Journal of Loss Prevention in the Process Industries xxx (2016) 1–7

Contents lists available at ScienceDirect

Journal of Loss Prevention in the Process Industries

journal homepage: www.elsevier.com/locate/jlp

Validation of geometry modelling approaches for offshore gas dispersion simulations

I. Ahmed [a, b, *], T. Bengherbia [a], R. Zhvansky [a], G. Ferrara [a], J.X. Wen [b], N.G. Stocks [b]

[a] DNV GL, London, UK
[b] Warwick FIRE, School of Engineering, University of Warwick, Coventry, UK

ARTICLE INFO

Article history:
Received 15 March 2016
Received in revised form
31 May 2016
Accepted 8 July 2016
Available online xxx

Keywords:
Gas dispersion
Geometry modelling
Computational fluid dynamics

ABSTRACT

Computational Fluid Dynamics (CFD) codes are widely used for gas dispersion studies on offshore installations. The majority of these codes use single-block Cartesian grids with the porosity/distributed-resistance (PDR) approach to model small geometric details. Computational cost of this approach is low since small-scale obstacles are not resolved on the computational mesh. However, there are some uncertainties regarding this approach, especially in terms of grid dependency and turbulence generated from complex objects. An alternative approach, which can be implemented in general-purpose CFD codes, is to use body-fitted grids for medium to large-scale objects whilst combining multiple small-scale obstacles in close proximity and using porous media models to represent blockage effects. This approach is validated in this study, by comparing numerical predictions with large-scale gas dispersion experiments carried out in DNV GL's Spadeadam test site. Gas concentrations and gas cloud volumes obtained from simulations are compared with measurements. These simulations are performed using the commercially available ANSYS CFX, which is a general-purpose CFD code. For comparison, further simulations are performed using CFX where small-scale objects are explicitly resolved. The aim of this work is to evaluate the accuracy and efficiency of these different geometry modelling approaches.

1. Introduction

Hazardous materials being released into the atmosphere is one of the main causes of accidents in oil and gas installations. The accidental release and subsequent dispersion of flammable gases could cause fires and explosions. Therefore, consequence analysis of gas dispersions is crucial when optimizing the location of gas detectors and to determine the risk of fire and explosion.

A number of modelling techniques are used for consequence analysis of accidental gas releases. These include empirical, phenomenological and Computational Fluid Dynamics models. Both empirical and phenomenological models can provide an estimate quickly and are widely used in the oil and gas industry. However, these models have a limited range of validity since they cannot capture the effects of obstacles and complex terrains.

In contrast, CFD models solve the equations of fluid mechanics in three-dimensions, thus, enabling the accurate representation of complex geometries that are often found in oil and gas production facilities. This makes CFD the preferred method to study gas dispersion, especially for offshore oil and gas installations.

However, the use of CFD is not without its own challenges. One of the major drawbacks of CFD is the time taken to generate the computational mesh required for the simulation. The computational and mesh size and, hence, the cost of such simulations will become prohibitively expensive if one were to resolve all geometrical obstacles (pipes, instrumentation, equipment etc.).

Most CFD codes used in the industry (for example FLACS, KFX and EXSIM) employ the porosity distributed resistance (PDR) approach (Patankar and Spalding, 1974), which treats objects smaller than the computational grid (sub-grid objects) as porous medium. The main advantage of the PDR approach is the reduction in computational mesh size, which in turn reduces the cost of simulations. However, there are some uncertainties regarding this approach, especially in terms of grid independency and turbulence generated by the sub-grid objects.

An alternative approach for gas dispersion simulations, which can be implemented in a general purpose CFD tool, is to body-fit the grid around large objects whilst combining multiple small-scale obstacles in close proximity and using porous media models to

* Corresponding author. Ricardo plc, Shoreham-by-Sea, UK.
E-mail address: irufan.ahmed@ricardo.com (I. Ahmed).

http://dx.doi.org/10.1016/j.jlp.2016.07.009
0950-4230/© 2016 Elsevier Ltd. All rights reserved.

represent blockage effects. A number of such studies can be found in the literature, for example, Gilham et al. (1999) carried out a ventilation study of a gas turbine compartment using the commercial CFD package STAR-CD, where small-scale objects were treated as porous media.

Fothergill et al. (2003) estimated the drag on the flow through the porous region used to represent small-scale geometries. This estimate was made by calculating the drag due to each object and obtaining a sum of the overall resistance. This method was implemented in another commercial CFD package called CFX. They compared their results with EXSIM CFD code that uses PDR method and showed that the turbulent kinetic energy and turbulent eddy dissipation predicted by CFX was much lower than EXSIM. The porous model they used simply reduced the volume of free flow and provided extra resistance, but did not generate turbulence. They proposed the inclusion of additional source terms in the turbulence kinetic energy and dissipation equations to account for turbulence generation by the obstacles.

Savvides et al. (2001) used FLUENT CFD code to simulate large-scale dispersion of fuel in an offshore module, where small-scale objects were treated as porous media, with the coefficients based on BP data. They showed that this approach can predict gas dispersion with good accuracy. Ivings et al. (2008) simulated an under-expanded jet impinging on an array of pipes. In the porous approach, momentum sink terms were included in the fluid flow equations to account for the effect of pipes. They also included turbulence source terms based on the explosion model by Hjertager et al. (1992). Their simulations showed that resolving the objects on the grid did not offer any significant advantages over the porosity based approach.

The main aim of this work is to carry out a validation study of the porosity approach by comparing with large-scale dispersion experiments carried out at DNV GL's Spadeadam test site. In addition, comparison is also made with simulations where the small-scale objects are full resolved. The commercial, general purpose CFD package, ANSYS CFX version 15.0 is used for the simulations carried out in this work.

2. Experimental test cases

The experimental test cases chosen for this validation exercise were carried out at DNV GL's Spadeadam test site as part of the Phase 3B project (Johnson and Cleaver, 2001). In these experiments, high pressure natural gas was released inside a test rig until steady-state flow rates were achieved. This rig measured 28 m long, 12 m wide and 8 m high and represented a full-scale offshore module. The structure contained a mezzanine deck at mid-height, with open bar grating.

Gas concentration was monitored at 50 locations inside the test rig using oxygen sensors. The gas concentrations reported by the experiments were when steady-state conditions have been reached. Meteorological conditions such as the ambient wind speed (horizontal component) and direction were measured using an ultrasonic anemometer, mounted at 20 m above ground.

Twenty-three gas dispersion experiments were carried out using two confinement configurations. In this work, only one confinement configuration is considered, in which one side of the module is open and the remaining 3 sides and roof of the module are closed. Two leak scenarios are considered, which are given in Table 1. The leak location, direction, orifice diameter are identical; the only difference being the leak rate. The wind speed and wind directions are averaged quantities measured during the experiments. Fig. 1 shows the CAD model of the experimental rig with the location of the leaks marked (the two leaks used in this study are pointing along positive *x*-direction).

3. Numerical setup

3.1. Governing equations

CFD involves the solution of highly non-linear, coupled partial differential equations. These are the equations of mass conservation, Navier-Stokes (momentum conservation) and energy equations. In addition, in dispersion problems one needs to solve additional transport equations for the gas.

Even with the current computational power it is computationally prohibitive to solve these equations directly for most industrial flows; one has to resort to some form of averaging. In this work, the Reynolds-averaged form of the governing equations is used, since it is computationally less expensive and can yield good agreement for time-averaged quantities.

3.2. Turbulence model

Reynolds-averaging the governing equations lead to the turbulence closure problem, which requires models to close the equations. A number of turbulence models have been developed over the years, the standard k-ε model being the most popular for gas dispersion studies. It is well known that the standard k-ε model yields poor results for separated flows, impingement and flows with extra strains (Argyropoulos and Markatos, 2015).

One improvement to the standard k-ε model is the Renormalization Group (RNG) k-ε model, which is better suited for separated flows and flows around bluff bodies (Argyropoulos and Markatos, 2015). Therefore, the RNG k-ε model is used in this work along with scalable wall functions to model the boundary layer near the walls and the terrain. Note that the default turbulence parameters used in ANSYS CFX v15.0 (ANSYS, 2013) are used in this work without any modification.

3.3. Geometry and mesh generation

Two geometrical representations of the offshore module with different levels of detail are considered in this work. In the first representation, all small-scale objects are resolved using the computational mesh. This geometry will be referred to as the *resolved geometry* in the rest of the paper. In the second representation, areas with large amount of small-scale objects (for example piping) are combined to form a porous region. This geometry will be referred to as the *porous geometry*. The cut-off criteria are: any pipe with diameter smaller than 15.24 cm and boxes with cross-sectional area smaller than 232 cm^2 are treated as small-scale objects.

Table 1
Two test cases to be simulated in this work; includes leak properties and atmospheric conditions during the test.

Test cases	Leak location (*x*, *y*, *z*)	Leak direction	Leak rate (kg/s)	Orifice diameter (mm)	Wind speed (m/s)	Wind direction (°)
16	(5.5, 6, 1.7)	+X	2.6	32.5	0.9	160
17	(5.5, 6, 1.7)	+X	6.9	32.5	4.2	245

Please cite this article in press as: Ahmed, I., et al., Validation of geometry modelling approaches for offshore gas dispersion simulations, Journal of Loss Prevention in the Process Industries (2016), http://dx.doi.org/10.1016/j.jlp.2016.07.009