Wind and Structures, *Vol. 25, No. 6 (2017) 537-549*
DOI: https://doi.org/10.12989/was.2017.25.6.537

Comparative study of turbulent flow around a bluff body by using two- and three-dimensional CFD

Muhammet Ozdogan, Bilal Sungur, Lutfu Namli* and Aydin Durmus

Department of Mechanical Engineering, Ondokuz Mayis University, 55139 Samsun, Turkey

(Received August 24, 2017, Revised November 20, 2017, Accepted November 29, 2017)

Abstract. In this study, the turbulent flow around a bluff body for different wind velocities was investigated numerically by using its two- and three-dimensional models. These models were tested to verify the validity of the simulation by being compared with experimental results which were taken from the literature. Variations of non-dimensional velocities in different positions according to the bluff body height were analysed and illustrated graphically. When the velocity distributions were examined, it was seen that the results of both two- and three-dimensional models agree with the experimental data. It was also seen that the velocities obtained from two-dimensional model matched up with the experimental data from the ground to the top of the bluff body. Particularly, compared to the front part of the bluff body, results of the upper and back part of the bluff body are better. Moreover, after comparing the results from calculations by using different models with experimental data, the effect of multidimensional models on the obtained results have been analysed for different inlet velocities. The calculation results from the two-dimensional (2D) model are in satisfactory agreement with the calculation results of the three-dimensional model (3D) for various flow situations when comparing with the experimental data from the literature even though the 3D model gives better solutions.

Keywords: two- and three-dimensional model; turbulent flow; bluff body; RNG k-ε model; computational fluid dynamics (CFD); wind flow; turbulence modelling

1. Introduction

Energy requirements continue to increase significantly. It is important not only to produce more energy but also to use energy efficiently. The investigations in the literature show that a significant portion of the world's energy consumption is used in the buildings. The aerodynamics of the buildings is the motivation for the great interest in bluff body flows. Therefore, numerical modelling and measuring of bluff body aerodynamics have found significant interest in engineering sciences. In this context, the prediction of the wind flow patterns in an urban is very important to design economic and environmental buildings (Shao *et al.* 2012). The flow patterns due to the presence of obstacle have been researched widely with wind tunnel experiments. The experiments are very expensive and take a long time. In recent years, computational fluid dynamics (CFD) technique has been used to calculate engineering problems such flow, combustion, heat transfer etc. CFD simulations especially turbulent flows are relatively inexpensive and can be

∗Corresponding author, Ph.D., E-mail: lnamli@omu.edu.tr

http://www.techno-press.org/?journal=was&subpage=7 ISSN: 1226-6116 (Print), 1598-6225 (Online)

executed in a short period of time by utilizing variety turbulence models. Although the numerical solution of the flow around the building takes less time than the experimental methods, this time, especially in the complex exterior building geometries, causes a considerable waste of time in the three-dimensional analysis. If the computation in numerical analysis can be performed in two-dimensions using symmetry, the wasted time is minimized. However, at this stage, it is useful to determine how much error is made in the computation by the two-dimensional analysis compared to the three-dimensional analysis. In the literature, a large number of two-dimensional analyses related to the exterior geometry of the buildings and the pitched roof were performed both two-dimensional and three-dimensional. The usage of CFD, flow around a bluff body researched by many authors (Tsuchiya *et al*. 1997, Irtaza *et al*. 2013, Nitatwichit *et al*. 2008, Yazid and Sidik 2013, Ntinas 2014, Ai and Mak 2015, Mochida *et al*. 2002, Bazdidi-Tehrani *et al*. 2013, Ai and Mak 2013, Tominaga *et al*. 2015, Vardoulakis *et al.* 2011, Lien *et al.* 2004, Mavroidis *et al.* 2015, Gao and Chow 2005, Ozmen *et al.* 2016). Tsuchiya *et al.* (1997) developed a new k-ε model with the name of MMK. They used this new model for modelling two-dimensional square, cube and flow around low-rise buildings and then they compared these results with standard k-ε model and experimental data. Irtaza *et al.* (2013) used standard k-ε model, RNG k-ε model, Realizable k-ε model, Reynolds Stress Method and Large Eddy Simulation model to determine the most appropriate turbulence model for wind engineering calculations and compared these results with experimental data. Nitatwichit *et al*. (2008) numerically investigated the effect of geometries and orientation to the air flow distribution around a school building. Yazid and Sidik (2013) modelled the flow around a cube with two-equation turbulence model. They determined the two-equation model's accuracy by comparing the model results with experimental data. They reported that all two-equation turbulence models could predict the separation point but couldn't predict the reattachment lengths near the walls of the cube. Shao *et al*. (2012) aimed to determine the wind flow around high rise buildings using various non-linear k-ε models. They said that Craft model provided good agreement with experimental measurements. Mochida *et al.* (2002) modelled the flow around the high rise buildings with different numerical methods and they validated the model results. Tominaga *et al*. (2015) investigated the air flow around gable roof buildings with different roof pitches (3:10, 5:10, 7.5:10) by wind tunnel experiments and CFD. They reported that, in general, the velocity values in the direction of the main flow, both the experimental results and the CFD results differ by less than 15% on average and these differences became up to 30% in the points behind the building. Vardoulakis *et al.* (2011) numerically modelled the wind and turbulence areas around two surface-mounted cubes with different dimensions and wall roughness, then compared with experimental data. They said that the models predicted well general flow characteristics around single-block buildings but at the near stagnation points in the windward side over-predictions of the turbulent kinetic energy were obtained. Lien *et al*. (2004) calculated the disturbed flow through and over a two-dimensional array of rectangular buildings numerically by using the steady-state Reynolds-averaged Navier-Stokes equations. Gao and Chow (2005) studied flow structure towards a cube with CFD and they compared some RANS predicted results at different upstream air velocities. They also investigated the flow separation at the corner above the top of the cube, separation levels and reattachment lengths. Ozmen *et al*. (2016) researched experimentally and numerically the turbulent flow regions with gabled roofs which have 15^{o}, 30^{o} and 45^{o} pitch angles in the atmospheric boundary layer.

Numerical modelling of the bluff body aerodynamics is important in engineering applications. The motivation for the great interest in bluff body flows depends on the complexity of basic physics phenomena related to these flows and the wide applicability such as buildings. Bluff body flows are especially challenging since they are highly three-dimensional and usually associated with a wide range of flow regimes such as stagnation, separation, recirculation, strong shear layers and unsteady vortex shedding (Joubert *et al.* 2015). These flow properties are more challenging to model more accurately. Besides, the numerical modelling of the building can be carried out by two-dimensionally and three-dimensionally. Generally, in the literature, many studies were about the effect of turbulence models, the geometry of the bluff body (aspect ratio, roof pitches, etc.). 2D modelling is faster and practical compared 3D modelling. On the other hand, the third direction effect on bluff body ignored by modelling with 2D models. The main motivation of the current study is to investigate the third direction effect on the flow around a bluff body at different wind velocities. For this purpose, in this study, the models of 2D and 3D in the problems of turbulent flow around a bluff body were compared. For this purpose, the airflow around a bluff body was numerically calculated in both two-dimensional (2D) and three-dimensional (3D) models by using RNG k-ε turbulence model, and then the results obtained from the calculations were compared with the data taken from CEDVAL database (2006) for the validation of chosen models. Also, the effect of the wind velocities to the flow pattern was investigated numerically in both 2D and 3D models. Models were constructed with a scale of 1:200 and in this context the bluff body height (H) is 0.08 m. The calculations were made both in the 2D and the 3D models with the average wind velocities of 5 m/s, 7 m/s and 9 m/s.

2. Materials and methods

The air flow around a bluff body was numerically calculated in both two-dimensional (2D) and three-dimensional (3D) models by using RNG k-ε turbulence model. For comparing the results from the calculations, CEDVAL database was used in this study. At the Hamburg University, extensive researches of the flow field around the bluff body with simple configurations were done in the BLASIUS wind tunnel to ensure well-documented validation data through CEDVAL database. The CEDVAL database was chosen due to the availability of data for validation the flow structure. To verify the simulation results, an experimental dataset for a single cube case (A1-2) from CEDVAL project was adopted.

In the simulation of flow problems, solutions were made with mass, momentum and other scalar conservation equations. These problems were solved using appropriate boundary conditions. For turbulent, steady-state flow with constant property fluids, time-averaged continuity and momentum equations were expressed as follows

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$U_j \frac{\partial U_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\nu \frac{\partial U_i}{\partial x_j} - \overline{u_i u_j} \right) - \frac{1}{\rho}\frac{\partial P}{\partial x_i} \tag{2}$$

In these equations, U_i , is the mean-velocity vector, P is the mean static pressure, ρ is the fluid density and ν is the fluids kinematic viscosity. Reynolds stresses ($\overline{u_i u_j}$) in momentum conservation equation is defined by

$$-\overline{u_i u_j} = \nu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3}\delta_{ij} k \tag{3}$$

where ν_t is turbulent viscosity and defined as

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

where k is the turbulent kinetic energy and ε is turbulent dissipation rate.

There are lots of turbulent models to simulate the turbulent flow and RNG k-ε model is one of them. Tominaga *et al.* (2009) used various type turbulent models and get the best results with RNG *k-ε* turbulence model. Due to this reason, RNG *k-ε* turbulence model was used in this study. RNG *k-ε* model is obtained from the instantaneous Navier-Stokes equations by using Renormalization group methods. k and ε are obtained from the following transport equations which were described for RNG model.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) - C_{2\varepsilon}\rho \frac{\varepsilon^2}{k} + C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k + C_{3\varepsilon} G_b \right) - R_\varepsilon + S_\varepsilon \tag{6}$$

where, α_k and α_ε represent the reverse effective Prandtl numbers of k and μ_{eff} , represents the effective (turbulent) viscosity. The constant values used in the RNG k-ε model are given in Table 1 (Fluent 2006).

For numerical simulation, a schematic diagram of the computational domain, dimensions (taken from CEDVAL database) and boundary conditions are shown in Fig. 1. While creating the solution area, many studies in the literature (Ai and Mak 2015, Yazid and Sidik 2013 etc.) have been considered and very small changes made in x and y-direction in accordance with the experimental dimensions. In this figure, all walls are defined with the assumption of the no-slip wall shear condition. Velocity inlet conditions are defined as velocity profile shown in Fig. 2 and outlet condition is defined as pressure outlet. The upper surface of the domain is defined as symmetry boundary condition. Models were constructed with a length scale of 1:312.5 and in this context the bluff body height (H) is 0.08 m.

Table 1 The constants of RNG k-ε model (Fluent 2006)

Constant	Value
$C_{1\varepsilon}$	1.42
$C_{2\varepsilon}$	1.68
C_μ	0.0845
η_0	4.38
β	0.012
α_k	1.393
α_ε	1.393

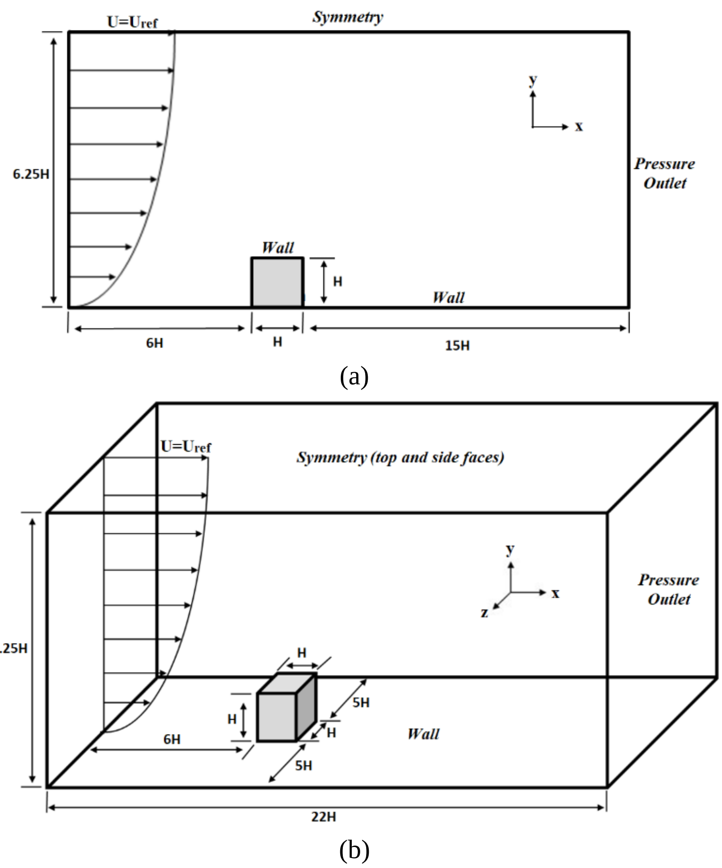

Fig. 1 Computational domain (a) 2D view and (b) 3D view

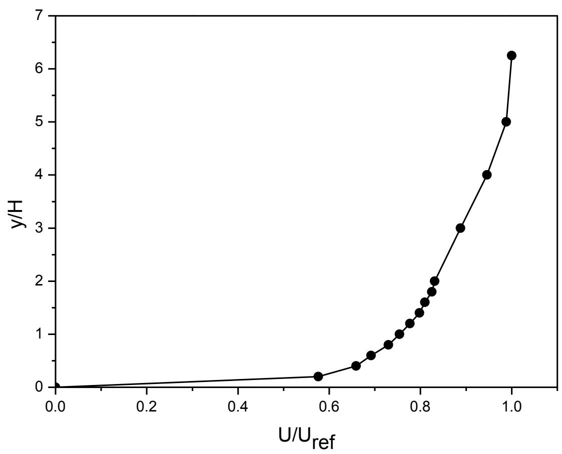

Fig. 2 Velocity profile defined at the domain inlet (CEDVAL database 2006)

The calculations were made both in the 2D and the 3D models with the average wind velocities of 5 m/s, 7 m/s and 9 m/s. Results of these calculations compared with each other. For the average velocity of 5 m/s, numerical results verified with experimental data (CEDVAL database 2006).

In this study, Fluent 6.3 software was used to simulate the air flow. Gambit program was used to mesh the whole solution domain. The mesh structure of 2D and 3D geometries were given in Fig. 3 and in Fig. 4, respectively. As shown in these figures, the structured quadratic mesh was used in all cases, and the fine grid was used in critical regions (especially around the bluff body). The effect of mesh structure used in calculations has been examined separately and all computations have been conducted by using appropriate mesh structure. Various mesh sizes (the values of these meshes were given in Table 2) were tested to determine the grid-independent solution.

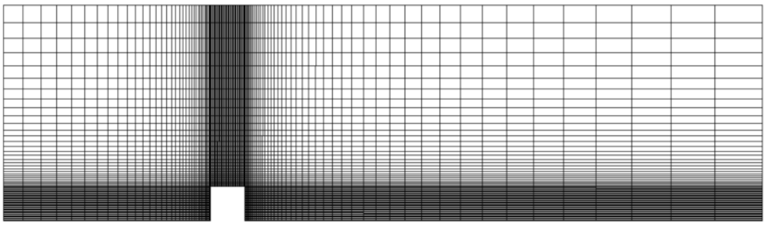

Fig. 3 Computational mesh structure in the 2D modelling

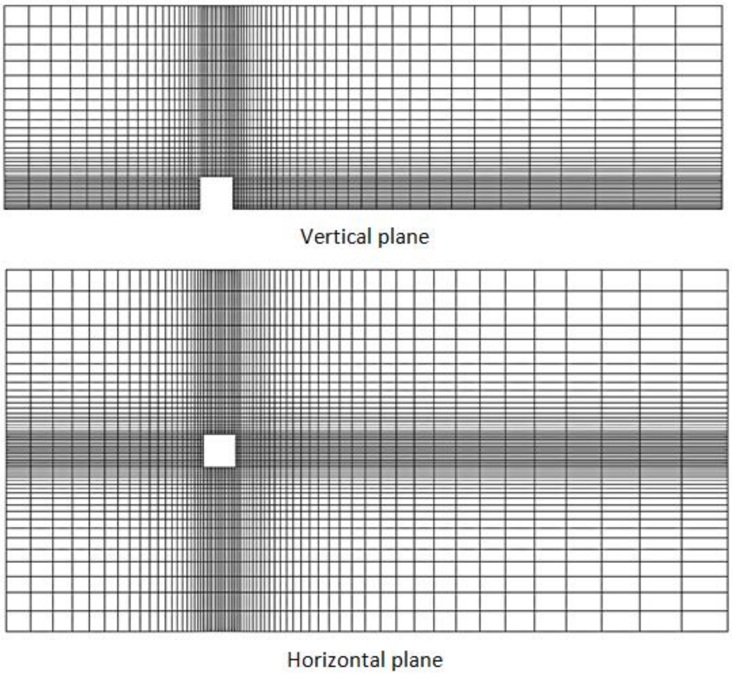

Fig. 4 Computational mesh structure in the 3D modelling

Table 2 Mesh configurations of 2D and 3D models

Mesh Type	No. of Cells in 2D	No. of Cells in 3D
Mesh 1	3650	172000
Mesh 2	6800	550250
Mesh 3	15800	1122930
Mesh 4	34600	1952000

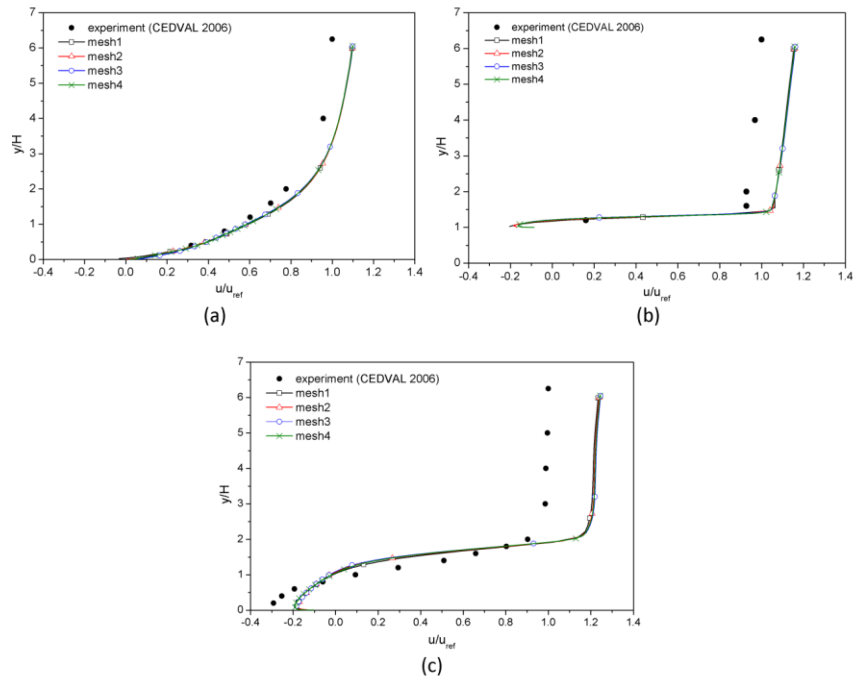

Fig. 5 Grid independence test for 2D modelling (a) x=-1.5H, (b) x=0 and (c) x=2.5H

In all calculations, SIMPLE algorithm was used to solve the discretized equations of pressure-velocity coupling. The standard scheme was used to discretize pressure and the second-order upwind scheme was used to discretize momentum, turbulent kinetic energy and turbulent dissipation rate. For all dependent variables, the convergence criteria for the residuals was set to 10^{-5}.

3. Results and discussion

3.1 Grid independence test

The aim of this study is to compare the models of 2D and 3D in the problems of turbulent flow around a bluff body. Firstly, to analyse the meshing accuracy, the grid independence tests were done by matching the numerical results to experimental data from CEDVAL database (2006) both in the 2D and the 3D models. In Fig. 5, the grid independence results in term of velocity in the x-direction were given for 2D modelling. As shown in Fig. 5, all the meshes were almost coincident at all the x-positions in the 2D model and for this reason, the coarse mesh (mesh 2) was used in this study to save time.

In Fig. 6, the grid independence results in term of velocity in the x-direction were given for 3D modelling. As shown in Fig. 6, it can be seen that the best mesh is the fine mesh (mesh 3) at all the x-positions. Decreasing the mesh size from mesh 4 to mesh 1, it can be seen that the accuracy of the mesh gets worsen especially at the position of x=2.5H. Therefore, during analyses by using the 3D model, the mesh 3 was used in this study to save time which is more important in the 3D analyses as there are many nodes in the solution region.

3.2 Comparison of the 2D and 3D models

In this section, validation and comparison of the 2D and 3D models with an experimental data were realized. Additionally, comparison of velocities in both 2D and 3D at different x-positions and contour of x-velocities distribution at different inlet wind velocities were examined.

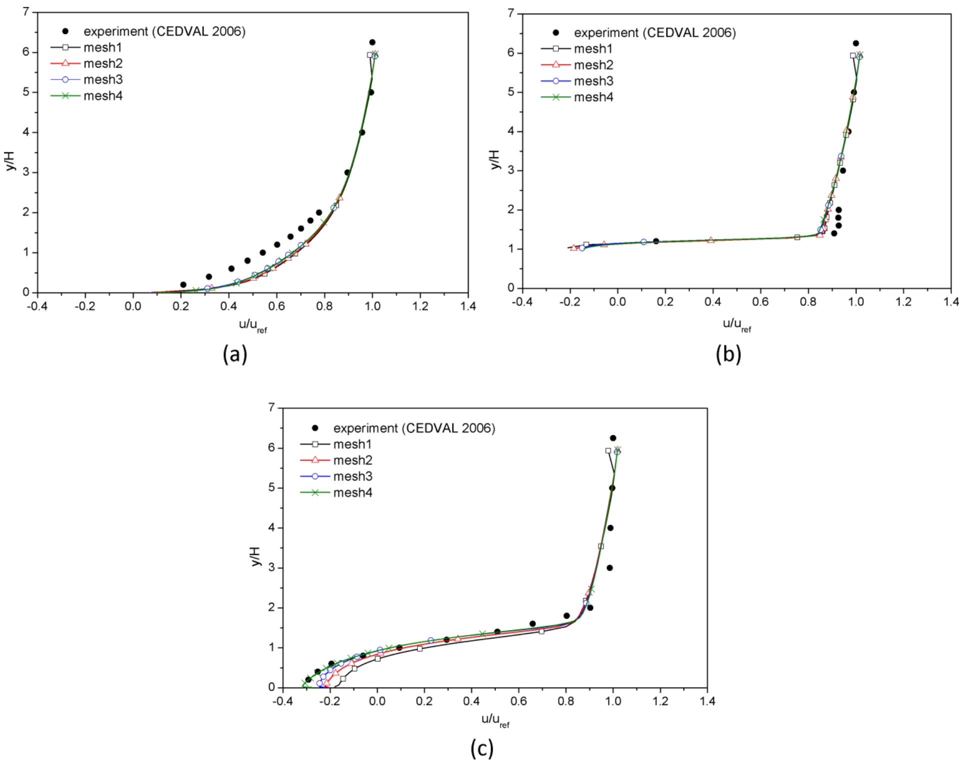

Fig. 6 Grid independence test for 3D modelling (a) x=-1.5H, (b) x=0 and (c)x=2.5H

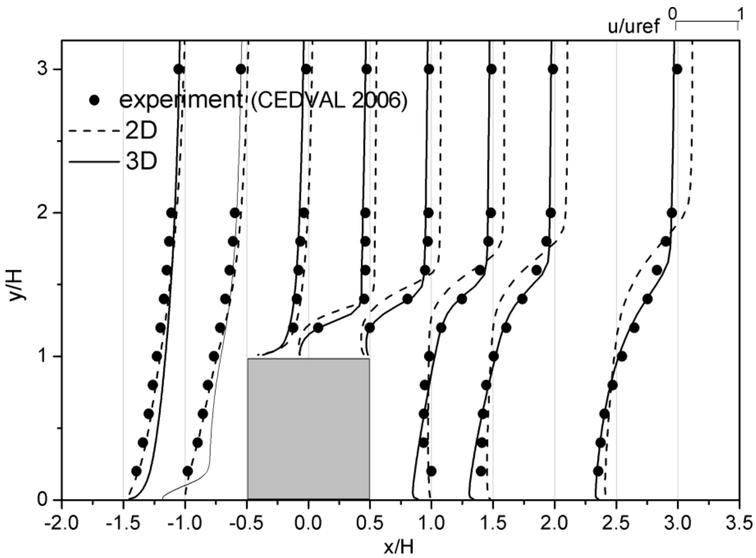

Fig. 7 Comparison of experimental and numerical results (2D and 3D)

Table 3 Average errors (%) of 2D and 3D models at different x-locations

Dimension	-1.5H	0	2.5H
2D	9.54	13.39	16.23
3D	4.45	1.13	3.21

Fig. 7 was given the comparison the wind velocities obtained from numerical computations with the experimental values at both the back and the front of the bluff body. As seen in Fig. 7, it can be observed that both the results obtained from two and three-dimensional models agree with experimental data in the aspect of the velocity profiles. It has also been seen that the velocities obtained from the two-dimensional model match up with the experimental data from the ground to the top of the bluff body. However, with increasing the height of the solution domain (>H), the velocity values differ from each other in the 2D model. The experimental data and the results obtained from the three-dimensional model are in good agreement, especially after the top of the bluff body (>H). Compared with the front part of the bluff body, results of the back part of the bluff body are better in the 3D model.

The errors quantitively compared by 2D and 3D models according to experimental data and the results were given in Table 3. In all x-location, the 3D model has better results than the 2D model. Especially, 3D models gave very close results to the experiments at x=0 location and in other x-locations, satisfactory results were obtained.

In Fig. 8, at different x-positions, the change of the wind velocities with y-position was given in both the 2D and the 3D models. In Fig. 8, V5, V7 and V9 denote the wind velocity magnitudes having values of 5, 7 and 9 m/s, respectively. It can be seen from the figure that at x=-1.5 H in the 2D model, the increasing wind velocity affects the flow nearly to the value of y~0.25 H, after this value it doesn't affect the flow structure very much, in the 3D model at the value of y~0.55-0.80 H the increasing wind velocity affects the flow. At the position of x=0, increasing wind velocity doesn’t affect the flow structure very much in both models. At x=2.5 H, the magnitudes of velocities increase with the increasing wind velocities until the value of y~1.1-1.6 H in the 2D model. However, the magnitudes of the velocities increase with the increasing wind velocities up to the value of y~1.5 H in the 3D model.

When comparing the 2D with the 3D models, at x=-1.5 H, u/u_{ref} values are higher in the 3D model than the 2D model until to the value of y~1.6 H, after this value in the 2D models, u/u_{ref} values are higher than the 3D model. At x=0, the curves are close to each other in both models until to the value of y~1.25 H, after this value in the 2D models u/u_{ref} values are higher than those in the 3D models. At x=2.5 H, u/u_{ref} values are higher in the 3D model than the 2D model until the value of y~1.7H, after this value in the 2D models, u/u_{ref} values are higher than those in the 3D model. Generally, in both 2D and 3D models the trends of the velocities were similar as can be seen in the Fig. 8.

In Fig. 9, the velocity distribution contours for three different wind velocities were given in both the 2D and the 3D models. In Fig. 9, the velocities for different wind velocity magnitudes (V=5, 7 and 9 m/s) are plotted as contour graphics. It can be seen from the figure that in the 2D models the reattachment point in the back of the bluff body and the maximum velocities in the solution domain were overestimated for all wind velocities.

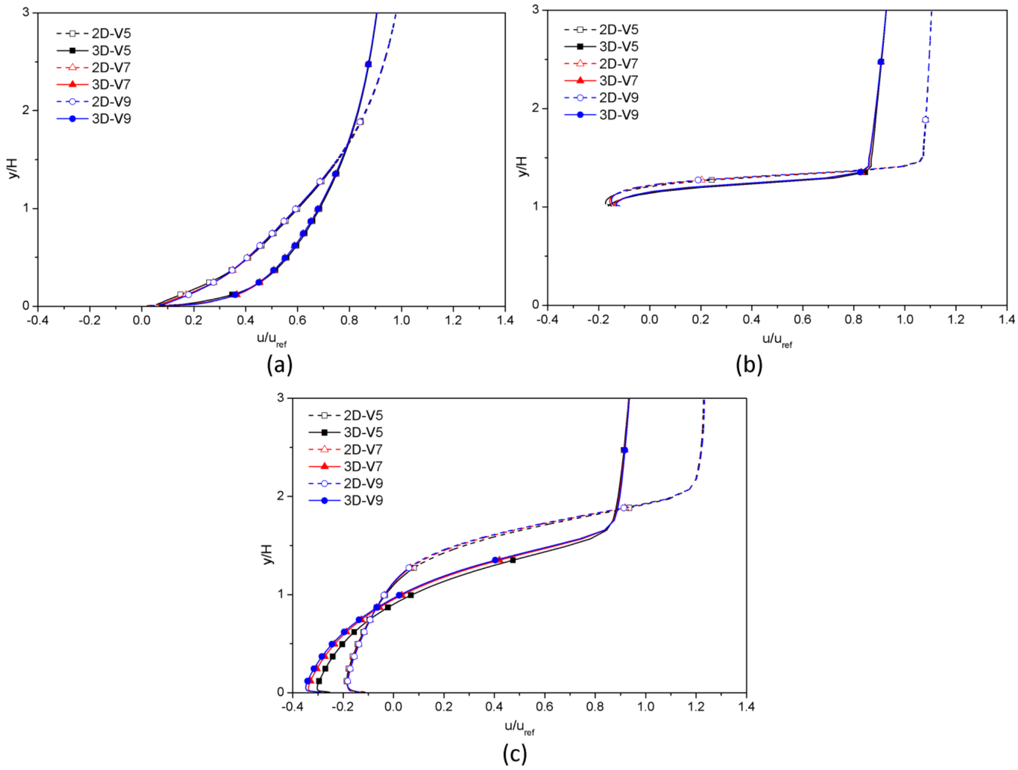

Fig. 8 Comparison of x-velocities in both 2D and 3D at different x positions (a) x=-1.5H, (b) x=0 and (c) x=2.5H

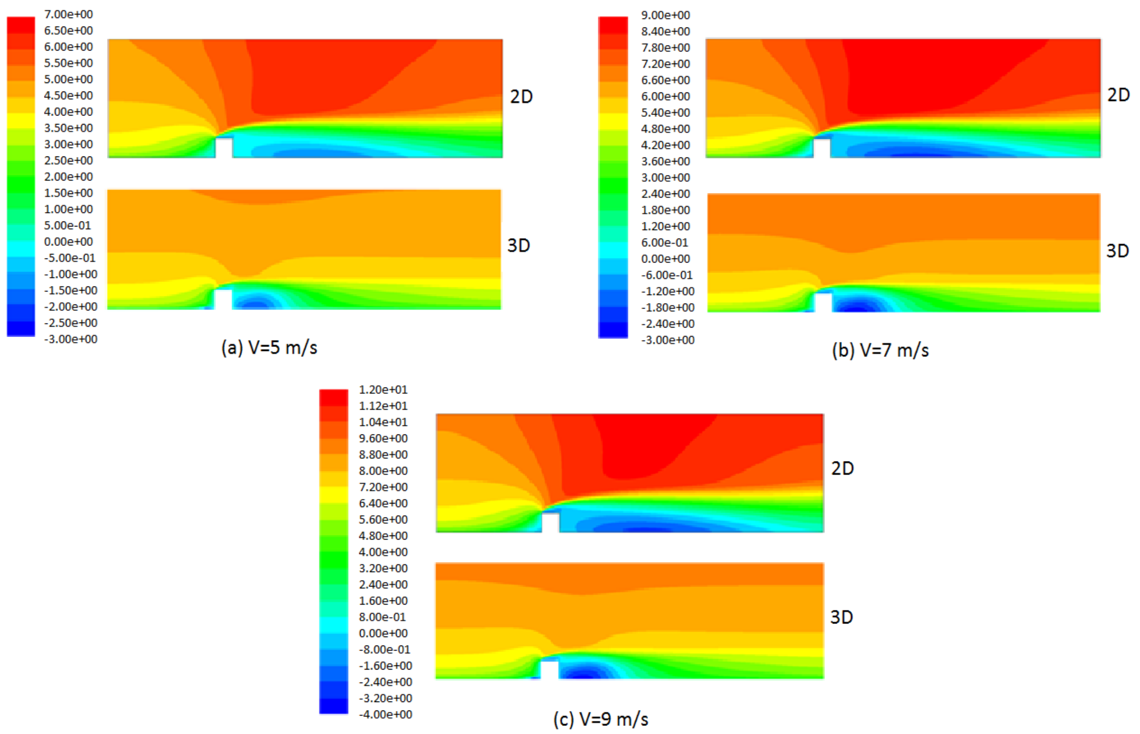

Fig. 9 Velocities distributions for different wind velocities

However, the flow patterns in front of the bluff body are almost similar in both 2D and 3D models for individual velocities. It can be declared that the results obtained from 3D models for all velocities are better than that of the 2D models in the back of the bluff body. However, in front of the bluff body, both 2D model and 3D model have given almost similar numerical results for all velocities. Moreover, in the 3D model, as the flow approaches the bluff body it slows down in the upstream stagnation region and then accelerates as it moves around the obstacle. Vertical flow movement is visible which forms a funnel shape where velocities converge in the wake regions. It can be seen that the side-wall recirculation region is larger at mid-height than at the base and tip of the bluff body. There is also a small recirculation present near the base upstream of the bluff body. This recirculation is expected to be related to the upstream near-wall horseshoe vortex noticed in numerous studies in the literature (Gao and Chow 2005, Irtaza *et al*. 2013, Joubert *et al*. 2015). As shown in Fig. 9, these effects are not very pronounced in the 2D model.

4. Conclusions

In this study, the flow characteristics around the bluff body were numerically calculated in both 2D and 3D, and then these results were compared with the data taken from the literature for validation. Additionally, the effect of the wind velocities on the flow structure was investigated numerically. It was observed that both the 2D and the 3D models results agreed with experimental data even though the 3D model has given better solutions. It was also seen that the velocities obtained from the 2D model matched up well with the experimental data from the ground to the top of the bluff body. In the 3D model, the experimental data and the results obtained model were in good agreement, especially after the top of the bluff body. The increasing wind velocities didn't affect the flow structure very much in the 2D model. In the 3D model, the increasing wind velocities raised the value of u/u_{ref}, especially at the position of x=2.5H and near to the ground. Consequently, 2D models could be preferable because of the simplicity, fastness and also because the results of 2D models were in satisfactory agreement with experimental data. But to see the effect of the third dimension and to get more sensitive solutions, using 3D models will be better.

References

Ai, Z.T. and Mak, C.M. (2013), "CFD simulation of flow and dispersion around an isolated building: Effect of inhomogeneous ABL and near-wall treatment", *Atmos. Environ.*, **77**, 568-578.

Ai, Z.T. and Mak, C.M. (2015), "Large-eddy Simulation of flow and dispersion around an isolated building: Analysis of influencing factors", *Comput. Fluids*, **118**, 89-100.

Bazdidi-Tehrani, F., Ghafouri, A. and Jadidi, M. (2013), "Grid resolution assessment in large eddy simulation of dispersion around an isolated cubic building", *J. Wind Eng. Ind. Aerod.*, **121**, 1-15.

CEDVAL Database (2006), Hamburg University. http://mi.uni-hamburg.de/Data-Sets.432.0.html.

FLUENT (2006), FLUENT 6.3 version. Fluent User's Guide. Fluent Incorporated.

Gao, Y. and Chow, W.K. (2005), "Numerical studies on air flow around a cube", *J. Wind Eng. Ind. Aerod.*, **93**, 115-135.

Irtaza, H., Beale, R.G., Godley, M.H.R. and Jameel, A. (2013), "Comparison of wind pressure measurements on Silsoe experimental building from full-scale observation, wind-tunnel experiments and various CFD techniques", *Int. J. Eng. Sci. Technol.*, **5**(1), 28-41.

Joubert, E.C., Harms, T.M. and Venter, G. (2015), "Computational simulation of the turbulent flow around a surface mounted rectangular prism", *J. Wind Eng. Ind. Aerod.*, **142**, 173-187.

Lien, F.S., Yee, E. and Cheng, Y. (2004), "Simulation of mean flow and turbulence over a 2D building array using high-resolution CFD and a distributed drag force approach", *J. Wind Eng. Ind. Aerod.*, **92**, 117-158.

Mavroidis, I., Andronopoulos, S., Venetsanos, A. and Bartzis, J.G. (2015), "Numerical investigation of concentrations and concentration fluctuations around isolated obstacles of different shapes. Comparison with wind tunnel results", *Environ. Fluid Mech.*, **15**, 999-1034.

Mochida, A., Tominaga, Y., Murakami, S., Yoshie, R., Ishihara, T. and Ooka, R. (2002), "Comparison of various k-ε models and DSM applied to flow around a high-rise building –report on AIJ cooperative project for CFD prediction of wind environment", *Wind Struct.*, **5**(2-4), 227-244.

Nitatwichit, C., Khunatorn, Y. and Tippayawong, N. (2008), "Computational analysis and visualization of wind-driven naturally ventilated flows around a school building", *Maejo Int. J. Sci. Technol.*, **2**(1), 240-254.

Ntinas, G.K., Zhangb, G., Fragos, V.P., Bochtis, D.D. and Nikita-Martzopoulou, C.H. (2014), "Airflow patterns around obstacles with arched and pitched roofs: Wind tunnel measurements and direct simulation", *Eur. J. Mech. B/Fluids*, **43**, 216-229.

Ozmen, Y., Baydar, E. and van Beeck, J.P.A.J. (2016), "Wind flow over the low-rise building models with gabled roofs having different pitch angles", *Build. Environ.*, **95**, 63-74.

Shao, J., Liu, J. and Zhao, J. (2012), "Evaluation of various non-linear k-ε models for predicting wind flow around an isolated high-rise building within the surface boundary layer", *Buildi. Environ.*, **57**, 145-155.

Tominaga, Y. and Stathopoulos, T. (2009), "Numerical simulation of dispersion around an isolated cubic building: Comparison of various types of k–ε models", *Atmos. Environ.*, **43**, 3200-3210.

Tominaga, Y., Akabayashi, S.I., Kitahara, T. and Arinami, Y. (2015), "Air flow around isolated gable-roof buildings with different roof pitches: Wind tunnel experiments and CFD simulations", *Build. Environ.*, **84**, 204-213.

Tsuchiya, M., Murakami, S., Mochida, A., Kondo, K. and Ishida, Y. (1997), "Development of a new k-ε model for flow and pressure fields around bluff body", *J. Wind Eng. Ind. Aerod.*, **67-68**, 169-182.

Vardoulakis, S., Dimitrova, R., Richards, K., Hamlin, D., Camilleri, G., Weeks, M., et al. (2011), "Numerical model inter-comparison for wind flow and turbulence around single-block buildings", *Environ. Model. Assess.*, **16**, 169-181.

Yazid, A.W.M. and Sidik, N.A.C. (2013), "Prediction of the flow around a surface-mounted cube using two-equation turbulence models", *Appl. Mech. Mater.*, **315**, 438-442.

AD