STUDY OF WAKE REGION BEHIND OF ISOLATED CUBIC BUILDING.PART II: NUMERICAL SIMULATIONS

Larisse Suzy Silva de Oliveira, larissesuzy@hotmail.com
Juliana Glicério Dazzi, juliana.dazzi@gmail.com
Mariane Gonçalves Miranda, mariane_cefetes@hotmail.com
Reginaldo Rosa Cotto de Paula, reginaldo@ifes.edu.br
Coordenadoria de Engenharia Sanitária e Ambiental, Ifes, Vitória – ES, Brasil.

Fernanda Capucho Cezana, fe.cezana@gmail.com
Coordenadoria de Matemática, Ifes, Vitória – ES, Brasil.

Marcos Sebastião de Paula Gomes, mspgomes@mec.puc-rio.br
Departamento de Engenharia Mecânica, Puc-Rio, Rio de Janeiro – RJ, Brasil.

Abstract. *This paper presents numerical simulation of wind flow around an isolated cubic building. The numerical simulations were carried out with the standard $\kappa-\varepsilon$ and Omega Reynolds Stress turbulence models. Experimental data from wind tunnel experiments in the neutral boundary layer were used to validate the numerical results. The Reynolds number, based on the height of the building was 1.5×10^4. The main purpose of the present study is to investigate the behavior of a wake region formated behind an isolated cubic building. In addition were determined the length of reattachement points in the roof and near the wake region of the body. The results showed that the DSM turbulence model was able to capture the separtion and the reattachment point on the roof of the building. The numerical predictions of reattachment point in the near wake behind of the cube obtained with both standard $\kappa-\varepsilon$ and Omega Reynolds Stress turbulence models were slightly larger in comparison with experimental data.*

Keywords*: numerical simulations, turbulence models, atmospheric boundary layer, building.*

1. INTRODUCTION

The atmospheric flow near the isolated building is extremely complex, with regions of intense circulation, turbulent movements, strongly anisotropic three-dimensionality (Murakami, 1993). Due to the complexity of the physical phenomena involved, many mathematical models used simplifying hypotheses in order to predict the turbulent flow around a building (Santos, 2000; Tominaga, and Stathopoulos, 2010).

With the advanced of technology and computers have been possible to improve the complexity of mathematical models. Therefore, many studies have been performed using the Computational Fluid Dynamics (CFD) models which are based on the fundamental equations of transport, such as Tsuchiya *et al*. (1997), Gao and Chow (2004) and Tominaga and Stathopoulos (2010). The main difference in these studies was the turbulence models that have been used to predict the turbulent flow around the obstacles.

Baskaran and Kashef (1996) showed the developments of the CFD technique as a powerful tool for prediction of wind flow around a variety of buildings configurations. They examined the following configurations: as single building flow, flow between parallel buildings and flow around multiple building configurations. The authors reported to that CFD method can provide very detailed predictions of air velocities around buildings.

Tominaga *et al*. (2008) presented comparison of CFD results using various revised κ-ε turbulence models (LK model, MMK model and Durbins's revised) and LES applied to flow around a high-rise scale building model placed within the surface boundary layer. The numerical predictions were compared to wind tunnel data. The results showed that all revised κ-ε models to capture the reverse flow on the roof, although it was a little larger than the experimental values. The reattachment length in near wake of building was larger than the experimental data. The reverse flow on the roof given by LES showed to be close with the experiment. The overestimation of reattachment length behind the building was improved in the LES computations. This improvement was mainly because to the periodic velocity fluctuation due to the vortex shedding behind the building was well reproduced in LES.

A review of the literature on this flow problem verifies that the turbulence model κ-ε was widely used to simulate the flow around buildings (Tominaga and Stathopoulos, 2009; Gousseau *et al.* 2011). However, comparisons with experimental data (Murakami, 1993) suggest that the models for the Differential Reynolds Stress (Differential Reynolds Stress Model - DSM) have obtained better results than the κ-ε turbulence models.

The objective of this study was to investigate the behavior of the flow around an isolated cubic obstacle, especially the region at the top of the building and the recirculation region behind the building, using two different turbulence models: standard κ-ε model and Differential Stress Model (DSM) based on the equation of ω.

2. METHODS

2.1 The Wind Tunnel Experiments

The flow considered in this work was investigated in open return wind tunnel with test section of 2.0 m × 0.5 m × 0.5 m of the Energy Laboratory of Ifes, Vitoria, Brazil. Wood spires (0.379 m) and roughness elements (cubical wooden blocks of 0.015 m) were installed upwind of test area of the wind tunnel in order to generate a neutral atmospheric boundary layer (Irwin, 1981). The mean streamwise velocity has the following power law profile:

$$\frac{U(z)}{U_\delta} = \left(\frac{y}{\delta}\right)^p \tag{1}$$

where δ = 0.30 m is the atmospheric boundary layer thickness. In this work the velocity profile was fit with p = 0.20, which corresponds to the velocity profile over flat terrain with low buildings (Blessman, 1988). The scale model building consists of a cubic block of height H_b = 0.10 m. Fig. 1 shows a schematic three dimensional flow around cubic building used for these wind tunnel experiments.

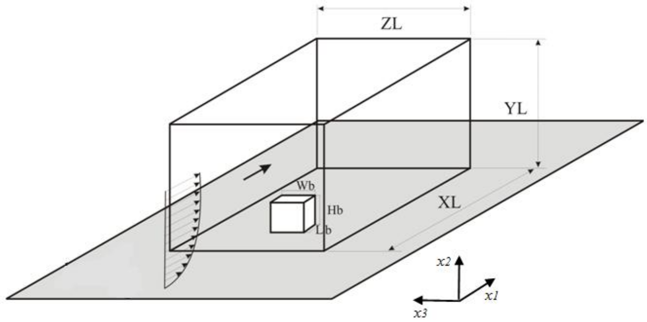


Figure 1. Sketches of the experimental flow field analyzed in this study.

2.2 Governing Equations

The governing equations for the atmospheric flow in the neutral conditions of an incompressible and Newtonian steady-state fluid with constant viscosity, μ, based on the Reynolds-averaged Navier-Stokes approach are:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial(\overline{u}_i \overline{u}_j)}{\partial x_i} + \frac{\partial(\overline{u_i' u_j'})}{\partial x_i} = \frac{\partial}{\partial x_i}\left[-\left(\frac{\overline{p}}{\rho} + \frac{2}{3}\frac{\mu}{\rho}\frac{\partial \overline{u}_k}{\partial x_k}\right)\delta_{ij} + \frac{\mu}{\rho}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)\right] \tag{2}$$

where u_i' and $\overline{u}_i$ are the fluctuating and mean velocities in the x_i-direction (i =1, 2, 3); $\overline{p}$ is the dynamic pressure and ρ is the density of the fluid. The kinematic Reynolds stress $\overline{u_i' u_j'}$ represents the turbulent fluxes of momentum. In this present work it were used the standard κ-ε model (Jones and Launder, 1972) and DSM based on the equation of ω (Wilcox (1988) for turbulence closure . The k-ε turbulence models and DSM differ precisely in the treatment of this term.

2.2 – Numerical Methods

A commercial CFD code (ANSYS CFX 13.0) was used to calculate the velocity field around building. The code solves three-dimensional Reynolds-average Navier-Stokes (RANS) equation using a UPWIND scheme. The computational domain was extended up to a distance of 3.0H_b upwind of the obstacle, 8.0H_b downwind (x_1 = 12.0H_b), 2.5H_b in each crosswind direction (x_3/2), and 5.0H_b in the vertical direction (x_2).

In order to evaluate the mesh sensitivity, three non-uniform meshes were tested: 100,000 nodes (coarse mesh), 200,000 nodes (regular mesh) and 400,000 nodes (fine mesh). The variation of the mean vertical velocity profile at

$1.0H_b$ and $3.0H_b$ downwind distance of windward building face showed little difference between the three mesh sizes. It was confirmed that the predictions results with κ-ε turbulence model did not change significantly with mesh size. Therefore, all simulations were performed using the regular mesh, Fig. 2. In this case the minimum cell width was $H_b/12$.

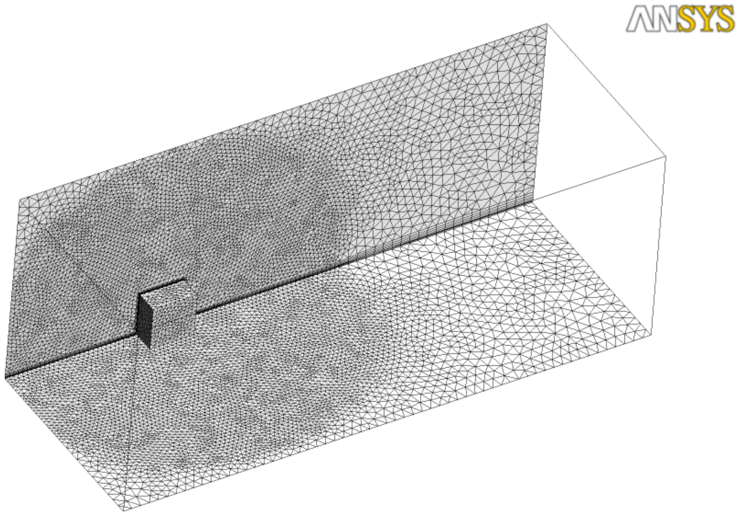


Figure 2. 3D non-uniform regular mesh for wind flow simulation around a cubic building.

2.3 – Boundary conditions of the computational flow

- In the inlet the vertical and lateral velocities are considered to be equal zero and the component in the main direction was given by equation (1).
- The inlet boundary condition for the turbulent kinetic energy was given by $\kappa = 0,025\left(U_{Hb}\right)^2$, where U_{Hb} is the velocity at the top of the building. The value of the dissipation of turbulent kinetic energy can be obtained by the expression $\varepsilon = \frac{C_\mu \kappa^{3/2}}{l}$, where $l = \left(C_\mu\right)^{1/2} \kappa^{1/2} \left(\frac{dU}{dz}\right)^{-1}$ and C_μ = 0,09 (Murakami, 1993).
- Symmetry boundary condition in central plane of the cube was imposed, because in this case exists a symmetrical nature of the flow and the equations were solved only in one half of the domain.
- The outflow boundary conditions the diffusion fluxes for all flow parameters in the direction normal were assumed to be zero. Therefore outflow velocity and pressure gradient were consistent with a fully developed flow assumption.
- At the lateral face and at the top of the computational domain was set to free-slip conditions.
- At the ground surface and at the building surfaces the velocity was set to zero by the non-slip condition.

3. RESULTS

The results of the numerical simulations of the flow around cubic building are presented. The flow structure in the vicinity of the cube was analyzed for a Reynolds number of 1.5×10^4, based on building height H_b and wind velocity at the building height, U_H = 2.30 m/s.

Figures 3 and 4 show the experimental results of the flow in front and behind the cubic building, respectively. Fig. 5 shows the side view pattern of the computed air flow in the vicinity of cubic building with two turbulence models investigated. In all figures the velocity fields can be analyzed within three different regions: frontal region, roof of the building and the recirculation bubble behind the body. Qualitatively various features of the flow were correctly captured by record of the images and numerical simulations with the two different turbulence models. In frontal face of the building the fluid flow towards the ground and return in the opposite direction to the main flow. This interaction between the incident flow and reverse flow generated the standing vortex near the ground. In the Figs. 3 and 5, was observed that the experimental and all numerical simulations clearly predicted the recirculation region behind the obstacle. It was evident that the results obtained by flow visualization and computed flow, across DSM model based on the equation of ω, were able to show the separation and the reattachment point of the boundary layer on the roof of the building (Figs. 3 and 5). However, the standard κ-ε fails to represent this phenomenon accurately. The numerical simulation using κ-ε can not to predict the separation region.

Figure 3. Side view of the incident flow around cubic building.

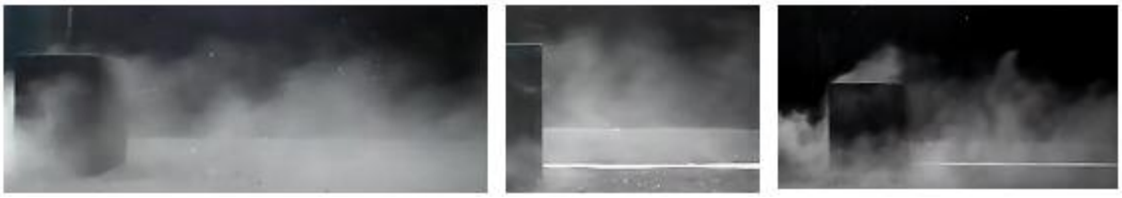
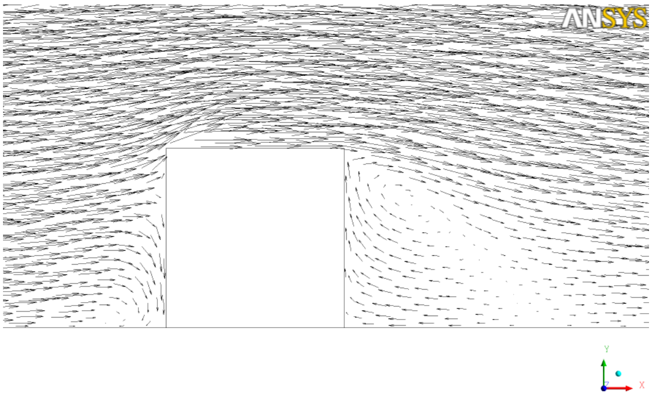
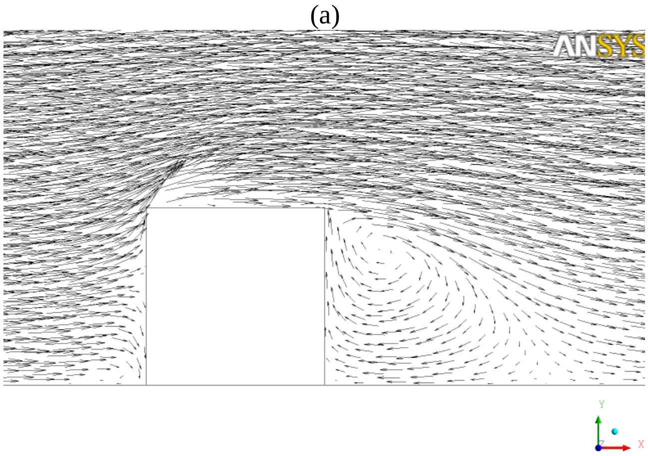


Figure 4. – Side view of the flow in near wake behind the cubic building.



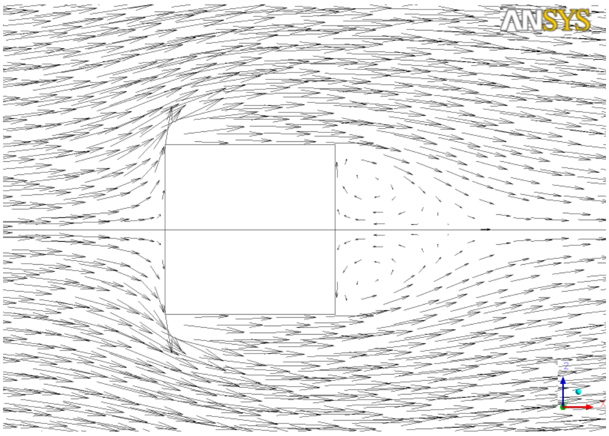
(a)



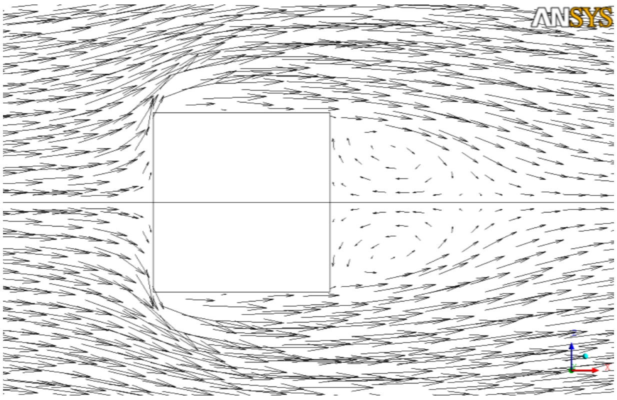
(b)

Figure 5. Side view of the computed air flow around cubic building: (a) standard κ-ε model and (b) DSM based on the equation of ω.

Figure 6 shows the computed air flow around cubic in *x-z* plane at the $y = H_b/2$ with the standard κ-ε model and DSM based on the equation of ω turbulence models. The topology of the flow patterns with both models showed that there were two symmetrical recirculation downstream of the building behind the backs faces.



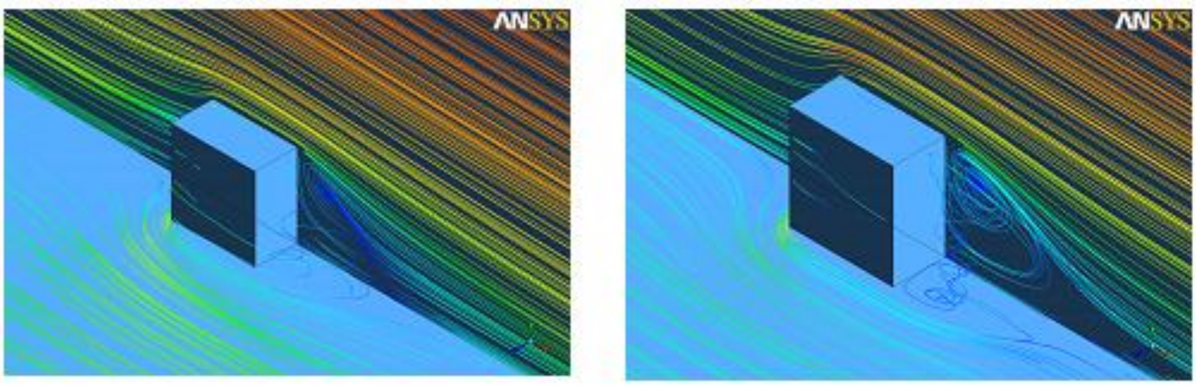
(a)



(b)

Figure 6. Computed air flow around cubic in *x*-*z* plane at the $y = H_b/2$: (a) standard κ-ε model and (c) DSM based on the equation of ω.

Figure 7 show the difference between computed streamline around cubic building with the standard κ-ε model and DSM based on the equation of ω turbulence models. The DSM turbulence model was able to capture the vortex shedding in the region of the separated flow field behind the obstacle a as show the Fig. 7.



(a) (b)

Figure 7. Streamline around cubic building: (a) standard κ-ε model and (b) DSM based on the equation of ω.

Table 1 shows the comparison of reattachment lengths on the roof (x_R/H_b) of the cube obtained by experiments and numerical simulations. These results showed that numerical predictions with standard κ-ε turbulence model was not able to capture the reverse flow on the roof. Tominaga *et al*. (2009) pointed out that the overestimation of κ in the standard κ-ε model contribute to the large mixing effects produced by the eddy viscosity which not reproduce the reverse flow on the roof. The results compared here, show the applicability of DSM based on the equation of ω the flow field around a building was good. However, this model overestimates the reattachment length on the roof.

Table 1. Reattachment lengths on the roof of the cube

Authors	x_R / *Hb*
Experiment (present work)	0.24
Standard κ-ε model (present work)	-
DSM model (present work)	0.31
Experiment (Lim *et al*., 2009)	0.20
LES (Lim *et al*., 2009)	0.85

Table 2 shows the comparison of reattachment lengths behind of the cube obtained by various experiments and numerical simulations. The x_W/H_b obtained experimentally in the present work showed good agreement with others experiments, performed to by Murakami *et al*. (1990) and Li and Meroney (1983). The numerical predictions of x_W/H_b obtained with standard κ-ε turbulence and DSM were slightly larger in comparison with those for experiments.

Table 2. Reattachment lengths behind the cubic building.

Authors	x_W / *Hb*
Experiment (present work)	1.08
Standard κ-ε model (present work)	1.1
DSM model (present work)	1.3
Experiment (Murakami *et al*., 1990)	1.2
Experiment (Li and Meroney, 1983)	1.33

4. CONCLUSIONS

In this work numerical simulations were used to investigate de wind flow in the vicinity of the cubic building and the results were compared with wind tunnel experiments. From the preceding discussions, the following conclusion can be made:

1. The numerical simulations reproduced the standing vortex in frontal face of the building like wind tunnel experiments.
2. The DSM turbulence model was able to capture the separation and the reattachment point of the boundary layer on the roof of the building. However, the standard κ–ε turbulence model has problems in predicting the reattachment point on the top of the obstacle, because this model fails to predict the separation region.
3. The numerical predictions of reattachment point in the near wake behind of the cube obtained with standard κ-ε turbulence and DSM were slightly larger in comparison with experimental results.

5. REFERENCES

ANSYS CFX 13, 2010. “Solver Manual”, CFX International, AEA Technology, UK.

Baskarn, A. and Kashef, A., 1996. “Investigation of air flow around buildings using computational fluid dynamics techniques”. *Engineering Structures*, Vol. 18, pp. 861-875.

Gao, Y. and Chow, W.K., 2005. “Numerical studies on air flow around a cube”. *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 93, pp. 115-1350.

Gousseau, P., Blocken, B. e Heijst, G. J. F. V., 2011. “CFD simulation of pollutant dispersion around isolated buildings: On the role of convective and turbulent mass fluxes in the prediction accuracy”. *Journal of Hazardous Materials*, Vol. 194, pp. 422-434.

Jones, W. P. e Launder, B. E., 1972. “The prediction of laminarization with two-equation model of turbulence”. *International Journal of Heat and Mass Transfer*, Vol. 15. pp. 301-314.

Li, W. and Meroney, R.N., 1983a. “Gas Dispersion near a cubical model building, Part I, concentration measurements.” *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 12, pp. 15-33.

Lim, H. C., Thomas, T.G. and Castro, I. P., 2009.” Flow around a cube in a turbulent boundary layer: LES and experiment.” *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 97, pp. 96-109.

Murakami S., Mochida A. and Yoshihiko H.. 1990. “Examining the k-e Model by Means of a Wind Tunnel Test and Large Eddy Simulation of the Turbulence Structure Around a Cube.” Journal of Wind Engineering and Industrial Aerodynamics, Vol.35, pp.87-100.

Murakami, S., 1993. “Comparison of various turbulence applied to a bluff body”. Journal of Wind Engineering and Industrial Aerodynamics, Vol. 46, pp. 21-36.

Patankar, S., 1980. “Numerical Heat Transfer and Fluid Flow”. Hemisphere Publishing Corporation, Second Edition, USA.

Santos, J.M., 2000. “Flow and Dispersion Around Isolated Buildings”. Ph.D. Thesis, Department of Chemical Engineering, University of Manchester, England.

Tominaga, Y., Mochidab, A., Murakami, S. and Sawaki, S., 2008. “Comparison of various revised κ-ε models and LES applied to flow around a high-rise building model with 1:1:2 shape placed within the surface boundary layer”. Journal of Wind and Engineering Industrial Aerodynamics, Vol. 96, pp. 398-411.

Tominaga, Y. and Stathopoulos, T., 2009. “Numerical simulation of dispersion around an isolated cubic building: Comparison of various types of κ – ε models”. Atmospheric Environment, Vol. 43, pp. 3200-3210.

Tominaga, Y. and Stathopoulos, T., 2010. “Numerical simulation of dispersion around an isolated cubic building: Model evaluation of RANS and LES”. Building and Environment, Vol 45, pp. 2231-2239.

Tsuchiya, M., Murakmi, S., Mochida, A., Kondo, K., and Ishida, Y., 1997. “Development of a new κ-ε model for flow and pressure fields around bluff body.” Journal of Wind Engineering and Industrial Aerodynamics, Vol. 67&68, pp. 169-182.

Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. & Speziale, C.G., 1992. “Development of turbulence models for shear flows by a double expansion technique”. Physics of Fluids A, Vol. 4, pp. 1510-1520.

Wilcox, D.C., 1988. “Mutiscale model for turbulent flows”. AIAA Journal, Vol.26.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.