World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:12, No:2, 2018

Numerical Simulation of Natural Gas Dispersion from Low Pressure Pipelines

Omid Adibi,融Nategheh Najafpour, Bijan Farhanieh, Hossein Afshin

***Abstract*—**Gas release from the pipelines is one of the main factors in the gas industry accidents. Released gas ejects from the pipeline as a free jet and in the growth process, the fuel gets mixed with the ambient air. Accordingly, an accidental spark will release the chemical energy of the mixture with an explosion. Gas explosion damages the equipment and endangers the life of staffs. So due to importance of safety in gas industries, prevision of accident can reduce the number of the casualties. In this paper, natural gas leakages from the low pressure pipelines are studied in two steps: 1) the simulation of mixing process and identification of flammable zones and 2) the simulation of wind effects on the mixing process. The numerical simulations were performed by using the finite volume method and the pressure-based algorithm. Also, for the grid generation the structured method was used. The results show that, in just 6.4 s after accident, released natural gas could penetrate to 40 m in vertical and 20 m in horizontal direction. Moreover, the results show that the wind speed is a key factor in dispersion process. In fact, the wind transports the flammable zones into the downstream. Hence, to improve the safety of the people and human property, it is preferable to construct gas facilities and buildings in the opposite side of prevailing wind direction.

***Keywords*—**Flammable zones, gas pipelines, numerical simulation, wind effects.

I. Introduction

Gas transporting pipelines are vital parts of worldwide countries for the operation of all economic and social activities. The functional defeat of these networks can have severe human and financial losses in numerous ways [1], [2]. In recent years, due to importance of the problem, lots of researches have been performed to improve safety of people and human property in the case of natural gas accidents. In these studies, besides considering accident preventions, lots of efforts were also prepared on the mitigation of accident consequences [3]-[6]. Natural gas accidents can be started with small leakage and then by combination of combustion triangle (fuel, oxygen and spark), get followed by devastating explosions [4].

One of the key points in prevention of gas accidents is mixture process of fuel and oxygen where lots of analytical, experimental and numerical studies have been performed in this field. Mixing process begins after pipeline failure and is followed by pollution problems or fire creation [7], [8]. In one of these analytical-numerical studies, Meysami et al. [9], by using “PHAST” (a commercial risk management package), provided a scheme to select the most appropriate conditions for gas dispersion modeling. This scheme approaches modeling was based on the worst-case scenario. Borujerdi and Rad [10] numerically studied transient turbulent gas flow in a ruptured pipe by using a combined finite element-finite volume method. In the simulations to predict the turbulent viscosity, for the near wall region and compressibility correction, a modified model with a two-layer equation was used. Results showed that, the released mass flow rate from the rupture area reaches 2.4 times of its initial value and then becomes constant. Liu et al. [11] determined the source strength and dispersion of CO_2 releases from high pressure pipelines using real gas equation of states. In this study, the results were compared with experimental data and the results of “PHAST”. The results indicated that PHAST can predict slightly better release flow rate but may considerably underpredict the dispersion concentration. Adibi et al. [12] numerically investigated heat and mass transfer through ruptures of high pressure gas reservoirs. In this study, numerical simulations were discretized based on the finite volume method, and flow variables were calculated using density-based algorithm. The results of parametric studies showed that the exhausted mass flow rate has direct relation with reservoir’s pressure. Also, the results illustrated that in the rupture area of high pressure tanks, chocking phenomena occur, and the structure of exhausted gas is similar to under-expanded free jets.

In this paper, following earlier studies, computational fluid dynamic methods are used to investigate the natural gas and air mixing process in ruptured pipelines. Besides, by performing parametric studies, the wind effects on mixing process and flammable zones are also explored. In the next section the model and methodology are introduced. Then, by analyzing concentration and flammability contours, the discussions on results are presented.

II. Model and Methodology

A. Problem Description

Flammables zones around gas pipelines are vital areas which should be protected from any probable spark (e.g. sparks from cables’ short circuit, sparks from workers lighter). Since, a tiny spark can release combustion energy and lead to terrible explosions. In this study, by simulating gas leakages through a 40 cm rupture, the mixing processes are investigated, and potential flammable zones around a pipeline

O. Adibi is PhD Candidate, N. Najafpour is PhD Candidate, and B. Farhanieh is Professor at School of Mechanical Engineering, Sharif University of Technology, Azadi Ave., Tehran, Iran (e-mail: oadibi@mech.sharif.edu, najafpour@mech.sharif.edu, bifa@sharif.edu).

H. Afshin is Associate Professor at School of Mechanical Engineering, Sharif University of Technology, Azadi Ave., Tehran, Iran (corresponding author, phone: +98-21-66165530; fax: +98-21-66000021; e-mail: afshin@sharif.edu).

are determined. In the all simulations, the average release rate of natural gas is 10.43 kg/s. Moreover, effects of wind on flammable zones are investigated by simulating different wind speeds of 0.1, 2.5, 5.0 and 10.0 m/s.

B. Governing Equations

In the numerical simulations, under these conditions, conservation of mass, momentum, energy equations for 2D, transient, compressible and turbulent flow were considered. Besides, ideal gas equation of state was used for calculation of density as a function of temperature and pressure. These equations are presented in detail in [12], [13].

C. Turbulence Modeling

In the numerical simulations, turbulence of flow was modeled by using Reynolds averaging method. In this method, the solution variables in the instantaneous momentum equations are decomposed into the mean and fluctuating components [14]:

$$\varphi = \overline{\varphi} + \varphi' \qquad (1)$$

where φ denotes a scalar variable (e.g. pressure, energy or species concentration). Also, over-bar and prime symbols stand for mean and fluctuating components, respectively.

In the numerical modeling, for reducing unknown variable, fluctuation values in Reynolds stress terms are related to mean values by using Boussinesq approach [14], [15]:

$$-\rho\overline{u'_i u'_j} = \mu_t\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t \frac{\partial \overline{u}_k}{\partial x_k}\right)\delta_{ij} \qquad (2)$$

In this equation, μ_t is referred as turbulence viscosity, ρ and u are fluid density and velocity. Then, for determination of turbulence viscosity, the k-epsilon RNG turbulent model was used. Detailed information about turbulent modeling can be found in [13]-[15].

D. Numerical Scheme

Numerical simulation of the problem is based on the finite volume method and pressure-velocity coupling was achieved using SIMPLE scheme. Also, second order upwind method was used to spatial discretization of momentum, turbulence, pressure, density, energy and concentration equations.

E. Model, Grids, Initial and Boundary Conditions

In Fig. 1, the two-dimensional model of geometry and the used boundary conditions are shown. The geometry was modeled as 2D (80 m × 160 m) rectangle.

For grid generation, structured method with orthogonal tetrahedrons was used. Due to systematic cell location, the solver efficiency in structured method is more as compared to the unstructured method [16]. Grid sizes varied from 5 to 40 cm where fine grid sizes were applied to area near to rupture, and coarse grid sizes were applied to distant area from the rupture. Total number of cells in the simulation was 368,920. In Table I, detailed description of grid generations is presented.

In the numerical simulations, five different conditions of velocity inlet, pressure outlet, mass flow inlet, symmetry and no slip wall were used in the boundaries of the model. The boundary conditions were chosen in such way similar to real case rupture accidents. Since, velocity inlet profile was imposed such as atmospheric wind power law correlation according to (3) [17]:

$$u = u_r\left(\frac{y}{y_r}\right)^{\alpha} \qquad (3)$$

In this equation, y and α denote height and wind shear exponent, and subscription r stands for reference value.

Also, to impose atmospheric pressure profile, the pressure outlet boundary condition was defined as a function of height. The boundary conditions were applied in the positions that are demonstrated in Fig. 1. Detailed descriptions of each boundary condition are also presented in Table II. It should be noted that to diminish incorrect effects, boundaries were chosen in far distances to rupture point.

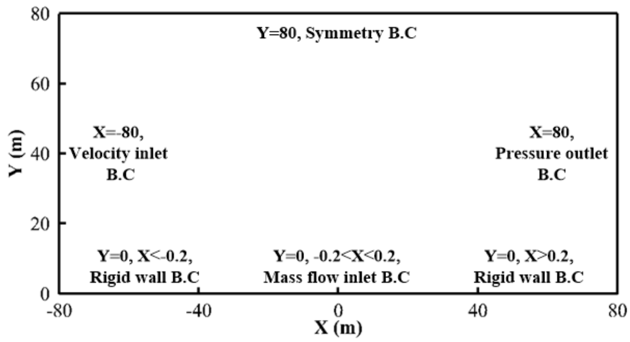

Fig. 1 Two-dimensional model and used boundary conditions

TABLE I
DETAILED DESCRIPTION FOR GRID GENERATION

Number of			Grid Sizes	
Cells	Faces	Nodes	Smallest	Largest
368,920	741,408	372,489	5 cm	40 cm

TABLE II
DETAILED DESCRIPTION FOR USED BOUNDARY CONDITIONS

Type	Imposed Position	DESCRIPTION
Velocity inlet	X=-80 m	Atmospheric velocity profile at the boundary & T=300 K.
Mass flow inlet	Y=0, -0.2 m<X<0.2 m	$\dot{m}$ =10.43 kg/s & T=300 K.
Pressure outlet	X=80 m	Atmospheric pressure profile the boundary.
Rigid wall	Y=0, X<-0.2m & X>0.2m	No-slip condition for velocity & adiabatic condition for temperature.
Symmetry	Y=80 m	Normal gradients of variable equal to zero.

III. RESULTS AND DISCUSSIONS

For examination of simulated atmospheric conditions, static pressure contours along with path lines are presented in Fig. 2. This figure is the results of simulation case with wind speed of

10 m/s. Results show that static pressure reduces with altitude, and horizontal streamlines demonstrate a steady air flow from left to right. Also, in Fig. 3 wind velocity profiles at left boundary are plotted versus altitude for four different reference wind speeds of 0.1, 2.5, 5.0, and 10.0 m/s. It should be noted that reference altitude in all cases was y_r=8 m. These results indicate that proper settings were applied in the boundaries and results are in accordance with atmospheric flows.

For investigation of natural gas and air mixing process in a stable atmospheric condition (reference wind speed=0.1m/s), the natural gas volume fraction contours at six different times (t=0.2, 0.4, 0.8, 1.6, 3.2, 6.4 s) after accident are presented in Fig. 4. The results show that after accident (the rupture time), natural gas immediately releases to the ambient and it reaches to 40 m altitudes in just 6.4 second. Also, it is clear that a circle with about 20 m radius is influenced with released natural gas.

For an unstable atmospheric condition with reference wind speed of 10 m/s, natural gas volume fraction contours at 6 above mentioned times are showed in In Fig. 5. In contrary with Fig. 4, the results indicate that natural gas penetrates in horizontal direction more than vertical direction. In the other words, an unstable atmospheric condition swipes the released gas in the wind direction.

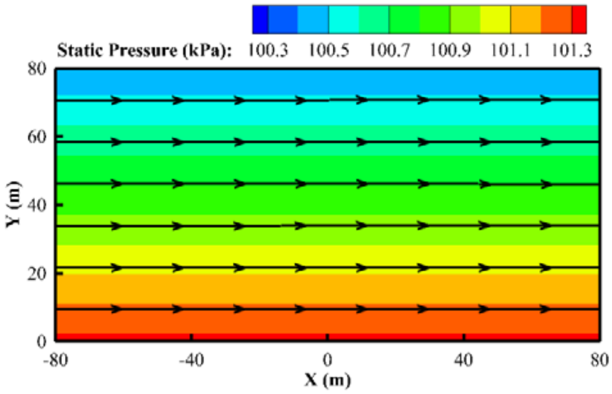

Fig. 2 Static pressure contours along with path lines

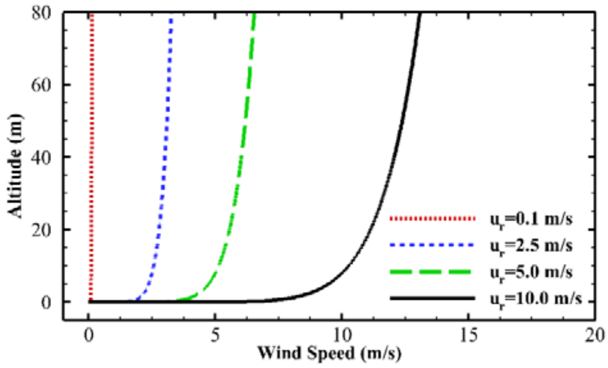

Fig. 3 Wind velocity profile versus altitude at left boundary

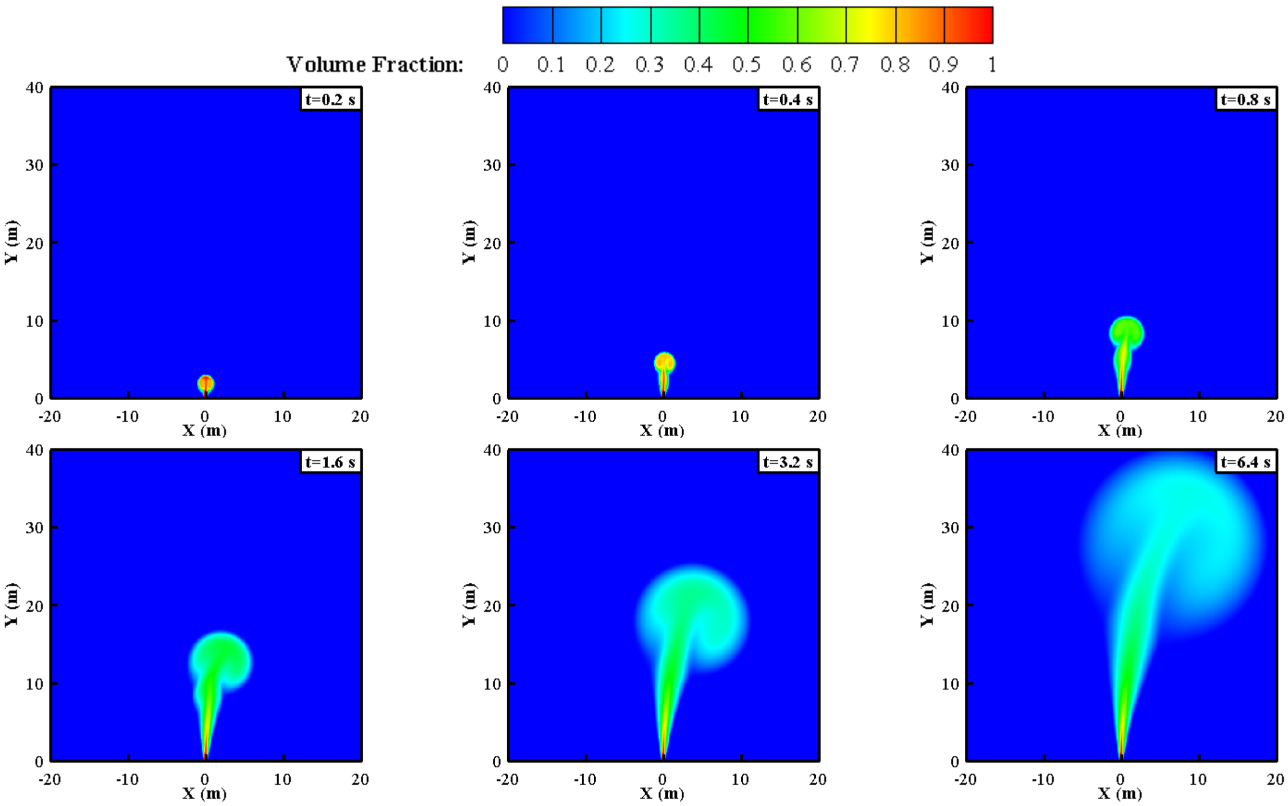

Fig. 4 Natural gas mole fraction contours at six different times after accident- wind speed=0.1 m/s

It should be noted that the flammability limits for this mixture is 0.05 to 0.15 volume fraction of natural gas in air [4]. So, the areas with lower or higher concentrations have not necessity conditions for combustion process. Accordingly, for determination of safety distances around ruptured points, illustration of flammability areas will be more useful. In Fig. 6, the flammability zones for four different wind speeds at 6.4 s after accident are presented. In this figure, areas which have volumetric natural gas concentration in the range of 0.05-0.15 are shown with red color. The results indicate that for wind speed of 0.1, 2.5, 5.0, 10.0 m/s, the safety distance around the ruptured point are approximately 19, 25, 39, and 68 m, respectively. Also, the penetration height for stable case (u_r=0.1 m/s) and windy cases (u_r=2.5, 5.0 and 10.0 m/s) are about 40 and 16 m, respectively. It should be mentioned that most of gas industry accidents have been started in lower altitudes [4], [7]. In the other words, potential sparks from sources such as lighters of workers, cables' short circuit are in areas near to ground. Since, due to smaller radius of flammability zones near ground, probabilities of explosion in stable days are less than windy days.

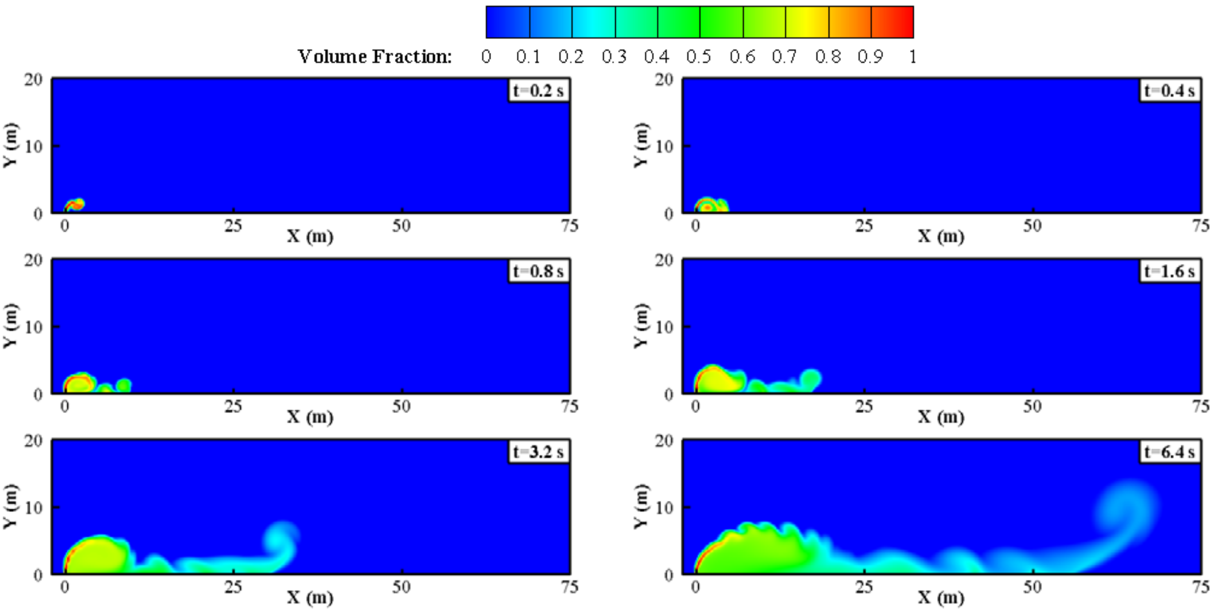

Fig. 5 Natural gas mole fraction contours at six different times after accident- wind speed=10 m/s

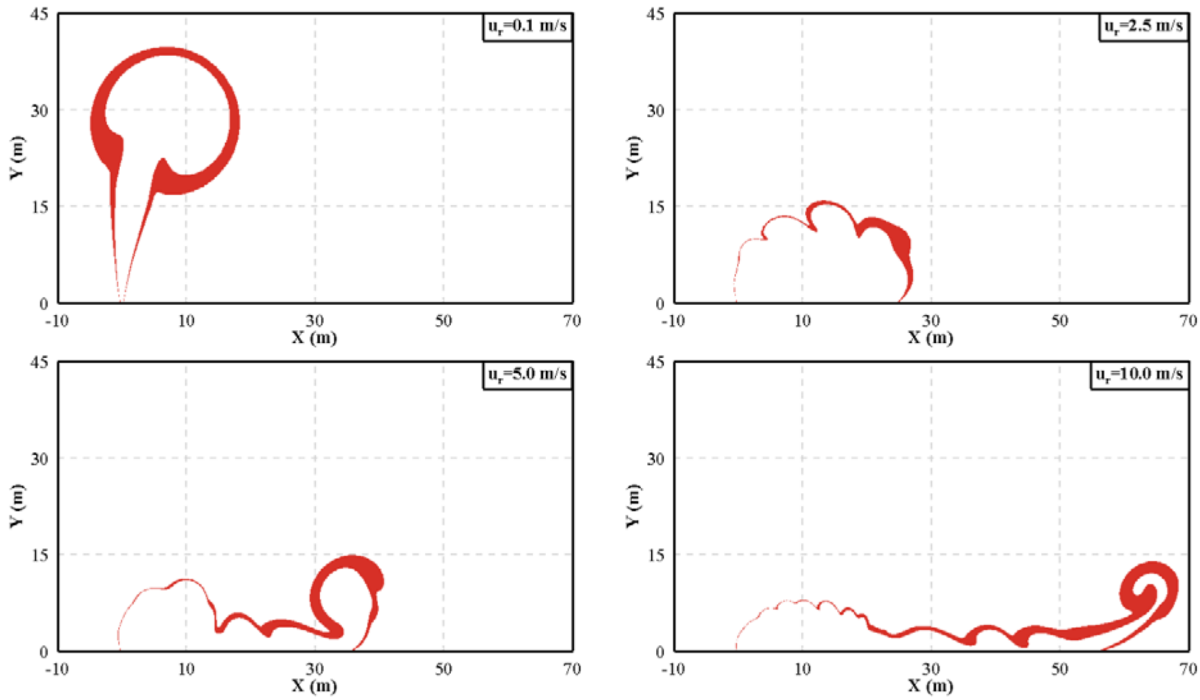

Fig. 6 Flammability areas around ruptured pipeline for four different wind speeds of 0, 2.5, 5, 10 m/s- 6.4 s after accident

IV. Conclusion

The main aim of this study was determination of flammability zones around ruptured low pressure natural gas pipelines in different atmospheric wind speeds. Release of 10.43 kg/s natural gas through a 40 cm rupture was simulated as a 2D, transient, compressible and turbulent flow. The simulations were based on finite volume method and k-epsilon RNG turbulent model. To consider the problem according to real gas pipeline rupture accidents, atmospheric wind velocity and pressure profiles were applied at the boundaries of the model. The results showed that, in a stable condition (wind speed=0.1 m/s), released natural gas penetrates to 40 m altitudes and 20 m in horizontal plane in just 6.4 s after accident. In the contrary, in an unstable case (wind speed=10.0 m/s), the natural gas just diffuses to lower altitudes (maximum 15 m in height) and mostly penetrates in horizontal direction (about 75 m) in just 6.4 s. Due to potential spark sources in areas near to ground, probabilities of explosion in windy days are more than stable days. Also, results suggest that to improve safety of people and human property, construct gas facilities and buildings in opposite side of prevailing wind direction.

Acknowledgment

This work has been supported by the National Iranian Gas Company under grant no. 194025.

References

[1] A. Golara, A. Esmaeily, "Quantification and enhancement of the resilience of infrastructure networks". *Journal of Pipeline Systems Engineering and Practice*, vol. 8, no. 1, pp. 1-10.

[2] A. Golara, H. Bonyad, and H. Omidvar, "Forecasting Iran's natural gas production, consumption", *Pipeline & Gas Journal*, vol. 242, no. 8, pp. 24-30, 2015.

[3] S. Sklavounos, F. Rigas, "Estimation of safety distances in the vicinity of fuel gas pipelines", *Journal of Loss Prevention in the Process Industries*, vol. 19, no. 1, pp. 24-31, (2006).

[4] H. Wilkening, D. Baraldi, "CFD modelling of accidental hydrogen release from pipelines", *International Journal of Hydrogen Energy*, vol. 32, no. 13, pp. 2206-2215, 2007.

[5] Q. Hou, W.Jiao, "Improved FDS analysis for the atmospheric impact of natural gas leakage and diffusion", *Journal of Computational Information Systems*, vol. 7, no. 13, pp.4702-4709, 2011.

[6] O. Adibi, A. Azadi, B. Farhanieh, and H. Afshin, "A parametric study on the effects of surface explosions on buried high pressure gas pipelines", *Engineering Solid Mechanics*, vol. 5, no. 4, pp. 225-244, 2017.

[7] R. Khaksarfard, M. R. Kameshki, and M. Paraschivoiu, "Numerical simulation of high pressure release and dispersion of hydrogen into air with real gas model", *Shock Waves*, vol. 20, no. 3, pp. 205-216, 2010.

[8] J. Choi, N. Hur, S. Kang, E. D. Lee, and K. B. Lee, "A CFD simulation of hydrogen dispersion for the hydrogen leakage from a fuel cell vehicle in an underground parking garage", *International Journal of Hydrogen Energy*, vol. 38, no. 19, pp. 8084-8091, 2013.

[9] H. Meysami, T. Ebadi, H. Zohdirad,and M. Minepur, "Worst-case identification of gas dispersion for gas detector mapping using dispersion modeling" *Journal of Loss Prevention in the Process Industries*, vol. 26, no. 6, pp. 1407-1414, 2013.

[10] A. N. Borujerdi, M. Z. Rad, "Numerical modeling of transient turbulent gas flow in a pipe following a rupture" *Scientia Iranica. Transaction B, Mechanical Engineering,* vol. 17, no. 2, pp. 108-120, 2010.

[11] X. Liu, A. Godbole, C. Lu, G. Michal, and P. Venton, "Source strength and dispersion of CO 2 releases from high-pressure pipelines: CFD model using real gas equation of state". *Applied Energy*, vol. 126, pp. 56-68, 2014.

[12] O. Adibi, B. Farhanieh, and H. Afshin, "Numerical study of heat and mass transfer in underexpanded sonic free jet", *International Journal of Heat and Technology*, vol. 35, no. 4, pp. 959-968, 2017.

[13] J. Wendt, "Computational fluid dynamics: an introduction", *Springer Science & Business Media*, 2008.

[14] S. B. Pope, "Turbulent flows", *Cambridge university press,* 2009.

[15] B. E. Launder, D. Spalding, "The numerical computation of turbulent flows", *Computer methods in applied mechanics and engineering*, vol. 3, no. 2, pp. 269-289, 1974.

[16] V. Venkatakrishnan, J. M. Dimitri, "Implicit solvers for unstructured meshes", *Journal of computational Physics*, vol. 105, no. 1, pp. 83-91, 1993.

[17] S. M. Tauseef, D. Rashtchian, S. A. Abbasi, "CFD-based simulation of dense gas dispersion in presence of obstacles" *Journal of Loss Prevention in the Process Industries,* vol. 24, pp. 371–376, 2011.