

Congresso ABRISCO 2017

Modelling of Accidental Gas Dispersion using CFD: What is the impact of small-scale geometry?

Sávio Souza Venâncio Vianna
School of Chemical Engineering
University of Campinas

INTRODUCTION

The comprehensive modelling of small scale geometry in CFD studies is of paramount importance when addressing numerical explosion [1]. The fluid flow ahead of the flame front interacts with various obstacles enhancing the turbulent field. The more efficient mixing caused by the turbulence works a positive feedback mechanism accelerating the combustion reaction and leading to more severe explosion.

More recently CFD users have been requested to consider the modelling of small scale geometry when simulating ventilation and gas dispersion studies [2]. The current paper investigates the influence of small scale geometry in the ventilation as well as the gas dispersion in an offshore module measuring 8 long x 3 wide x 3 high metres.

A series of simulations addressing 4 wind directions (North, South, East and West) is conducted considering the large scale objects and small objects down to 2 inches. The ventilation rate, air entrainment rate and flammable cloud volumes are compared for three scenarios. The first analysis considers only the large geometrical objects, such as vessels and tanks while the second analysis considers the piping discipline with pipes along the fluid flow and perpendicular to the fluid flow.

The third analysis considers a combination of parallel and perpendicular arrangements with pipes as small as 2". The small scale geometry is fully resolved in the mesh. Prior to conducting the investigation analysis, the solver set up is compared with available experimental data. The ventilation and gas dispersion simulations are conducted using ANSYS-CFX2 and the turbulence problem is closed using the two equations model for the turbulent kinetic energy and its associated rate of dissipation.

The paper is organised as follows. The next section addresses the formulation of the problem followed by the description of the scenarios investigated. Further the results are discussed and the closing remarks are drawn.

When reading the current work is important to bear in mind that subject discussed here deserves a far more robust attention. Preliminary results are presented and from the point of view of the author future analysis are of paramount importance and required at the actual stage of the research.

FORMULATION

The Navier-Stokes equation (1) was solved alongside the continuity equation (2). The turbulence problem was closed based on Boussinesq formulation using the two equation k-epsilon model.

$$\frac{\partial(\beta_v \bar{\rho})}{\partial t} + \frac{\partial(\beta_k \bar{\rho} \tilde{u}_k)}{\partial x_k} = 0 \quad 1$$

$$\frac{\partial(\beta_v \bar{\rho} \tilde{u}_i)}{\partial t} + \frac{\partial(\beta_k \bar{\rho} \tilde{u}_k \tilde{u}_i)}{\partial x_k} = -\beta_v \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial(\beta_k \sigma_{ki})}{\partial x_k} + \beta_v \bar{\rho} g_i + R_i \quad 2$$

The overall formulation can be written as stated in equation (3) where phi represents the various variables of the formulation.

$$\frac{\partial(\bar{\rho} \phi)}{\partial t} + \frac{\partial(\bar{\rho} \tilde{u}_k \phi)}{\partial x_k} = \frac{\partial}{\partial x_k} \left(\bar{\rho} \Gamma_\phi \frac{\partial \phi}{\partial x_k} \right) + S_\phi \quad 3$$

The mesh sensitivity analysis was conducted for three different mesh sizes. The largest independent mesh size was applied in the current analysis. The steady state mode of the solver was considered since the main interested was the final flammable cloud size and the velocity field pattern in the module. The upwind scheme was applied in the convective term and central scheme blend was applied elsewhere.

A prescribed velocity was set at the inlet of the computational domain and zero gradient was set at the opened boundaries. Low Reynolds dumping wall functions were used accordingly where solid boundaries were found [3].

Leak scenario

In order to investigate the influence of the small-scale geometry a set of CFD (Computational Fluid Dynamics) simulations were conducted. All simulations considered one leak direction pointing down and four wind directions. The wind speed was set as 6.0 m/s.

It is a common practice to parametrise the geometry via PDR (Porosity Distributed Resistance) approaches as shown in Figure 1. Figure 1 (b) shows an example of a porous model of the base geometry presented in Figure 1 (a).

In the current analysis such approach was not considered and the small-scale geometry was fully resolved in the computational mesh.

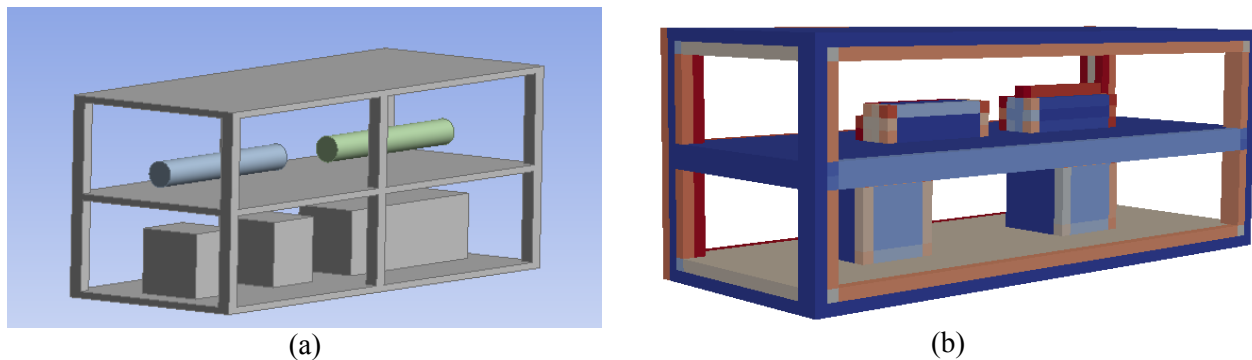


Figure 1 – Geometrical offshore module 8 metres long and 3 metres wide and 3 metres high (a) and the associated parametrised geometry (b) using PFS (Porosity Flow Solver) for porosity calculation [4].

Figure 2 shows the geometries considered in the CFD calculation. Figure 2 (a) shows the base case in which only the large-scale geometry is present. Figure 2 (b) shows a set of pipes down to 2 inches in the upper level of the process module. The small-scale geometry was arranged in two sets comprising pipes parallel and perpendicular to the flow. The volume blockage ratios (VBR) were considered in the analysis (7% and 8%).

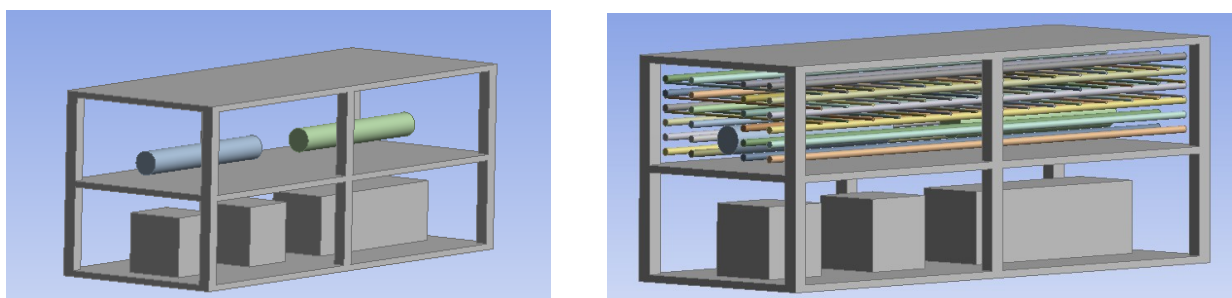


Figure 2 – Base geometry (a) used in the analysis and the congested geometry (b) considering additional number of pipes in the upper level for VBR (Volume Blockage Ratio) of 7%

RESULTS

Ventilation in the module

Analysis of Figure 3 shows the scalar velocity for the wind blowing from West (xlow). Figure 3 (a) shows the the velocity contours for the case in which only the large-scale geometry was considered. Figure 3 (b) shows the velocity contour for the case where the small-scale geometry was resolved in the mesh. The overall velocity in the module is found to be significantly reduced for the case where the small-scale geometry was considered in the computational mesh.

Figure 3 (c) and (d) show the velocity contour at the region close to the largest objects in the geometry. Figure 3 (c) shows regions in between the large pipes where the velocity is higher than the region at the wake of the flow. On the other hand, a closer look at Figure 3 (d) shows a reduced velocity field when compared with Figure 3 (c). A significantly lower velocity field is noticed at the wake of the largest object as well as in between the large vessels

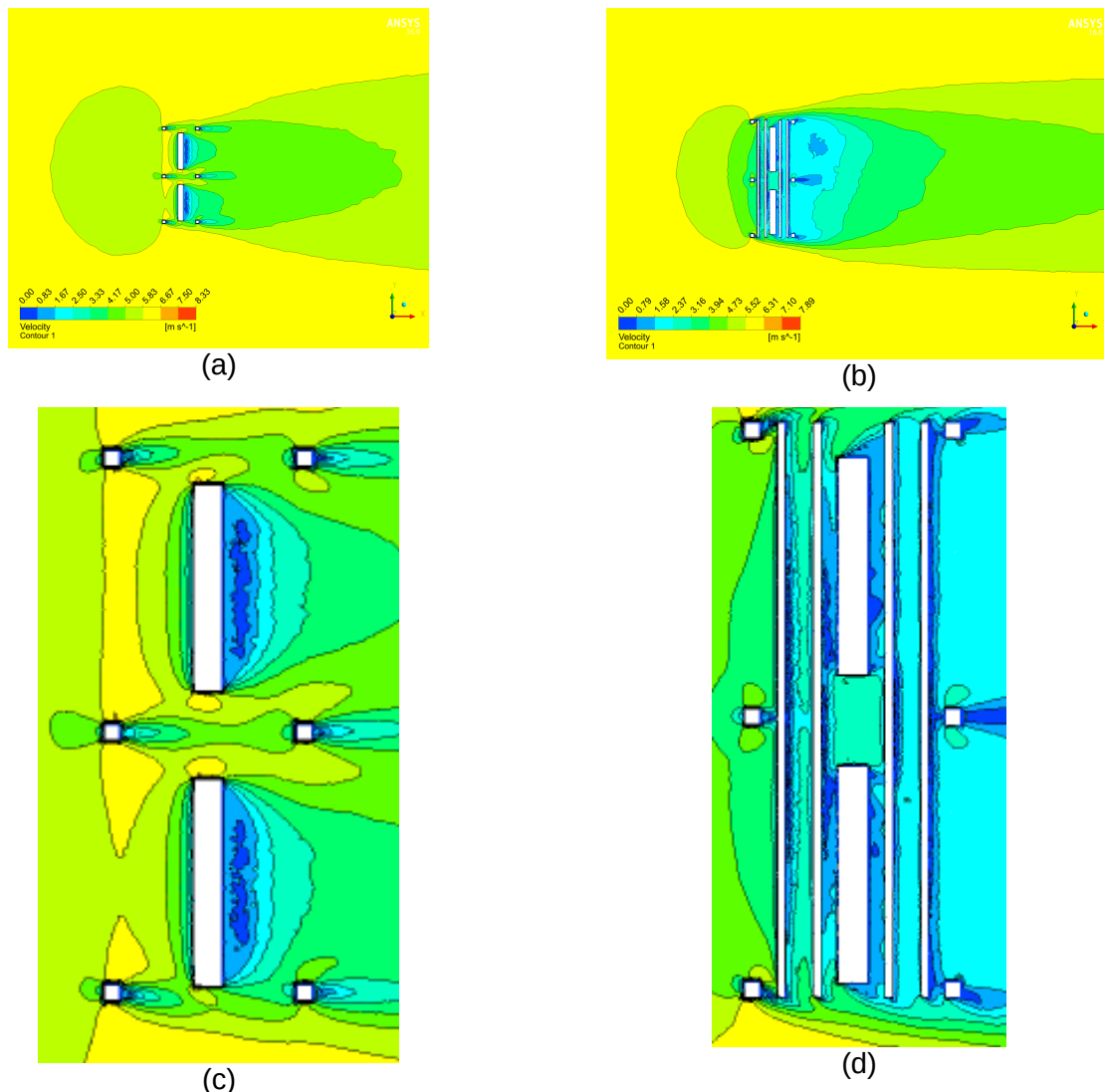


Figure 3 – Velocity contour for the case where the wind is blowing from West (xlow). North is pointing up. Velocity contours in the module for the case where only the large-scale objects were considered (a). Velocity contour for the case where the small-scale geometry was considered (b). Detailed view of the area in the module where the largest vessels are located considering the large-scale objects only (c) and the small-scale objects (d).

Analysis of Figure 4 shows the scalar velocity for the wind blowing from South (yflow). Figure 4 (a) shows the the velocity contours for the case in which only the large-scale geometry was considered. Figure 4 (b) shows the velocity contour for the case where the small-scale geometry was resolved in the mesh. The overall velocity in the module is found to be reduced for the case where the small-scale geometry was considered in the computational mesh. Comparison with Figure 3 shows the reduction of the velocity in the module caused by the small-scale geometry is not as significant as the cases where the wind is blowing for direction. In the cases considered in Figure 4 the wind flow is aligned with the geometry and the mechanical energy drop caused by the drag is reduced since the cross sectional area is smaller than the orthogonal flow.

Figure 4 (c) and (d) show the velocity contour at the region close to the largest objects in the geometry. Figure 4 (c) shows regions in between the large pipes where the velocity is higher than the region at the wake of the flow. On the other hand, a closer look at Figure 4 (d) shows a reduced velocity field when compared with Figure 4 (c).

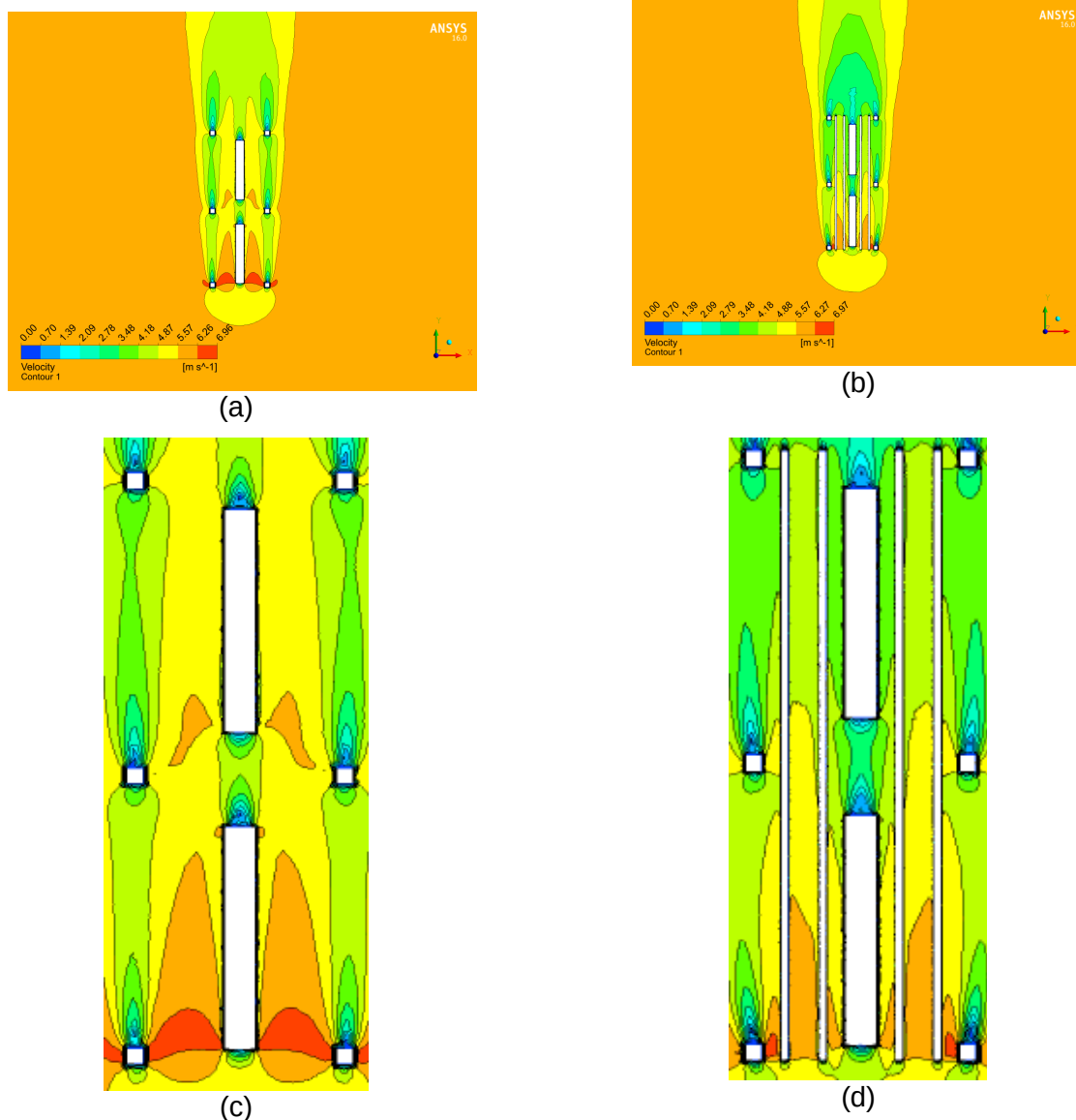


Figure 4 - Velocity contour for the case where the wind is blowing from South (yellow). North is pointing up. Velocity contours in the module for the case where only the large-scale objects were considered (a). Velocity contour for the case where the small-scale geometry was considered (b). Detailed view of the area in the module where the largest vessels are located considering the large-scale objects only (c) and the small-scale objects (d).

Figure 5 shows the local velocity normalised by the bulk velocity in the module for the wind blowing in the x-direction. Three different arrangements were considered in the analysis. The base case is assumed as the simulation in which only the large-scale objects were addressed. The VBR 7% and 8% cases are those where the volume blockage ratio (VBR) was modelled as 7% and 8% respectively.

Analysis of Figure 5 (a) and Figure 5 (c) shows that the velocity in the module is significantly reduced at specific regions when the small-scale geometry was considered in the analysis. On the other hand there are regions where the velocity was increased when the small geometry was addressed in the simulation. Overall the wind pattern in the module varies significantly when the small-scale is considered. It is important to notice that for the cases where the VBR was 7% and 8% a similar velocity pattern in the module is achieved.

Figure 5 (b) and (d) shows that the turbulence intensity increases with the level of detail of the geometry. The cases where the VBR was 7% and 8% presented higher values of the turbulent kinetic energy

(normalised by the square of the bulk velocity) than the cases where the large-scale geometry was considered.

Analysis of Figure 5 (b) and (d) also shows that at specific regions the turbulence intensity is similar to the base case. As previously observed in the plots where the normalised velocity was discussed a different flow pattern is noticed when the small-scale geometry is resolved in the mesh.

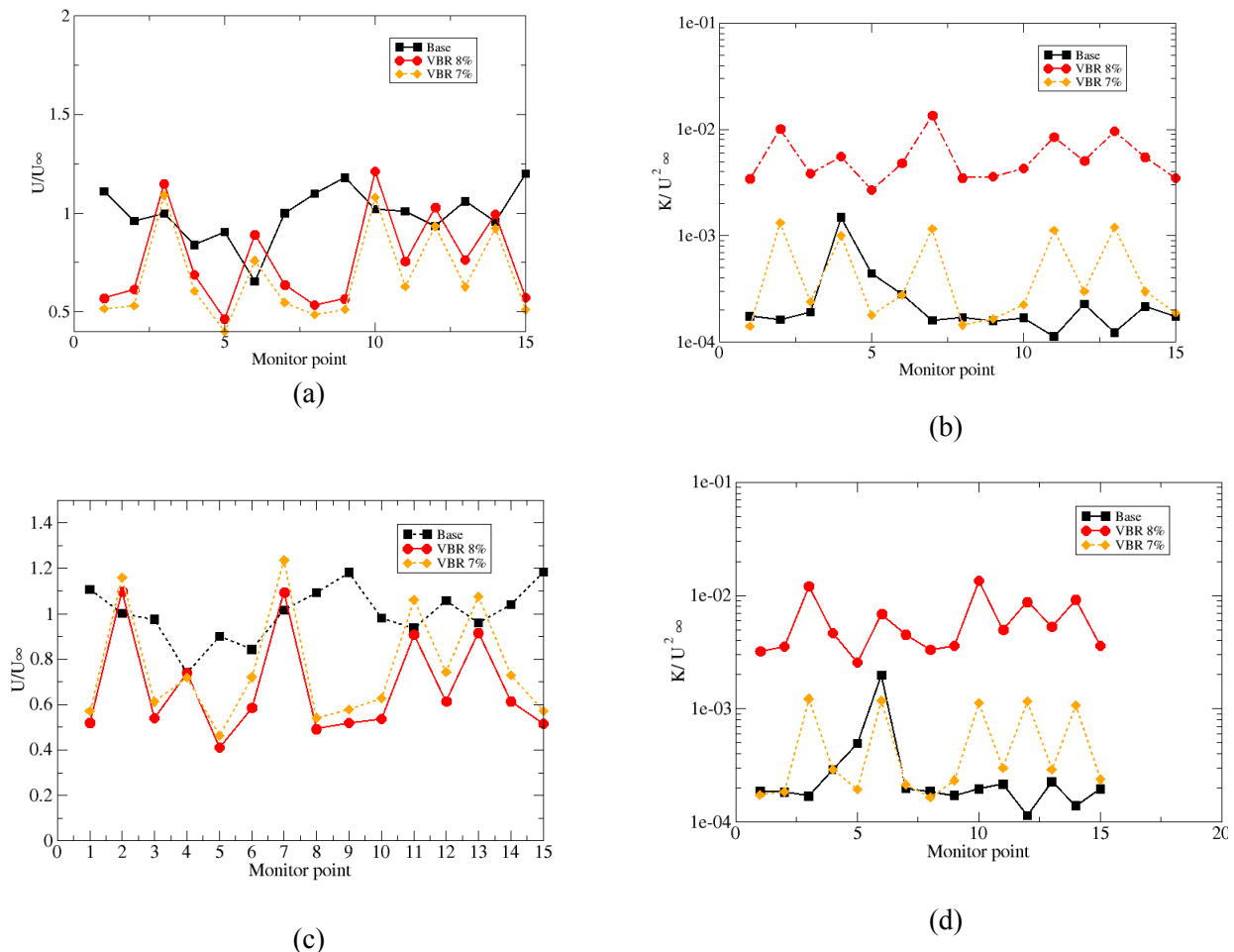


Figure 5 – Non-dimensional velocity (a) and turbulence intensity (b) for wind blowing from East (xhigh) and Non-dimensional velocity (c) and turbulence intensity (d) for wind blowing from West (xlow).

Figure 6 shows the local velocity normalised by the bulk velocity in the module for the wind blowing in the x-direction. Three different arrangements were considered in the analysis. The base case is assumed as the simulation in which only the large-scale objected were addressed. The VBR 7% and 8% cases are those where the volume blockage ratio (VBR) was modelled as 7% and 8% respectively.

Analysis of Figure 6 (a) and Figure 6 (c) shows that the velocity in the module is significantly reduced at specific regions when the small-scale geometry was considered in the analysis. The velocity pattern in the module varies significantly when the small-scale is considered. For the wind blowing in the y-direction there was no significant difference between the base case and the case where the VBR was 7%. It was noticed, however, significant drop of the velocity in the module for the case where the VBR was 8%.

Figure 6 (b) and (d) shows that the turbulence intensity increases with the level of detail of the geometry. The cases where the VBR was 7% and 8% presented far higher values of the turbulent kinetic energy (normalised by the square of the bulk velocity) than the cases where the large-scale geometry was considered.

Analysis of Figure 6 (b) and (d) also shows that at specific regions the turbulence intensity is similar to the base case.

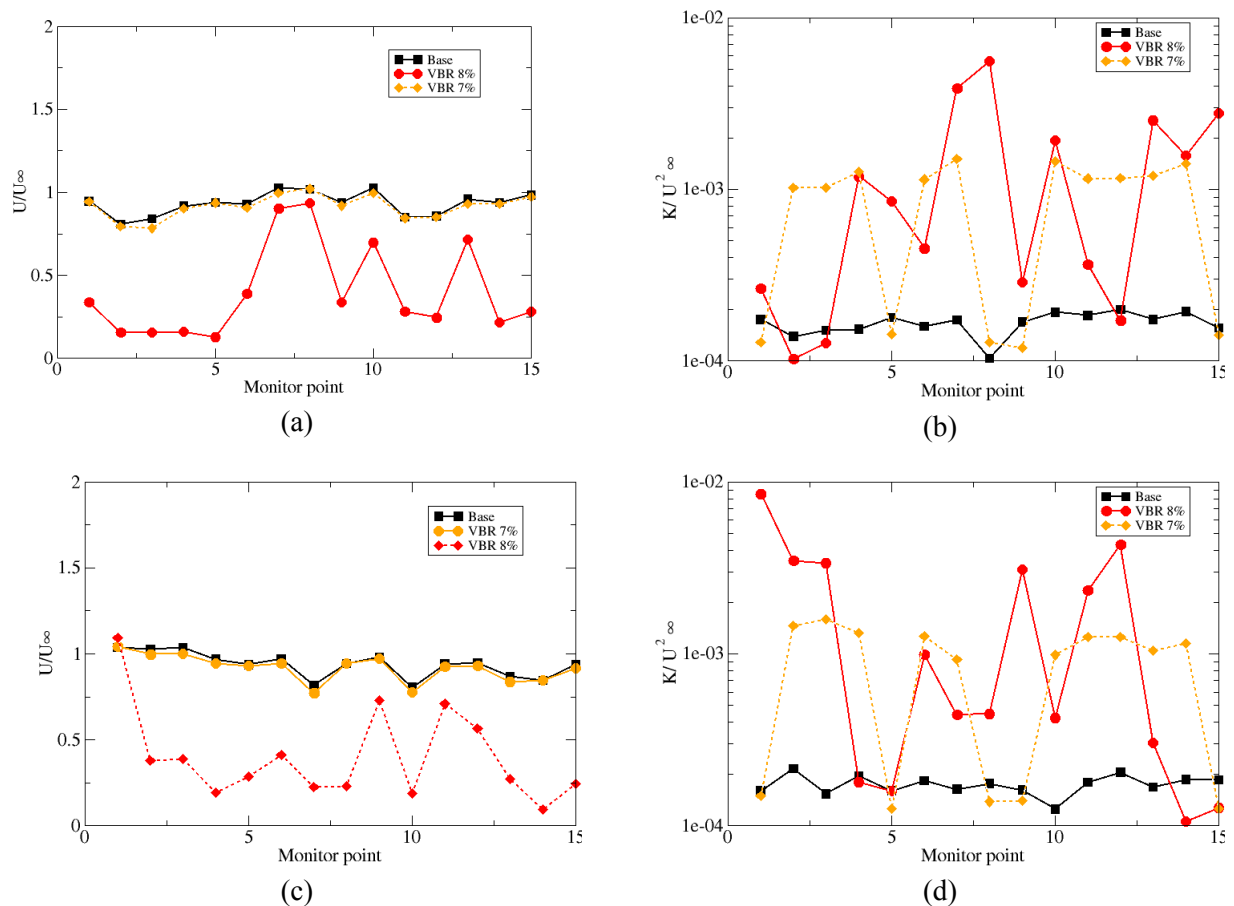


Figure 6 – Non-dimensional velocity (a) and turbulence intensity (b) for wind blowing from North (yhigh) and Non-dimensional velocity (c) and turbulence intensity (d) for wind blowing from South (yflow).

Gas dispersion in the module

The gas dispersion analysis considered one leak location pointing down and 4 wind directions namely; North, South, West and East. The base case considered the large-scale geometry only. The VBR (Volume Blockage Ratio) cases considered 8% of blockage.

Analysis of Figure 7 shows that the cases considering the small-scale geometry led to the flammable cloud sizes smaller than the cases where the large-scale geometry was considered. The different velocity field pattern caused by the ventilation in the module seems to be enhancing the dilution of the gas cloud.

For the cases considered in the current analysis the factor was as close as 2.0. It is important to bear in mind that in some cases the difference was even higher than 2.0.

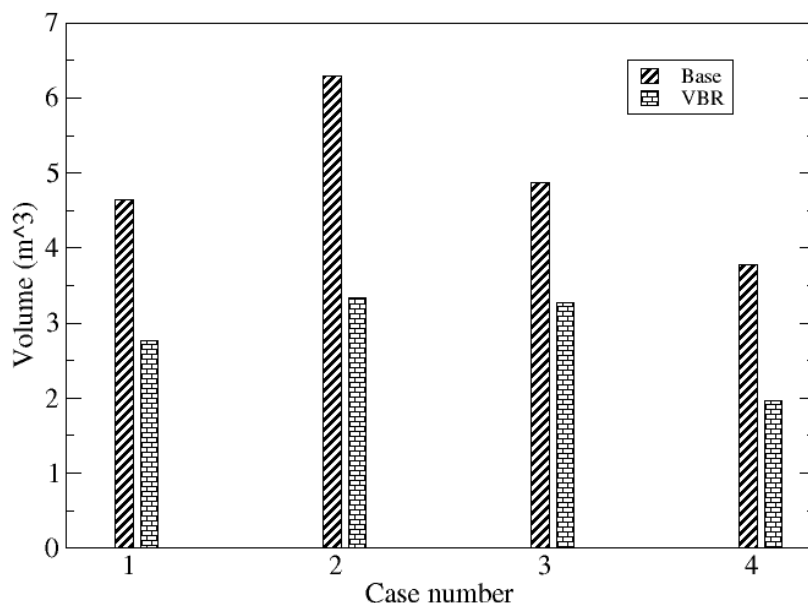


Figure 7 – Flammable gas cloud volume for wind blowing from South (case 1), North (case 2), West (case 3) and East (case 4). The base case considered the large-scale geometry only. The cases labeled as VBR (Volume Blockage Ratio) were those where the ratio was set to 8%.

CLOSING REMARKS

It has been observed that the small-scale geometry plays a significant role in the ventilation inside the module considered in the current analysis. Significant velocity field patterns as well as turbulence intensity were observed for cases the VBR was increased.

The cloud volumes calculated when considering the small scale geometries were smaller than the cases where only the large-scale geometry was considered. Higher mixing caused by small scale geometry seems to dilute the gas cloud more effectively than the base case addressed in scope of this work. It is important to mention that it seems that at early stages of design or for a quick analysis neglecting the small-scale geometry might pay off since the base case analysis presented a conservative flammable gas cloud when compared with the VBR cases.

It is not clear, however, if the small scale geometry is really essential when modelling gas dispersion. Further studies are required. It is necessary to investigate the influence of the leak direction as well as the wind speed in order to confirm the trend observed in the analysis conducted in this preliminary research.

REFERENCES

- [1] I. Ahmed et al. / Journal of Loss Prevention in the Process Industries 44 (2016) 594-600
- [2] Petrobras. Safety Philosophy. Guidelines of E & P Production Engineering. DR-ENGP-II.3
- [3] S.S.V. Vianna, R.S. Cant. Modified porosity approach and laminar flamelet modelling for advanced simulation of accidental explosion. *Journal of Loss Prevention in the Process Industries*, 23, 3–14, (2010).
- [4] V.D. Moreira; R.G.Santos; T.D.Ferreira ; S.S.V.Vianna. “Towards Gilbert-Johnson-Keerthi Algorithm Distance for Fluid Flow Numerical Analysis”. *23rd ABCM International Congress of Mechanical Engineering*. Rio de Janeiro, BR (2015).