عنوان مقاله:

Numerical Investigation of Hazardous Gas Dispersion Over Obstacles and Residential Areas

نویسندگان:

E. Fatahian - *Department of Mechanical Engineering, Nour Branch, Islamic Azad University, Nour, Iran*

H. Salarian - *Department of Mechanical Engineering, Nour Branch, Islamic Azad University, Nour, Iran*

H. Fatahian - *Department of Mechanical Engineering, Nour Branch, Islamic Azad University, Nour, Iran*

خلاصه مقاله:

In the present study, an attempt has been made to use Computational Fluid Dynamics (CFD) in the assessment of hazardous gas dispersion over obstacles. For this aim, the accidental dispersion of hazardous gas from the hole and the effect of different parameters such as changes in inlet wind velocity, the direction of the pollutant cloud and its movement, mass fraction of gas dispersion, and the pressure distribution were numerically analyzed. The flow was assumed as three-dimensional, unsteady, turbulent, and compressible. Different turbulent models were used in modeling the gas release and the most accurate one was suggested. The numerical simulation demonstrated that the gas mass fraction increased significantly due to the sudden dispersion of the gas. The amount of gas concentration gradually decreased after the formation of pollutant clouds by moving in the horizontal direction. Moreover, gas mass fraction had decreased by increasing the height. Comparing the results revealed that the pollutant cloud did not cover the surrounding area in the wind velocity of ۱ m/s. Therefore, the pollutant clouds generated in this case could not impose a threat. In higher wind velocities (۳ m/s and ۵ m/s), the pollutant cloud approximately covered the surrounding areas, which caused a severe threat. The maximum overpressure at the hole is ۵.۷ Pa for a wind velocity of ۵ m/s, while the maximum negative pressure was about -۷.۱ Pa. The influencing radius was obtained about ۹.۳ m. The overpressure did not cause obvious damage to buildings but led to a slight hurt to humans.

کلمات کلیدی:

Gas Dispersion, Computational Fluid Dynamics, Mass Fraction, Obstacles, Turbulence

لینک ثابت مقاله در پایگاه سیویلیکا:

https://civilica.com/doc/1185294