Simulation study on the influence of cofferdam on the leakage of liquid ammonia storage tank

Xingxing Zhan1, Hongtao Liu1 and Xuefeng Han 1*

1College of Safety Science and Engineering, Nanjing Tech University, Nanjing, Jiangsu, 211800, China
* Xuefeng Han’s e-mail: safety@njtech.edu.cn

Abstract. This paper mainly researches the influence of cofferdam on the leakage liquid and gas diffusion after the liquid ammonia storage tank leaks. In the experiment, different leak-diffusion models were constructed for comparative analysis, and the fluid mechanics model was used for numerical simulation. At the same time, the calculation domain size was determined according to the blocking rate principle, and a physical model was established for simulation analysis. The experimental results show that the reasonable setting of the cofferdam can effectively reduce the diffusion distance of ammonia gas in the downwind direction and the concentration of the ammonia gas in the downwind direction after the leakage of the liquid ammonia storage tank, and is of great significance for reducing casualties and accident losses.

1. Introduction
Due to the complicated environment for the production and storage of liquid ammonia and the toxicity and corrosive nature of liquid ammonia, it is easy to cause leakage accidents, which may lead to explosions, personnel poisoning and environmental pollution accidents.

Questions about the impact cofferdam leakage and diffusion of hazardous gases, domestic and foreign scholars have done some theoretical and experimental studies. In the 1980’s, the Livermore National Laboratory on LNG leak five large-scale field trials using four holes have a lot of each other at right pipe, leaking into a rectangular (length 60 m, width 40 m) the pool, the pool has a square outer cofferdam (length 88 m, width 44 m), wind baffle further (high 13.3 m, 17.1 m in width rectangular), for the purpose of the experiment to study LNG vapor diffusion cofferdam inhibitory of LNG and complex environmental impact of vapor diffusion, there is provided a numerical simulation research data later. Jiang Fangjin [1] analyzed the situation without leakage where different cofferdam and the LNG storage tank leak diffusion studied dam height, diffusion of the distance from the source of leakage of LNG leakage parameter. Since my country cofferdam for liquid ammonia tank without clear defined area is set to carry out simulation of liquid ammonia tank set up so the paper cofferdam liquid ammonia diffusion influence right.

2. Simulation calculation based on fluent

2.1. Establishment of model
According to GB50351-2014“code for design of fire dike in storage tank farm” and SH3125-2001 “code for design of petrochemical fire dike”, this article takes a 400m³ liquid ammonia storage tank as the research object, the ammonia tank diameter 9m, tank 70% of the volume capacity, leakage at the pipe interface port is on the bottom of the tank having a diameter of 0.016m circular mouth leak, leak
port vertical height between the ground and the cofferdam is 0.8m. Herein is provided at the periphery of the leak source of a cofferdam quadrilateral which the ground size of 22 m × 22 m, the height of the dam is 1.5 m, the distance between the cofferdam and the storage tank and the effective volume in the cofferdam meet the relevant design standards requirements.

2.2. Model calculation

This paper assumes that the natural wind speed is the annual average wind speed of 3m/s, the temperature in the storage tank is 20°C, and the pressure is 857120 Pa. The leakage time is 1 min and the leakage rate is 4.03 kg/s. The leakage rate is determined according to the calculation formula of the leakage source, which is calculated as follows:

\[ Q_m = C_0 \rho A \sqrt{\frac{2(P_1 - P_0)}{\rho} + 2gz_0} = 4.03 \text{ kg/s} \]  

(1)

In the formula, \( Q_m \) is the leakage rate of ammonia gas, \( C_0 \) is a dimensionless parameter; when the leakage hole is circular, 0.65 is taken; \( \rho \) is the density of liquid ammonia, 610.26 kg/m\(^3\); \( z_0 \) is the height of the leakage hole to the liquid surface, 6 m is taken, \( P_0 \) is the outside atmospheric pressure of 101325 Pa.

2.2.1. Build computational domain

In engineering applications, liquid ammonia tanks are generally in an open atmosphere, but numerical simulation is impossible to simulate the entire atmosphere, while the computational domain is too large, too much will affect the grid computing speed, the computational domain is too small and affect the accuracy of the simulation results, thus setting a reasonable computational domain is essential. Under normal circumstances, in accordance with the principle of blocking rate seen:

\[ \varphi = \frac{A_b}{A_d} \]  

(2)

Where \( A_b \) is the Obstruction of frontal area; \( A_d \) is the cross-sectional area of the computational domain.

The results of Baetke and Wemer [2] show that the blocking rate is not more than 3%, then the flow field near the obstacle is basically not affected by the size of the calculation domain, so many studies basically determine the calculation domain according to this principle. Further, Yin Huijun [3] and Lakehal [4] found that in addition to meet the blocking probability further principles associated with the presence position of the object. The study pointed out that the flow surface of the computational domain should be greater than the distance to the object 10H (H is the height of the tank), 6H minimum distance face the inflow and the computational domain width of at least 8H. Oliveira and Younis’ [5] research on three-dimensional simulation found that the height of the computing domain has little effect on the calculation results.

In this paper, the height of the obstacle is selected 5 times. Under the condition of following the obstacle, the length, width and height of the calculation domain are determined to be 150m, 80m and 45m respectively according to the actual situation. The blocking rate of the model is 2.2%, and the blocking rate is not more than 3% request. The schematic diagram of the calculation domain cofferdam is shown in figure 1.
2.2.2. Grid division
Before Fluent calculation, to be meshed, the quantity and quality of the grid has an important influence on the accuracy of the results. This article selects the mesh function that comes with ansys workbench for meshing. Due to the large computation space, the leakage of ammonia rapid change in the vicinity of the mouth, so to close the mouth of the grid leak encrypted to improve the results. The meshing results and information are shown in figure 2 and table 1.

Table1. Meshing Information

| Level | Cells  | Faces   | Nodes  | Partitions |
|-------|--------|---------|--------|------------|
| 0     | 896114 | 247468  | 721529 | 3          |

2.2.3 Simulation parameter setting in the calculation domain
(1) Calculation fluid material defined domain
Dividing the mesh into fluent good in checking the mesh, material in the defined region is calculated, using the fluent material library ammonia directly read.

(2) Select model
The standard k-ε model and the unreacted component transport model were selected to simulate and analyse the process of ammonia diffusion in the calculation domain.

(3) Set the internal environment
Consider the influence of gravity, so when you set up within the context of the gravity option is turned on, is set to -9.8m/s². A separate solver and SIMPLE are used to solve the difference equation.

3. Simulation Results Analysis

3.1. Analysis of turbulent kinetic energy
Figures 3 and 4 are turbulent kinetic energy clouds under the influence of cofferdam and natural wind speed on the y=40m and z=1m planes at t=60s, respectively.
Figure 3. Turbulent energy cloud diagram under the action of natural wind speed and water curtain at 
y=40m plane (a: natural wind speed; b: cofferdam effect)

Figure 4. Turbulent energy cloud diagram under the action of natural wind speed and water curtain at 
z=1m plane (a: natural wind speed; b: cofferdam effect)

It can be seen from the figure 3 that in the plane of y=40m, the turbulent kinetic energy at the upper 
edge of the cofferdam is significantly higher than that at natural wind speed. From figure 4, the 
z=1m plane shows that the turbulent kinetic energy near the inner wall of the cofferdam is 
significantly higher than that at natural wind speed. In addition, the turbulent kinetic energy between 
the leakage source and the cofferdam is also significantly increased. This is because the blocking 
effect of the cofferdam causes the backflow of gas to increase the turbulent kinetic energy. The 
increase in turbulent kinetic energy in the area near the cofferdam is due to the change in the direction 
of the cofferdam and the increase in the wind speed after the ambient wind in the computing domain 
encounters the cofferdam, which accelerates the turbulent flow of ammonia gas and air and improves 
the interior of the cofferdam. The turbulent kinetic energy of the upper edge of the cofferdam 
accelerates the rate of ammonia gas mixing with the air near the cofferdam, which is beneficial to 
reduce the concentration of ammonia gas.

3.2. Concentration field analysis

Figure 5 is a comparison chart of the ammonia gas mass distribution on the y=40m surface with and 
without cofferdam, and figure 6 is the horizontal z=1m surface without and with cofferdam a 
comparison chart of the ammonia mass fraction distribution over time. In this section, the mass 
fraction of the directly damaged area (IDLH is approximately equal to 0.0003) is selected as the 
critical concentration. It can be seen from figure 5 that when the gas moves to the cofferdam, due to 
the blocking effect of the cofferdam, the gas moves upward to bypass the cofferdam in the downwind 
direction and continues to diffuse downwind. Comparing the IDLH area with or without cofferdam in 
figure 5, the IDLH area is significantly reduced when the cofferdam is added, and the presence of the 
cofferdam improves the vertical diffusion height of ammonia gas, which is helpful to reduce the wind 
direction to the ground under the cofferdam ammonia concentration.
Figure 5. Ammonia gas mass distribution on vertical surface with or without cofferdam (y=40m)

Under the influence of wind speed and gravity, the ammonia gas diffuses from the top of the cofferdam and continues to diffuse downward. As shown in figure 6 below, the cofferdam has a significant effect on the diffusion of ammonia gas, and the wind direction under the cofferdam is symmetrical. The vortex caused ammonia to deviate from the centerline behind the cofferdam, forming two bifurcated clouds, reducing the time for the ammonia to move downwind. From figure 6 (a1) (b1), we can see that the IDLH iso-concentration line with cofferdam is at most 91m, and 108m without cofferdam, which obviously reduces the diffusion distance of ammonia gas in the downwind direction.

Figure 6. Ammonia mass fraction distribution on horizontal plane with or without cofferdam (z=1m)

4. Conclusion
This article refers to China's relevant design codes and standards, establishes a physical model of the storage tank, compares the diffusion of ammonia gas under natural wind speed and the diffusion of ammonia gas under the cofferdam, and analyzes the cofferdam leakage to the liquid ammonia storage tank under natural wind speed. The influence of ammonia diffusion is mainly that the cofferdam's blocking effect on ammonia leaves ammonia in the cofferdam, and increases the turbulence intensity
near the cofferdam, accelerates the flow of ammonia gas near the cofferdam and increases the mixing rate of ammonia and air. Therefore, the cofferdam can effectively reduce the diffusion distance of ammonia gas in the downwind direction and the concentration of ammonia gas in the downwind direction.

References

[1] Jiang FQ. (2016) Simulation study on the influence of cofferdam on leakage of LNG storage tank. D. Huazhong University of Science and Technology.
[2] Baetke F, Werner H, Wengle H. (1990) Numerical simulation of turbulent flow over surface-mounted obstacles with sharp edges and corners. J. Journal of Wind Engineering & Industrial Aerodynamics, 35(1-3):129-147.
[3] Yin HJ. (2006) Numerical simulation of wind load on membrane structure. D. Tongji University.
[4] Lakehal D. (1999) Computation of turbulent shear flows over rough-walled circular cylinders. J. Journal of Wind Engineering and Industrial Aerodynamics, 80(1):47-68.
[5] Oliveira P J, Younis B A. (2000) On the prediction of turbulent flows around full-scale buildings. J. Journal of Wind Engineering & Industrial Aerodynamics, 86(2):203-220.
[6] Ji J, Chen XX, Han XF. (2014) Rapid Simulation and Visualization Analysis of Liquid Ammonia Tank Leakage Risk. J. Procedia Engineering, 84:682-688.
[7] Ji J, Han XF. (2015) Numerical calculation of the risk of leakage of LNG storage tanks. J. Natural Gas and Petroleum. 33 (03):54-57+10.