3D CFD Simulation of Manufactured Gas Pipeline Leakage and Dispersion in Urban Areas

Haoli CHENG, Dejun LIU
College of Petroleum Engineering
Liaoning Shihua University
Fushun, China

Abstract: Release of city gas from the pipelines distributed in urban areas is a major concern in urban community safety with the municipal construction. Once gas releases through damaged pipes, accidents would happen involving substantial economic losses and even victims amongst the population. To assist in emergency response decisions in case of released gas from broken pipelines in urban areas, the transport and dispersion of the Manufactured gas has been simulated by the 3D Computational Fluid Dynamics (CFD) model. The release is simulated as a flow through a big hole on an underground pipeline. Results on the effect of wind and the buildings are described, and it can be also used as reference for taking an emergency measures.

Keywords: Computational fluid dynamics (CFD); Manufactured gas; Coke oven gas; Leakage; Urban areas

1. Introduction

At Present, the construction of city gas pipeline is rapidly extending in China, but due to the aging of pipeline, the aggravating corrosion, the welding flaw or the municipal construction, the pipeline is easily ruptured or broken. The presence of high population density in such areas multiplies the magnitude of the consequences and the buildings with complex geometries are involved leading to 3D flow fields that strongly influence gas dispersion. Once the gas diffusing into the atmosphere mixes with air and forms inflammable premix gasses, the hazards of fire and explosion are looking up. Even if the fire or explosion doesn't occur, the toxic constituents of the leaking gases cloud carried by wind may reach threshold limit values (TLVs) in some area, which is toxic to people around and pollutes environment. Therefore to carry out numerical simulation study on gas pipeline leakage and diffusion will mean a lot to transportation security and life and property safety.

2D or 3D Computational Fluid Dynamics (CFD) tools have increasingly begun to play an important role in simulating the transport and dispersion of the gas cloud release in an industrial site or large city[1-5]. In the present work the dispersion of the coke oven gas release was simulated by the 3D and 2D CFD approach referring to the study on pollutants dispersion in urban street canyon .

2 Accidental scenario definition

2.1. Physical processes and Release conditions

The situation considered is an accidental leakage of coke oven gas from damaged city gas pipeline as a result of urban construction. Coke oven gas was chosen because it is important by-product of coal chemical industry and is widely used as the City gas (Manufactured gas). The problem can be modeled by a domain consisting of hole from the damaged pipe as a point source of manufactured gas. The manufactured gas was composed of CH_4 (27%), H_2 (58%), and N_2 (15%). The release was modeled as a ground-level area source with inflow rate equal to 420 m/s for 60 s, whereas atmospheric conditions (1atm, 300K), and wind speed equal to 1 m/s, 5 m/s and 10 m/s were considered.

Aiming to assess such a hazard and to take proper preventive measures, CFD modeling has been used to predict how the leakage concentration varies with time at fixed coordinates.

2.2. The 3D model and Boundary conditions

In order to simulate complex geometries and to analyze the effect of obstacles on gas dispersion, 3D simulations have been applied. As the 3D approach requires a larger amount of computer resources and consumes much longer CPU time, 2D ones were also performed to some situation.

The modeling area used was 150 m×170 m×100 m in the x, y, z directions, respectively as is shown in Fig.1 and Table 1. Two 20 m long, 20 m wide and 20 m high buildings are placed along the road of the domain. The road between the two buildings was 30 m wide. While, A dug hole with 0.2 m diameter in the middle of the road is shown in the figure at the coordinate x=0 m, y=0 m, z= 0 m. The release is modelled as a flow through the hole between the pipeline and the outside environment.

 978-1-935068-37-2

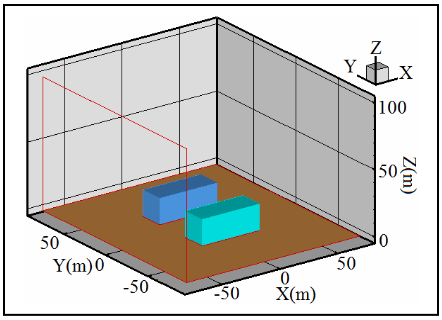

Fig. 1. Schematic drawings for 3D scenarios

Table 1. Computational domain for 3D geometries

Cells	Faces	Nodes
2080594	4279226	407446

Two inlet mean direction of the wind were required, along the +x and +y , giving flow rates of 1 m/s, 5 m/s and 10 m/s. The initial condition is zero concentration of the contaminant everywhere in the domain. The near-wall grid spacing should also be made refined to ensure the validity of the model using the local grid refinement method, and it is the same near the gas inlet. The Realizable k-ε model was also chosen for the test cases presented here. The time step size used for the solution was 1 s, and the max interactions per time step was 20.

The boundary conditions can be seen in Table 2.

Table 2. Table Type Styles

Boundary	Type	Notes
Wind inlet	Velocity inlet	Wind velocity, tempertuare et al.
Wind outlet	Outflow	
Top	Outflow	
Lateral	Outflow	
Buildings	Wall	No-slip conditions
Gas inlet	Velocity inlet	Mass flow
Ground	Wall	No-slip conditions

3. Results and discussion

Fig.1 shows the flow field around the buildings before the release occurred in the case with wind velocity of 5 m/s at different directions. The velocity contour around the buildings can be seen in Fig.1 (a) and (b). The wind blows along the +x axis direction for (a), and +y axis direction for (b).

Fig.1 (c) and (d) shows the corresponding pressure contour of (a) and (b). In the wake of the building, a large low wind velocity zone has developed (a). Fig.1 (a) shows that the wind velocity will be accelerated when traveling through the urban street canyon, and the pressure is much lower in the canyon. Both the pressure and wind velocity are when the wind blows along the +y axis direction (that is the road).

The turbulence due to the wake of the building becomes the most important contribution to the upwind diffusion of the gas, which will be described in the following work.

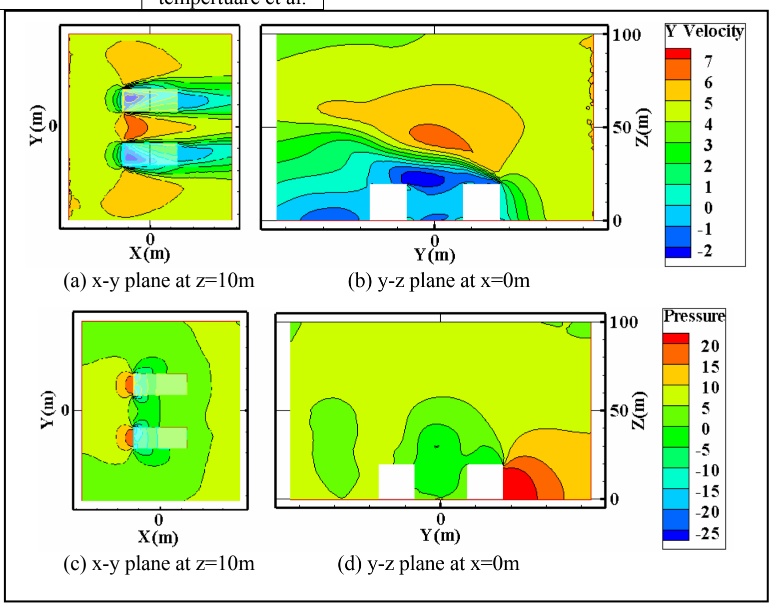

Fig.1. Pressure and Velocity contour of a steady-state flow field around buildings, generated by wind at 5 m/s with different directions.

Fig.2 shows methane concentrations at 10s and 30 s after the release started under different wind speed. As is shown in the simulation, flow can be easily driven by the wind down to the ground when the wind is stronger. That means ignition is more likely at ground level or close to buildings.

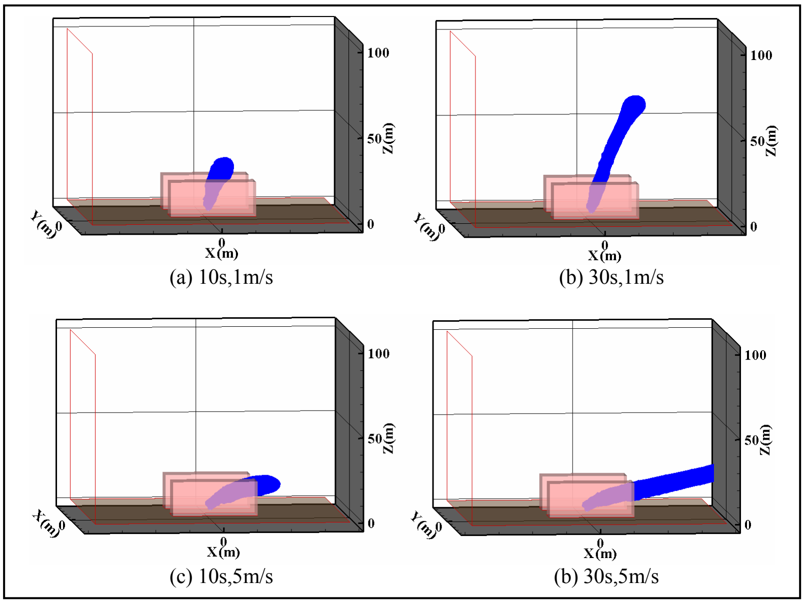

Fig.2. Concentration distribution of the gas 10s and 30 s release under wind velocity of 1 m/s and 5 m/s blowing along the +x axis direction

Fig.3 shows methane concentrations (the regions outside the alarm concentration are uncoloured) at 30 s after the release started. The jet has enough impulse to reaches high up into the environment when the wind velocity is not strong enough (1 m/s as is shown in Fig.3), which is able to trap some methane near one of the buildings. As the wind get stronger, the impulse of the gas is not strong enough to go higher up and the flow became driven by the wind down to the ground.

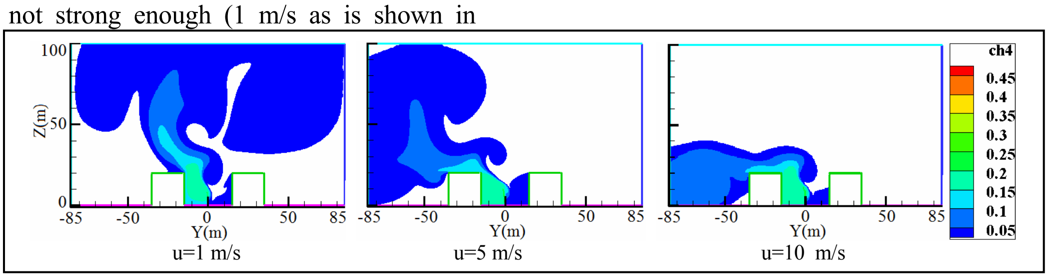

Fig.3. Simulated dispersion after 30 s release under different wind velocity .

In Fig.4 the regions above the alarm concentration (1% as recommended in GB 50494—2009[7]) are coloured in blue with different time under wind velocity of 5 m/s. Due to the wind the gas is transported downstream (to the left) as the continuous. The gas diffuses along the wall of the building downwind and larger amounts of methane would be trapped behind the building as the gas spilled over the top of the left building. Alarming mixtures can be observed in two regions: within the road between the two buildings, and in the lee side next to the left building. This is explained by the turbulence effect related to building wake as is shown in Fig.1.

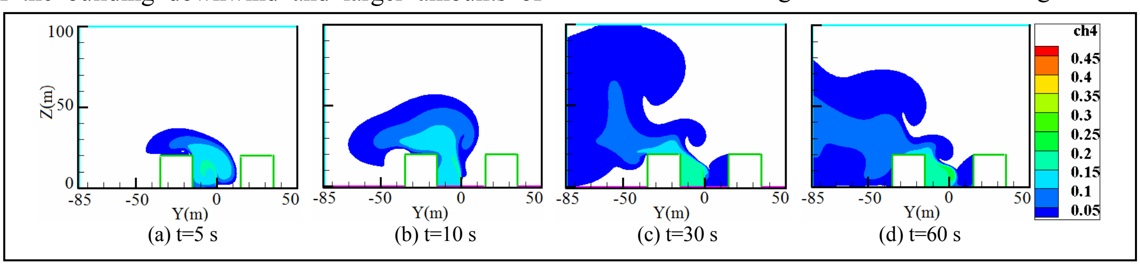

Fig.4. Simulated dispersion at time(a)5 s,(b)10 s,(c)30 s, and(d)60 s after release under wind velocity of 5 m/s

4. Conclusions

The 3D and 2Dsimulations from accidental manufactured gas releases from city gas pipelines has been presented in this work. The 2-D approach was followed because of the prohibitive computer run-time of a fully 3D simulation. With 2D or 3D calculations, it was possible to investigate different configurations such as environments with different wind conditions and the effect of buildings. It is stressed that the CFD models did not take into account the influence of vegetation obstacles, such as trees and bushes. Because of the limited number of simulations performed, in order to assess the unsteady state leakage, further investigations are required and that goes beyond the scope of the present work.

Finally, the key stones of the presented methodology are summarized into a few recommendations:

- Appropriate meshing and boundary conditions must be applied before undertaking the calculation runs. In order to limit the CPU time, a non-uniform grid is recommended, with minimum grid space applied close to the obstacles and the release position, and with larger space at the far-field boundaries. It is strongly recommended to use realistic (experimental) data for the wind velocity and wind direction of the leakage area and introduce them into the domain via inlet boundary conditions.
- If the CPU has great performance, it is important that the selected model considers the most conditions and build up 3D city model[8], a more complete reconstruction of the urban environment, where buildings are accurately modeled. As the urban areas are characterized by high population densities and complex geometries resulting from the large number of buildings of varied shapes and dimensions, representing such complex geometries simply but realistically can prove difficult for 2-D models.
- When there is a leakage, the valve of the pipe must be cut off as soon as possible. The result of this paper indicated that the wind velocity has a great effect on gas diffusion. The gas diffuse along the downwind direction and the flammable mixture will be trapped in the low pressure and low wind speed zones around the buildings, the probability of an accidental ignition. So it is crucial that warning sign or forbidden zone must be set up under the terms of national standards[9] to avoid causing fire and explosion danger or poisoning hazards resulting from local residents, pedestrians or vehicles.

References

[1] Gerdes F, Olivari, “Analysis of pollutant dispersion in an urban street canyon”, Journal of Wind Engineering and Industrial erodynanucs 1999, 82: 105-124.

[2] M.Pontiggia, M.Derudi, M.Alba et al.. Hazardous gas releases in urban areas: Assessment of consequences through CFD modeling[J]. Journal of Hazardous Materials, 2010,176: 589-596.

[3] Y.Yang,Y.Shao,Numerical simulations of flow and pollution dispersion in urban atmospheric boundary layers[J]. Environmental Modelling & Software, 2008,23: 906-921.

[4] Hanna,S.R., Brown,M.J., Camelli, F.E. et al.. Detailed simulations of atmospheric flow and dispersion in urban downtown areas by Computational Fluid Dynamics (CFD) models–an application of five CFD models to Manhattan[J]. Bull. Am. Meteorol. Soc., 2006, 87: 1713-1726.

[5] Steven R.Hanna, Olav R.Hansen, Mathieu Ichard et al.. CFD model simulation of dispersion from chlorine railcar releases in industrial and urban areas[J]. Strimaitis Atmospheric Environment, 2009,43: 262-270.

[6] Meroney R N, Pavageau M, Rafailidis S, et al. Study of line source characteristics for 2-D physical modelling of pollutant dispersion in street canyons[J].Journal of Wind Engineering and Industrial Aerodynamics, 1996, 62(1):37-56.

[7] GB 50494-2009

[8] M.Pontiggia et al. Hazardous gas releases in urban areas: Assessment of consequences through CFD modeling. Journal of Hazardous Materials 176(2010)589-596.

[9] Ministry of Housing and Urban-Rural Development of the People's Republic of China (MOHURD). GB 50493-2009. Specification for design of combustible gas and toxic detection and alarm for petrochemical industry gas. Beijing: China Planning Press, 2009