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

Determination of Safety Distance Around Gas Pipelines Using Numerical Methods

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

***Abstract*—**Energy transmission pipelines are one of the most vital parts of each country which several strict laws have been conducted to enhance the safety of these lines and their vicinity. One of these laws is the safety distance around high pressure gas pipelines. Safety distance refers to the minimum distance from the pipeline where people and equipment do not confront with serious damages. In the present study, safety distance around high pressure gas transmission pipelines were determined by using numerical methods. For this purpose, gas leakages from cracked pipeline and created jet fires were simulated as continuous ignition, three dimensional, unsteady and turbulent cases. Numerical simulations were based on finite volume method and turbulence of flow was considered using k-ω SST model. Also, the combustion of natural gas and air mixture was applied using the eddy dissipation method. The results show that, due to the high pressure difference between pipeline and environment, flow chocks in the cracked area and velocity of the exhausted gas reaches to sound speed. Also, analysis of the incident radiation results shows that safety distances around 42 inches high pressure natural gas pipeline based on 5 and 15 kW/m^2 criteria are 205 and 272 meters, respectively.

***Keywords*—**Gas pipelines, incident radiation, numerical simulation, safety distance.

I. Introduction

HIGH pressure gas pipelines are among the vital parts of each country which must resist against damaging effects of landslides, earthquakes, and explosions [1]. These pipelines are the most common tool for transferring natural gas from processing plants to customers. Russia, the United States, Canada, and Iran have the world longest natural gas pipelines distribution network. These systems are vital parts for the functioning of all economic and social activities of worldwide nations. The functional loss of these networks due to external forces can have severe economic impacts in numerous ways [2], [3]. Also, due to large amount of stored energy in high pressure gas transmission pipelines, smallest defect in equipment or workers negligence will lead to irreversible disasters. For example, in 2007, negligence of drilling workers led to leakage of natural gas which made jet fire and killed at least 15 people in Belgium [4]. Therefore, to reduce possible number of casualties and financial damages, a safe distance is assigned around gas pipelines where building constructions and people entry are allowed just in compliance with certain conditions.

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).

For determination of safety distance around high pressure pipelines several methods were introduced by researchers. The simplest model that is called “point source model” assumes that source of radiant heat is located at the center of the flame. In this model, heat flux decreases with the square of the distance. The point source model predicts incident heat flux with good approximations for far distances but in the areas near to the jet fire, it breaks down [5], [6]. In an improved model, a hypothetical line is assumed as source term of radiant heat and is referred as “*line source model*”. In this model, the total heat flux is calculated by integrating the flux in the source line [7]. Another model that is called “*titled cylinder model*” can predict incidents heat flux with better approximations in near and far away distances to the jet fire. In this model, the flame is replaced with cylinder where heat flux is radiated from its surface [8].

There are also numerous number of computer models which were developed in the field of safety analysis. WHAZAN is one of them where can predict leakage rate, chemical behavior of gasses, fire event and hazard [9]. SAFEMODE is another one which is developed for the United States Coast Guard. This computer model can predict pressure wave and thermal radiant of fire explosions. These computer models comprise hazard and consequence formulas to calculate safety distances in simplified geometries [10]. Another computer model is FLACS which is used in different explosion scenarios. This model can calculate safety parameters such as: pressure, temperature and incident radiation as a function of time [11].

Nowadays, by developments in computer and computational fluid dynamic science, different engineering problems can be solved in actual condition. Since, the main aim of this paper is determination of the safety distance around high pressure gas transmission pipeline with real assumptions. In this paper, by using continuous ignition, unsteady and turbulent flow assumptions, gas leakages and generated jet fire were simulated in 3D.

II. Model and Methodology

A. Problem Description

Incident radiation is the key factor in determination of safety distance around high pressure natural gas pipelines. In this paper, by using three-dimensional numerical modeling, the incident radiation of jet fire which can be created from gas release through 42-inch natural gas pipeline is investigated.

B. Governing Equations

Three main governing equations of the problem are conservation of mass, momentum and energy which are presented in the following equations, respectively. These equations can be used for unsteady, compressible, and viscous flows [12].

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho u_i)+u_i\frac{\partial}{\partial x_i}(\rho u_j)=-\frac{\partial P}{\partial x_i}+\frac{\partial \tau_{ij}}{\partial x_i}+\rho g_i \quad (2)$$

$$\frac{\partial}{\partial t}(\rho e)+u_i\frac{\partial}{\partial x_i}(\rho e)=-\frac{\partial q_i}{\partial x_i}-P\frac{\partial u_i}{\partial x_i}-\tau_{ij}\frac{\partial u_j}{\partial x_i} \quad (3)$$

In (1), ρ and u are the fluid density and velocity. Also, in (2) and (3) P, τ and e are pressure, shear tension and internal energy, respectively.

C. Turbulence Modeling

In three-dimensional simulation of a jet fire, effects of turbulence should be considered. In this problem, due to large scale of geometry using DNS and LES methods will impose heavy computational costs. Therefore, turbulent flow was modeled by RANS equations. In this method, turbulence terms in momentum equations (Reynolds stresses) are get substituted with average velocities according (4). In this method, real scale models can be simulated with low computational costs and acceptable accuracy [13].

$$-\rho\overline{u'_i u'_j}=\mu_t\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial \bar{u}_k}{\partial x_k}\right)\delta_{ij} \quad (4)$$

In this equation, μ_t is referred as turbulence viscosity, u' is velocity fluctuation, and $\bar{u}$ is average velocity.

D. Numerical Scheme

Numerical simulation of the problem is based on the finite volume method. In this method, geometry is divided to small volumes and integral forms of governing equations are solved for each volume. In Table I, discretization methods for each equation are presented. The capability and accuracy of these methods in determination of jet flows was achieved in previous study [14].

TABLE I
DISCRETIZATION METHODS OF GOVERNING EQUATIONS

Equations	Discretization Methods	Order of Accuracy
Transient terms	Implicit	First Order
Convection terms	Upwind Scheme	Second Order
Pressure terms	PRESTO	Second Order
Turbulence equations	Upwind Scheme	Second Order

E. Three-Dimensional Model and Grid Generation

In Fig. 1, three-dimensional model of geometry and generated grids are shown. In this problem, grids were generated using structured method. In this method, model is covered by orthogonal hexahedrons which reduce computational cost and is proper for large scale and simple geometries. It is clear that grid size in the area near to the jet fire is smaller than other parts. Fine grids enhance predicted structure of jet fire and accordingly improve accuracy of results. Overall shape of geometry is a rectangular cube with dimensions equal to $600\,m\times600\,m\times400\,m$.

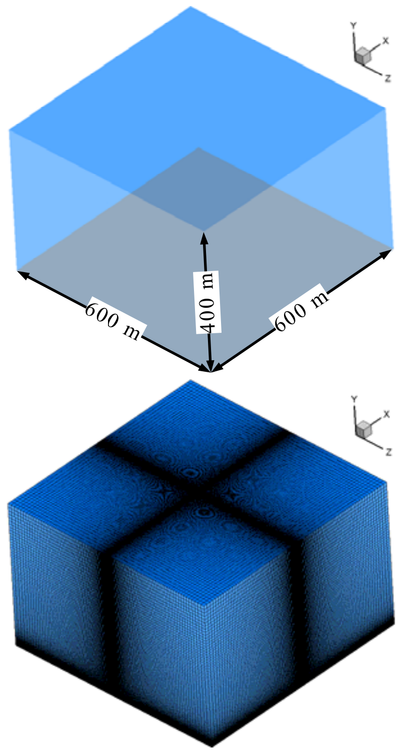

Fig. 1 Three-dimensional model and generated grid for model

III. RESULTS AND DISCUSSIONS

In Fig. 2, isosurfaces with T=1000 K at different times after accident are shown. In this figure, gray surface is ground, and green contours are surfaces with temperature equal to 1000 K. The results show how an unsteady jet fire growth in time. It is clear that due to huge volume of available natural gas which is releasing through pipeline, dimension of jet fire increases continuously. Also, due to turbulent nature of jet fire, there are oscillations in its growth progress.

In Fig. 3, temperature contours at x=0 plane are presented. Results show that the maximum temperature of air is reached to 2000 K. Also, results show that fresh air with low temperature (T=300 K) enters to the left and right side of fire and gets mixed with released natural gas. The fresh air and fuel mixture create a fire with huge heat load.

In Fig. 4, velocity contours at x=0 plane are presented. Results show that due to the high pressure difference between pipeline and environment, the choking phenomenon occurred in the cracked area and velocity of the exhausted gas is reached to sound speed

In Fig. 5, incident radiation of created jet fire is plotted in different times after accident. Results show that, due to high temperature of created fire, sudden increase in incident radiation is seen. This amount of radiation can make massive damages and will burn nearby facilities. Closer analysis of the results shows that safety distances around 42 inches high pressure pipeline based on 5 and 15 kW/m^2 criteria are 205 and 272 meters, respectively.

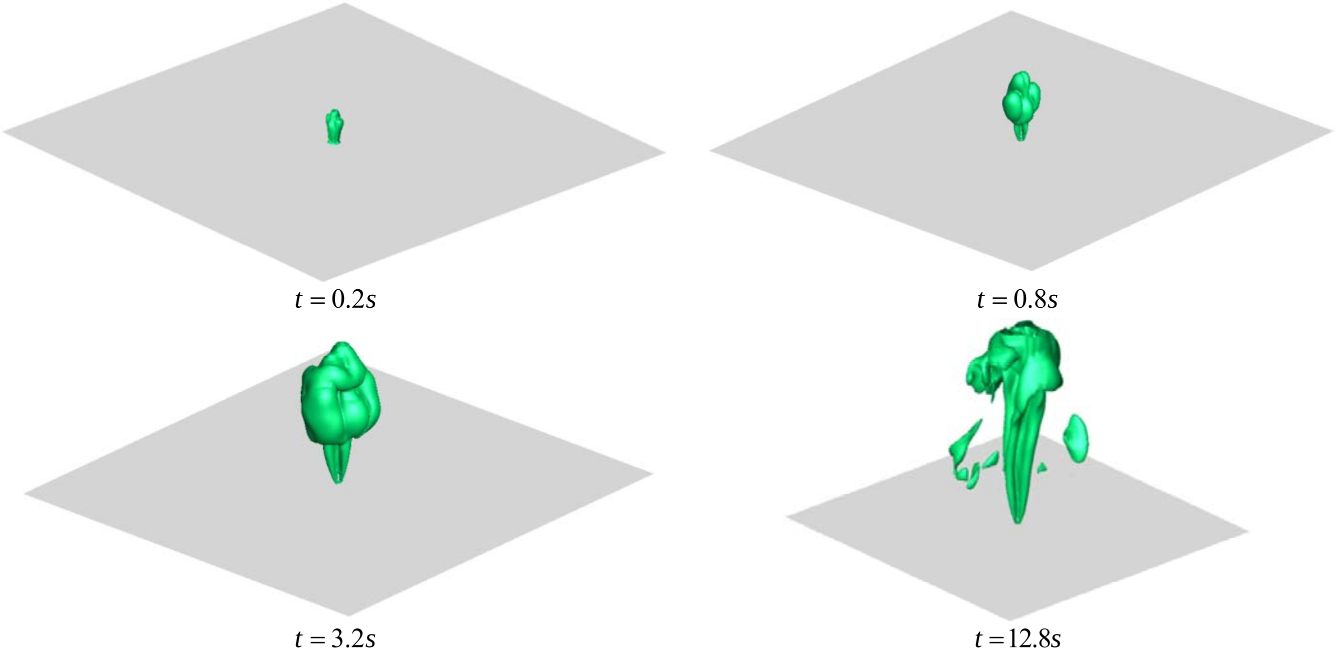

Fig. 2 Isosurfaces with T=1000 K at different times after accident

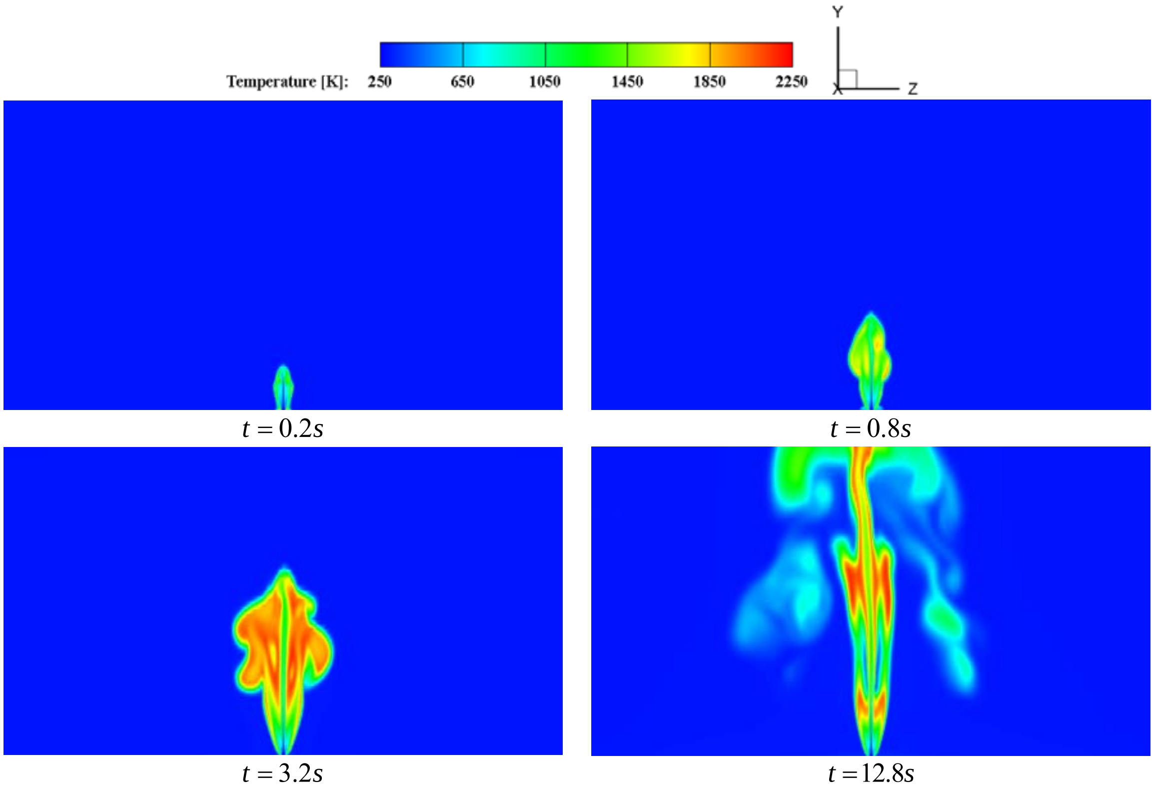

Fig. 3 Temperature contours at different times after accident

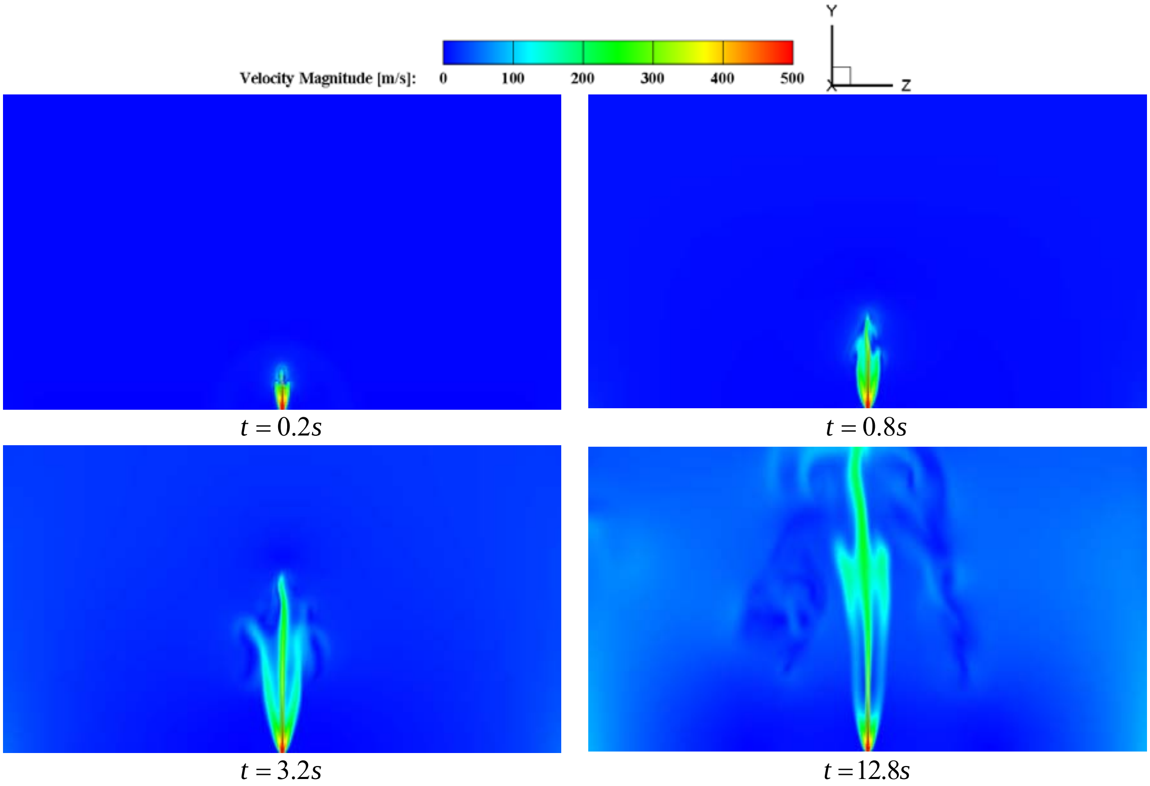

Fig. 4 Velocity contours at different times after accident

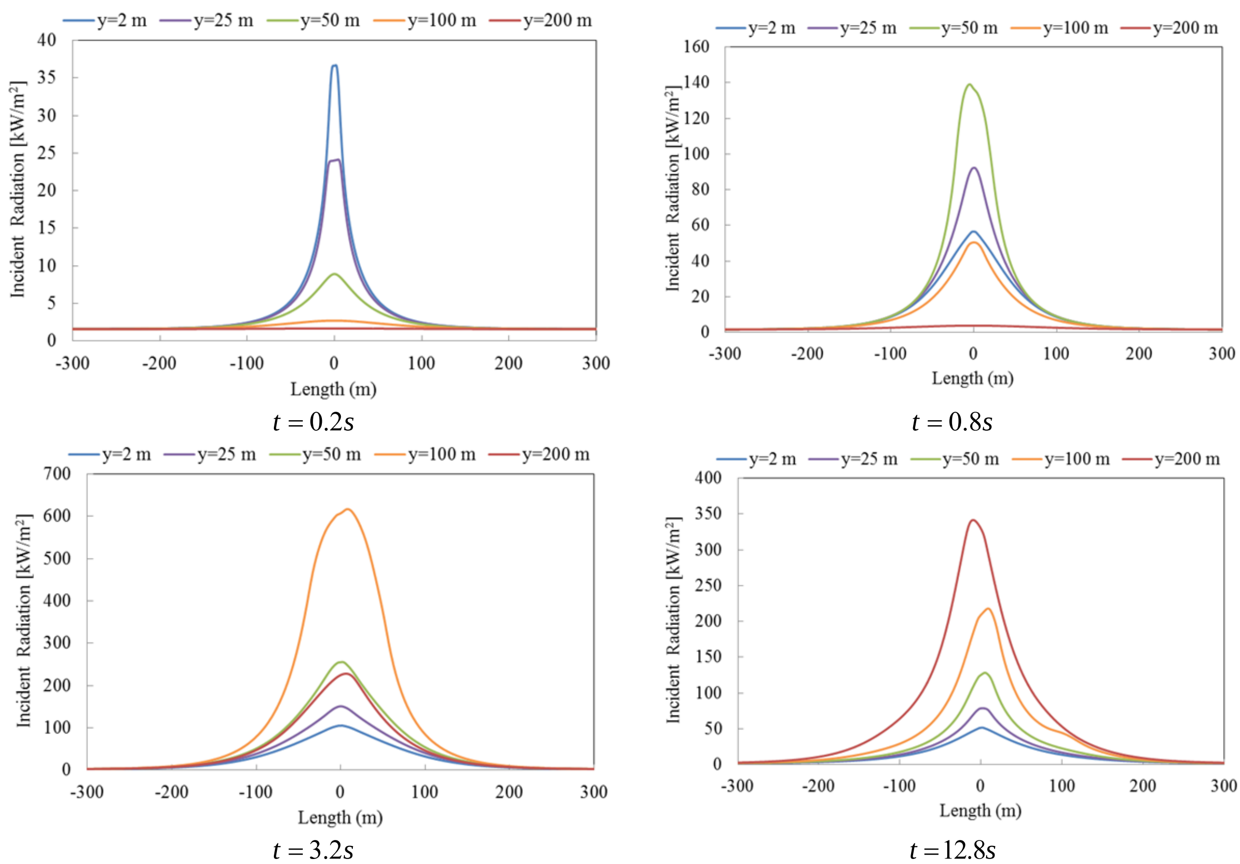

Fig. 5 Incident Radiation at different times after accident

IV. Conclusion

In this study, safety distance around high pressure gas transmission pipelines were determined by using improved and more realistic assumptions (continuous ignition, 3D, unsteady and turbulent cases). Simulations contained three steps of natural gas leakage, mixing and combustion processes. In the numerical simulations, finite volume method was used for discretization of governing equations, and k-ω SST model was used for considering turbulent effects. Moreover, the eddy dissipation method was applied for combustion simulations. Results of numerical simulation showed that, due to high pressure difference between pipeline and environment, choking phenomenon occurred in the cracked area and velocity of released gas reached to sound speed. Also, scrutiny through the incident radiation results indicated that safety distances around 42 inches natural gas pipeline based on 5 and 15 kW/m^2 criteria are 205 and 272 meters, respectively.

Acknowledgment

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

References

[1] 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.

[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] 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.

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

[5] P. R. Oenbring, T. R. Sifferman, "Flare design-Are current methods too conservative", *Hydrocarbon Processing*, vol.60, no. 5, pp. 124-129, 1980.

[6] M. J., Stephens, L. Keith, and K. M. Daron, "A model for sizing high consequence areas associated with natural gas pipelines", In *4th International Pipeline Conference*. American Society of Mechanical Engineers, 2002.

[7] G. Fumarola, D. M. de Faveri, R. Pastorino, and G. Ferraiolo, "Determining safety zones for exposure to flare radiation". In *IChemE Symposium Serie*, vol. 3, pp. 23-30, October 1983.

[8] J. S. Haklar, "Analysis of safe separation distances from natural gas transmission pipelines", *New Jersey Institute of Technology*, PhD Thesis, 1997.

[9] "WHAZAN User Guide", *Technica international Ltd,* March 1988.

[10] P. K. Raj, J. A. Morris, "Computerized spill hazard evaluation models", *Journal of Hazardous Materials*, vol.25, pp. 77-92, 1990.

[11] J. R. Bakke, D. Bjerketvedt, and M. Bjùrkhaug, "FLACS as a tool for safe design against accidental gas explosions," In *IChemE Symposium Series*, no. 122, pp. 141-152, September 1990.

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

[13] 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.

[14] 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.