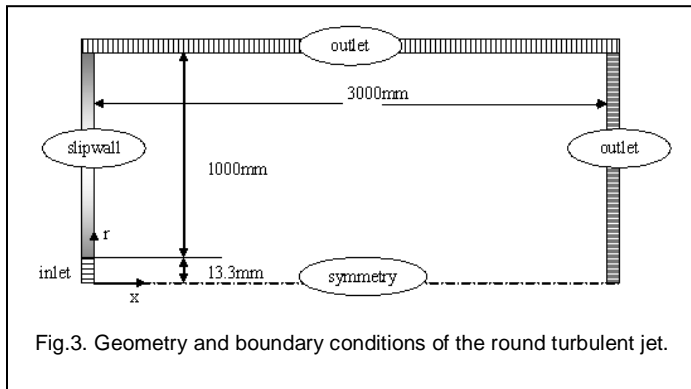


Fig.3. Geometry and boundary conditions of the round turbulent jet.

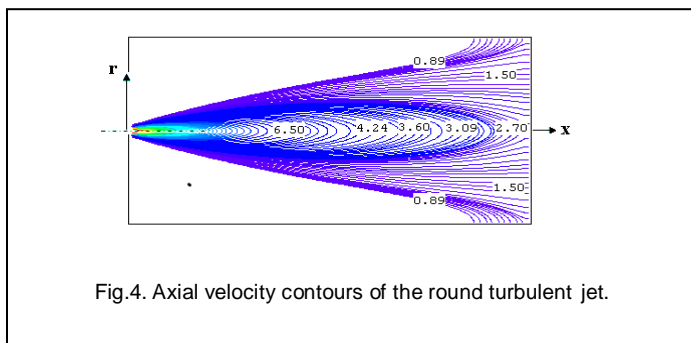


Fig.4. Axial velocity contours of the round turbulent jet.

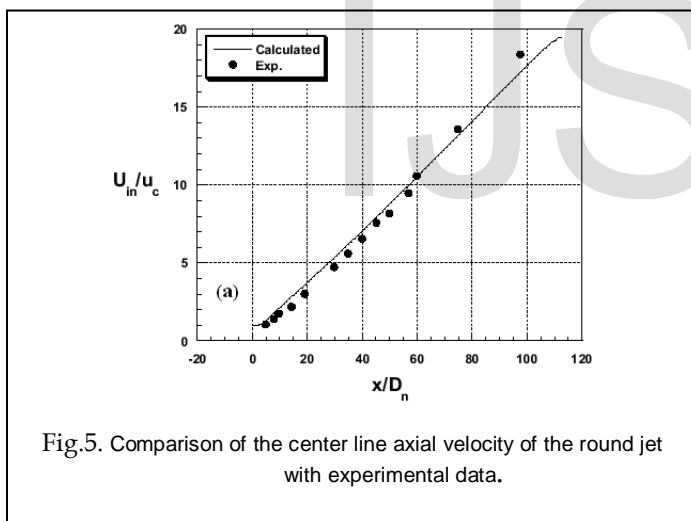


Fig.5. Comparison of the center line axial velocity of the round jet with experimental data.

The axial velocity graph for the axisymmetric jet is shown in Fig. 4. From the comparison shown in Figs. 5, 6, 7 and 8, the performance of the numerical method and the turbulence model adopted is clearly revealed. For the radial velocity, the area where the jet entrains the surrounding air is clearly marked with the region of the negative radial velocity. The match between experimental data and calculation results is relatively good. When drawing this comparison, one has to keep in mind that the $k-\epsilon$ model assumes local isotropy which, at least for this test case, is rather questionable. The measurements indicate a noticeable anisotropy that lies beyond the capabilities of the standard $k-\epsilon$ model. That can be considered in the future work regarding this problem by using non-linear $k-\epsilon$ models.

Yet, in summary, it can be stated that the used turbulence model performs well in the test cases considered and can be expected to produce reliable results in the jetting system configurations.

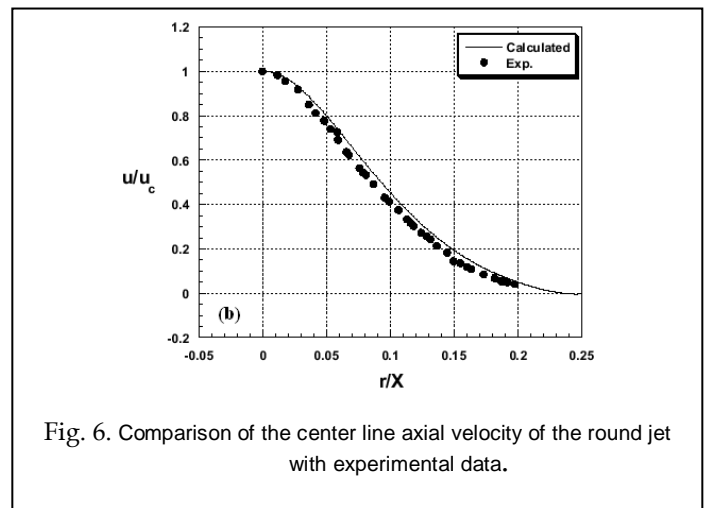


Fig. 6. Comparison of the center line axial velocity of the round jet with experimental data.

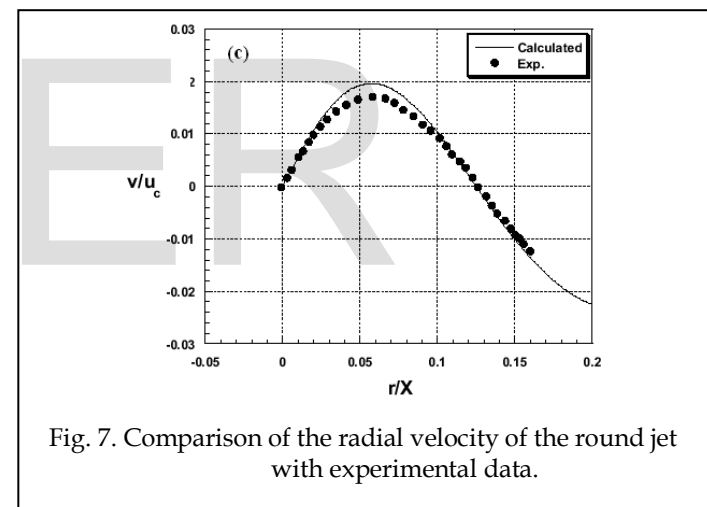


Fig. 7. Comparison of the radial velocity of the round jet with experimental data.

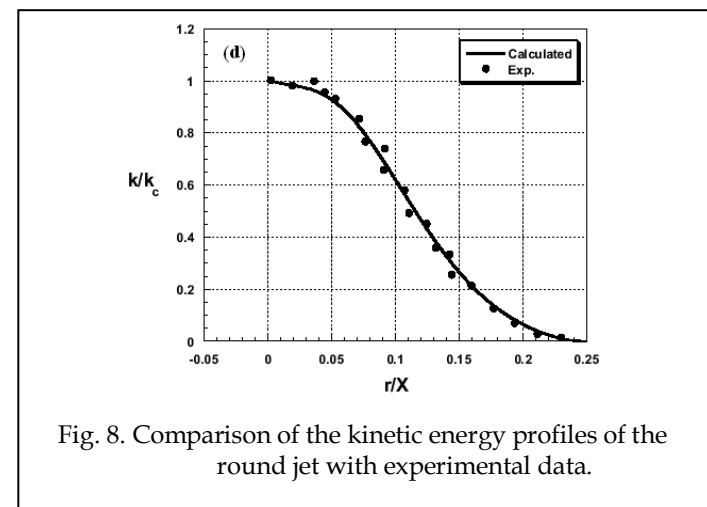


Fig. 8. Comparison of the kinetic energy profiles of the round jet with experimental data.

4 RESULTS AND DISCUSSION

In the present section, the numerical simulation of the axisymmetric jet emerging in air co-flow with different velocity ratio $RU=U_{co-flow}/U_{jet}$. The effect of the velocity ratio between two streams on the spreading rate of the jet is investigated. Three different velocity ratios are considered; namely, $RU=0.05, 0.1$ and 0.5 . For the direct conclusion, both fluids in jet and co-flow are considered as air. From the point of view of the simulation it makes no difference if the working fluid is considered as water or air, so long as the Reynolds number is matched.

4.1 Effect of co-flow velocity on jet decay

Figure 9 shows the axial velocity contours for the different co-flow/jet velocity ratio. It can be seen that, as the velocity ratio increases, the spreading rate of the jet is decreased. This result can also be indicated by plotting the centerline velocity decay, shown in Fig. 10, and the normalized centerline velocity decay, shown in Fig. 11.

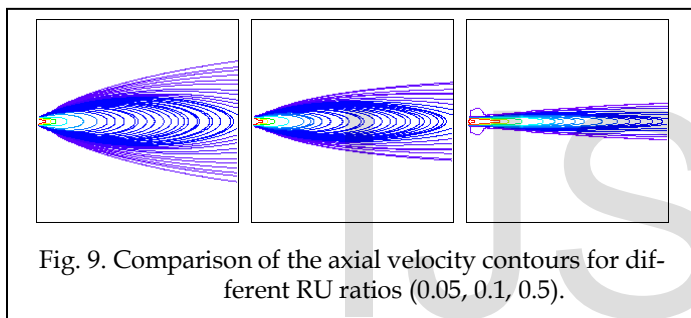


Fig. 9. Comparison of the axial velocity contours for different RU ratios (0.05, 0.1, 0.5).

In general, the important visible effect of the co-flow is the jet decay rate reduction, in comparison with the free jet case. Therefore, the jet without co-flow tends to mix more rapidly with the ambient air than the co-flowing jets.

4.2 SIMULATION OF AIR/WATER FLOWS

In the present section, we consider the real situation of the large spray fans used in HARAM piazzas. The input data of the numerical simulation, see Fig. 2, for the different three cases considered are given in Table 2 and the standard liquid/air properties are used in the present numerical simulation

TABLE 2

THE INITIAL AND INPUT DATA FOR NUMERICAL SIMULATION

	U_{jet}	$U_{co-axial}$	$U_{co-flow}$	D_{n1}	D_{n2}	D_{n3}	Axial length
Case I	50 m/s	0.0	150 m/s	0.05 m	0.0	0.1m	0.15 m
Case II	50 m/s	250 m/s	0.0	0.05 m	0.025 m	0.0	0.15 m
Case III	50 m/s	250 m/s	150 m/s	0.05 m	0.025 m	0.075m	0.15 m

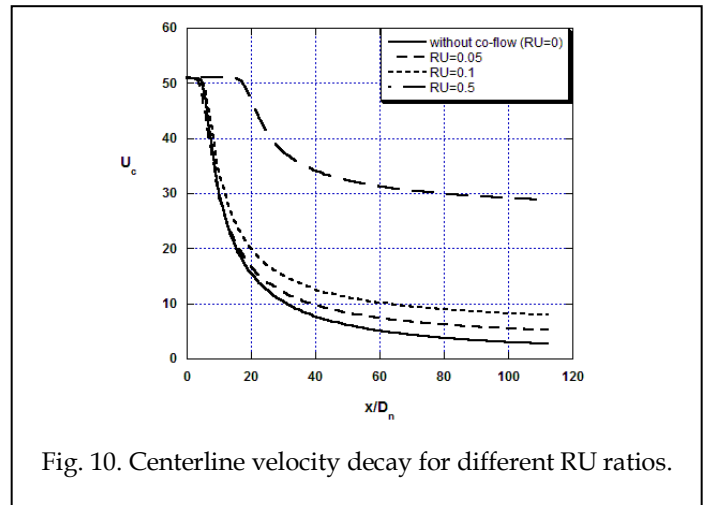


Fig. 10. Centerline velocity decay for different RU ratios.

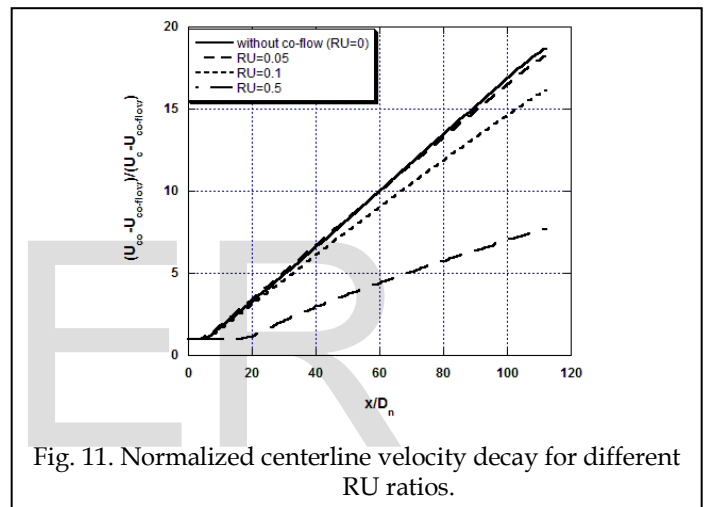
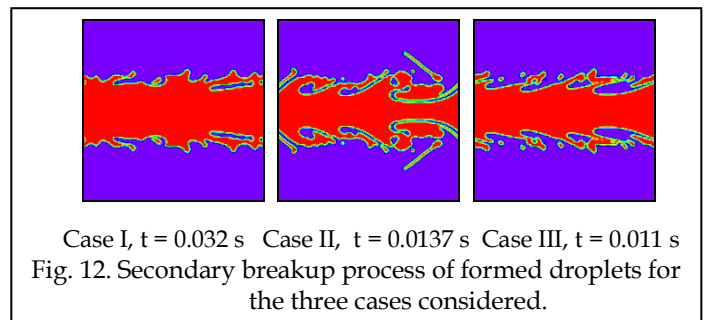


Fig. 11. Normalized centerline velocity decay for different RU ratios.

The numerical simulation is carried out until the secondary breakup occurs and the fine droplets are formed. Figure 12 shows the formation of the droplets from the breakup of the liquid ligaments, which known as the secondary breakup process. As can be seen from, the secondary process is dependent on the preceding processes of ligament formation and detachment. Consequently, it is predicted that the secondary breakup process is more efficient and faster in Case III than in both Case II and Case I. This leads to an efficient atomization process with preferred characteristics. Therefore, it is recommended to use the coaxial spray nozzle in addition to the air co-flow in the large spray fans used in Harams piazzas.



Case I, $t = 0.032$ s Case II, $t = 0.0137$ s Case III, $t = 0.011$ s
 Fig. 12. Secondary breakup process of formed droplets for the three cases considered.

5 CONCLUSION

The performance of the large spray fans used in HARAM piazzas for obtaining better climate conditions is investigated. The numerical method adopted is based on the solution of the Reynolds-Averaged Navier Stokes (RANS) equations for time-dependent, axisymmetric and incompressible two-phase flow in both phases separately and on regular and structured cell-centered collocated grids using the control volume approach. The turbulence characteristics are calculated using the standard $k-\epsilon$ turbulence model. The topological changes of the interface separating the two streams are predicted by applying the level set method.

The effects of the air/air co-flow streams are investigated firstly to determine the effects of the co-flow/jet velocity ratio on the jet decay and spreading rate. Furthermore, the air/liquid flows are used in the numerical simulation allowing the real dynamics to be predicted. The numerical results obtained confirmed that the coaxial spray nozzle should be used in the large spray fans in order to increase the atomization efficiency in terms of the fine droplet formed in the secondary atomization regime.

ACKNOWLEDGMENT

The present research is supported by Taif University, Saudi Arabia, under the **Project Contract No. 1-434-2430**. The Taif University is highly appreciated for the financial support of the project.

REFERENCES

- [1] E.J. Smith, J. Mi, G.J. Nathan and B.B. Dally, Preliminary Examination of a "Round Jet Initial Condition Anomaly" for the $k-\epsilon$ Turbulence Model, 15th Australasian Fluid Mechanics Conference, The University of Sydney, Sydney, Australia, 13-17 December 2004.
- [2] Werner J.A. Dahm, and Robert w. Dibble, Combustion Stability Limits of Co-flowing Turbulent Jet Diffusion Flames, AIM 26th Aerospace Sciences Meeting, January 11-14, 1988/Reno, Nevada.
- [3] Wang, F. and Li, P., Combustion Regimes of a jet diffusion flame in hot co-flow, Energy Fuels, Vol. 27 (6), pp. 3488-3498, 2013.
- [4] Dano, B., D. Kirk and Zha, G., Experimental investigation of jet mixing of a co-flow jet airfoil, 5th flow control conference, 28 June 2010, Chicago, Illinois.
- [5] M. Gazzah, N. Boughattas, H. Belmabrouk, R. Said, The dynamic field in turbulent round jet discharging into a co-flowing stream, Natural Science, Vol.2, No.6, 635-640 (2010)
- [6] Eggers, J. (1997), Nonlinear Dynamics and Breakup of Free-Surface Flow, *Revs. Modern Phys.*, 69, No. 3, pp. 865-929.
- [7] Mayer. W., (1994) Coaxial atomization of a round liquid jet in a high speed gas stream: A phenomenological study, *Experiments in Fluids* 16, 401-410.
- [8] FUNG, M. C., Inthavong, K., William Yang, W. and TU, J., (2009) External characteristics of spray atomisation from a nasal spray device, Seventh International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne.
- [9] Dinesh, K. R., Savill, A. Jenkins, K. and Kirkpatrick, M., (2010) A study of mixing and intermittency in a coaxial turbulent jet, *Fluid Dynamics Research*, 42 (2), 1-40.
- [10] Ibrahim, E., Kenny, J. and Walker, N., (2010) Atomization of coaxial liquid jets, *Applied Physical Research*, 2 (1), 3-18.
- [11] A. Balabel and W.A. El-Askary, "On the performance of linear and nonlinear $k-\epsilon$ turbulence models in various jet flow applications", *European Journal of Mechanics B/Fluids EJMFLU*, vol. 30, pp. 325-340, 2011.
- [12] A. Balabel, "Numerical Prediction of Droplet Dynamics in Turbulent Flow using the Level Set Method", *International Journal of Computational Fluid Dynamics IJCFD*, vol. 25, No. 5, pp. 239-253, 2011.
- [13] A. Balabel, "Numerical Modelling of Turbulence-Induced Interfacial Instability in Two-Phase Flow with moving Interface", *Applied Mathematical Modelling*, vol. 36 (8), pp. 3593-3611, 2012.
- [14] A. Balabel, "A Generalized Level Set-Navier Stokes Numerical Method for Predicting Thermo-Fluid Dynamics of Turbulent Free Surface", *Computer Modeling in Engineering & Sciences (CMES)*, vol.82, no.1, pp.29-54, 2012.
- [15] A. Balabel, Numerical Modelling of Turbulence Effects on Droplet Collision Dynamics using the Level Set Method, *Computer Modeling in Engineering & Sciences (CMES)*, Vol. 89, No. 4, pp. 283-301, 2012.
- [16] Wagnowski, I. and Fiedler, H. (1969). Some Measurements in the Self-Preserving Jet, *J. Fluid Mech.*, Vol. 38(3), 577-612.