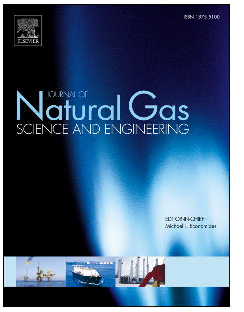
Accepted Manuscript



Numerical simulation and experiment on the law of urban natural gas leakage and diffusion for different building layouts

Aihua Liu, Jian Huang, Zhiwen Li, Jieyun Chen, Xiaofei Huang, Ke Chen, Wen bin Xu

PII: S1875-5100(18)30121-5

DOI: 10.1016/j.jngse.2018.03.006

Reference: JNGSE 2497

To appear in: *Journal of Natural Gas Science and Engineering*

Received Date: 7 August 2017

Revised Date: 18 January 2018

Accepted Date: 8 March 2018

Please cite this article as: Liu, A., Huang, J., Li, Z., Chen, J., Huang, X., Chen, K., Xu, W.b., Numerical simulation and experiment on the law of urban natural gas leakage and diffusion for different building layouts, *Journal of Natural Gas Science & Engineering* (2018), doi: 10.1016/j.jngse.2018.03.006.

Numerical simulation and experiment on the law of urban natural gas leakage and diffusion for different building layouts

Aihua Liu,[a] Jian Huang,[a] Zhiwen Li,[a] Jieyun Chen,[a] Xiaofei Huang,[a] Ke Chen,[a] Wen bin Xu[a,*1]

[a]School of Environmental Science and Engineering, Guangdong University of Technology, Guangzhou, 510006, China

Abstract: To provide a more reliable theoretical basis for emergency-management decisions following accidental natural gas leaks, a numerical simulation and an experiment were conducted in this study to investigate the effects of complex construction environments on natural gas leakage and diffusion laws. The design of a building was divided into an enclosed layout, a patch layout, and a street canyon layout, from the perspective of their impact on environmental winds. Natural gas leakage and diffusion in three layouts were simulated using a three-dimensional computational fluid dynamics (CFD) model, in which both the distribution of natural gas concentration and dangerous areas were determined through comparative analysis. The results of a small-sized experiment showed that the blocking function of an enclosed layout for environmental wind was the highest, its vortex effect was the strongest, and its range of high gas concentration was the widest among the layouts. A cavity among the buildings was the site of major gas accumulation, making it the most crucial area for the emergency management of accidental natural gas leaks. The proposed CFD model was demonstrated be able to be used to simulate and predict the diffusion of natural gas in cases of accidental leakage. And the results of this study can guide building layout planning and gas pipeline construction to prevent accidents.

Keywords: natural gas pipeline; natural gas leakage; building layout; numerical simulation; concentration distribution; small-sized experimental model

1. Introduction

The urban natural gas industry in China has grown significantly in recent years. This has led to an increase in accidental fires, explosions, poisonings, and similar mishaps caused by natural gas pipeline leakages resulting in loss of life and property as well as environmental pollution. As the area of gas diffusion and the corresponding accidental consequences cannot be accurately forecast following a leak, the absence of such knowledge affects emergency decision making and can increase the damage in such situations. Therefore, the laws of gas leakage and diffusion are a popular area of research in gas transportation safety. The relevant research methods mainly involve field tests, wind tunnel experiments, and computational fluid dynamics (CFD) simulations. As CFD can be used to accurately simulate the flow field, is cheap, and has easy settings, it has emerged as an important method to research the gas diffusion law. To prevent erroneous simulation results due to the inappropriate choice of calculation model or parameter settings, some important parameters often need to be verified through a series of experiments to ensure that the results of the CFD simulation can be used to forecast the diffusion rule of gas clouds. Prankul et al. (2010), Christof and Bodo (2007), and Cuiwei et al. (2015) studied gas leakage and diffusion through CFD and verified the accuracy and feasibility of the simulation results through experiments. In order to verify the rationality of the CFD simulation, Peng-Yi et al. (2016) combined CFD and wind-tunnel measurements to investigate buoyancy turbulent flows and pollutants dispersion in different scale urban areas. Venetsanos et al. (2009) simulated and analyzed the mixing and distribution processes of hydrogen after short- and long-term releases by using CFD, and designed a corresponding experiment platform to compare the differences between the simulation and experimental results. Ebrahimi-Moghadam et al. (2016) developed a few equations to estimate leakage from above-ground and buried urban natural gas pipelines according to simulation results.

The influence of different factors on the gas diffusion law has been explored by several researchers. Wilkening and Baraldi (2007) studied the dispersion of hydrogen and methane with different properties by using CFD, and introduced the wind effect on the gas diffusion process. Zhang et al. (2006) analyzed the influence of wind speed and direction on the gas diffusion law. Qian-Xi and Dong (2016) pointed out that the range of gas

*Corresponding author: E-mail: xuwenbin@tom.com