Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30768

COMPUTER SIMULATIONS OF AIRFLOW FIELD AROUND A CUBE – COMPARISONS OF LES AND RANS MODELS

Behtash Tavakoli
Department of Mechanical and Aeronautical Engineering Clarkson University
Potsdam, NY, USA

Goodarz Ahmadi
Department of Mechanical and Aeronautical Engineering Clarkson University
Potsdam, NY, USA

ABSTRACT
Simulations of flow field around wall mounted square cylinders have been used extensively for validation of computational models in the literature. In this paper the airflow fields around a square cylinder were simulated using the Reynolds Averaged Navier-Stokes (RANS) models as well as the Large Eddy Simulations (LES). Particular attention was given to the case with Reynolds number of 80,000 for which the experimental data of Hussein and Matinuzzi [1] are available. The nature of the 3D wakes behind the cube as well as the vortices in front and at the back of the cube were investigated. The simulation results were compared with the experimental data and the accuracy of different models were studied. While the LES better captured the features of this separated flow, it is computationally intensive. The Reynolds Stress Transport Model (RSTM) did not properly predict some features of this separated flow, but is comparatively more economical. The accuracy of RSTM for predicting the turbulence features of separated flows was discussed, and its application for the flow around a realistic model of a building was pointed out.

Keywords: Turbulent Flow; LES; Reynolds Stress Transport model; Separation; Reattachment.

INTRODUCTION
The experimental measurements of flows around wall mounted cubes in 3D were started in the 1990s. While most reported numerical studies are for 2D geometries, 3D studies have been performed for a cube suspended in the channel or for an elongated mounted rectangular cylinder. The three dimensional numerical and experimental studies for the wall-mounted cube in a channel flow shows that the flow around the cube is distinguished by the horse shoe type vortices that initiate at the front face of the cylinder and an arc shaped vortex in the wake of the cube. Other important features of the flow are the separation at the top and side faces of the cube and vortex shedding from the block. Hussein and Martinuzzi [1,2] conducted their experiment for an inhomogeneous, three dimensional flow around a surface mounted cube in a channel. Nakmara et al. [4] experimentally investigated the flow field and the local heat transfer for the same geometry and found that the turbulent boundary layer thickness was about two times higher than the cube height. A LES model of the turbulent flow around the cylinder was carried out by Shah [5]. Several steady state RANS computations with the variety of k-ε models were presented by Rodi [6,7]. Iaccarino et al. [3] used unsteady Reynolds averaged Navier-Stokes turbulence model and showed that this model predicts the periodic vortex shedding. Furthermore, the poor agreement of steady state simulation with the experimental data is due to the assumption of statistical stationarity. Also, Murakami et al. [8], Brzoska et al. [9] and Li [10] studied the flow field and pollution around a 3D model of a cube. Robins et al. [11] studied the effect of the different packing densities of cube arrays on the flow field.

In this study, the accuracy of unsteady simulation using the Reynolds stress transport turbulence model and large eddy simulation for the flow field around a wall-mounted cube in a channel is compared together. The numerical simulation in the wake region in the back of the cube shows that the unsteady RSTM predicts the location of the reattachment point with a reasonable accuracy.

Afterward, the unsteady RSTM model is applied to a model of the Center of Excellence Building, which was recently constructed in Syracuse NY. The velocity profiles in two cross sections of the building were evaluated and compared with the experimental data.

NOMENCLATURE

H	Cube height

h Channel height

NUMERICAL SCHEME

The geometry of the model of the CoE building in the wind tunnel and the wall mounted cube is reconstructed using the GAMBIT program. The dimensions of the cube in the channel flow are the same as the ones in the experiment of Hussein and Matinuzzi [1]. Accordingly, the cube height is H=25mm and the channel height is twice the height of the cube (50mm). The inlet and outlet of the channel are, respectively, at distances of 10H and 15H from the cube. For this configuration, about 2.5×10^{6} structured and unstructured cells in 4 multiblocks were generated. Attention was given to keep the grid quite fine in the boundary layer around the cube and cell size then increased with the distance from the cube's surface.

A mesh study is considered with 1.5×10^{6} cells which the results of the velocity at 8 different points at the top and back side of the cube did not have more than 5% difference.

Similarly, in order to control the number of grids in the computational model for the CoE building, 8 mesh blocks with a combination of 3.9×10^{6} structured and unstructured cells were generated.

The numerical simulation is performed by using the commercial software of FLUENT 12.1 implementing a 12G RAM computer. The inlet is considered as a uniform velocity with 5% turbulent intensity and the boundary condition at the outlet is outflow in order to not to restrict the pressure field inside the domain.

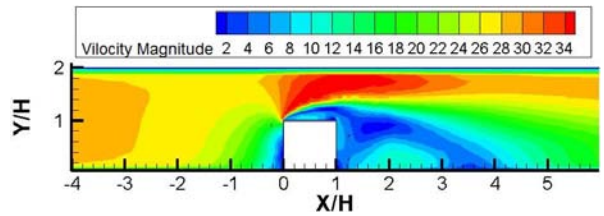
GOVERNING EQUATIONS

In this study an unsteady RANS model for an incompressible fluid were used. For simulating the turbulent air flow, the stress transport models of LRR Launder, Reece and Rodi (LRR) and the large eddy simulation of Smagorinsky-Lilly were implemented in the analysis.

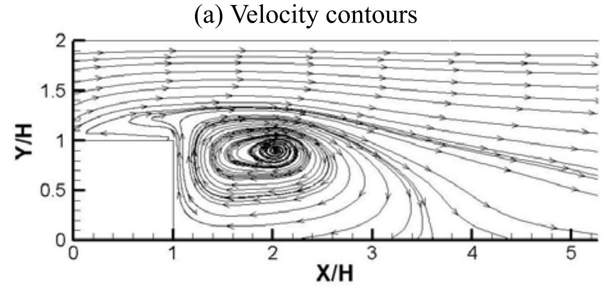
RESULTS AND DISCUSSION

The numerical simulation results for the velocity fields of the flow around a surface mounted cube with half channel height in a fully developed turbulent channel flow are presented in the first section of this study. The flow Reynolds number, based on the channel height and bulk velocity, is 80,000 which is the same as the Reynolds number used in the experiment of Hussein and Martinuzzi [1]. The cube is located sufficiently far from the channel inlet so that the airflow is nearly fully developed in front of the cube. Also, in the numerical study the channel is designed long enough to capture the wake flow behind the cube.

Figure 1 shows the streamlines and the velocity contours downstream of the leading edge of the cube using RSTM turbulence model along the plane z/H=0. In addition figure 2 depicts the velocity profiles captured from LES turbulence model at the 0.1 second of the time iteration. Comparing figures 2 and 3shows a good agreement between the RSTM and LES turbulence model in the case of the flow around the wall mounted cube.

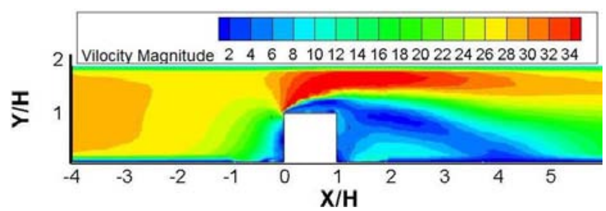


(a) Velocity contours

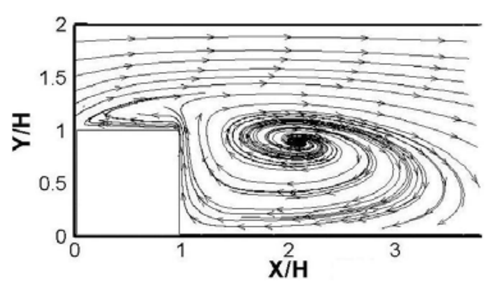


(b) Streamlines

Figure 1. Velocity at plane z/H=0 using RSTM



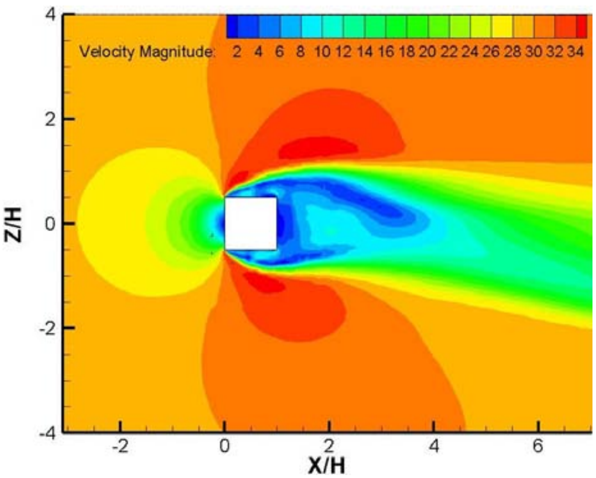
(a) Velocity contours



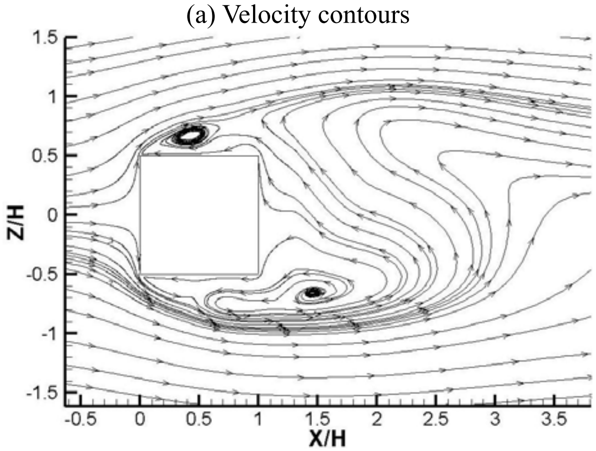
(b) Streamlines

Figure 2. Velocity at plane z/H=0 using LES

The velocity contours represent the magnitude of the projected velocity on the planes and in parallel with the bulk velocity. Figure 3 shows a snap shot of the unsteady RSTM simulation of the flow field around the cube at plane y/H=0.25, which represents the horse shoe feature of the flow. The saddle points appear on the plane parallel to the channel floor close to the side walls of the cube, which illustrates the unclosed bubble form at the wake recirculation region.



(a) Velocity contours

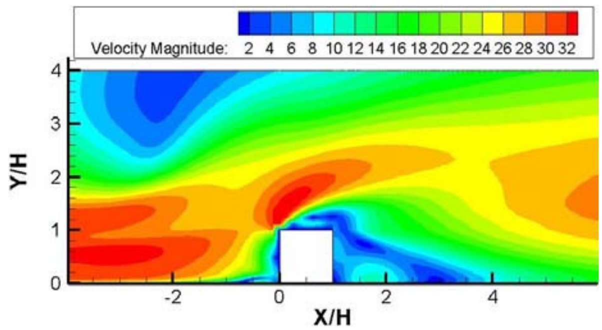


(b) Streamlines

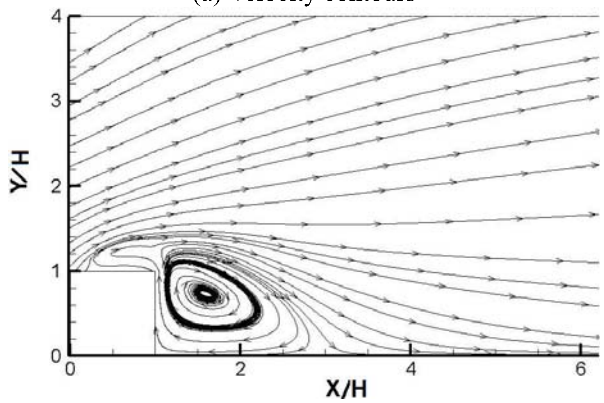
Figure 3. Velocity at y/H= 0.25

Regarding figure 1, no reattachment appears on top side of the cube at downstream of the leading edge. The gradients are so large in the region of the horseshoe vortex and the arching wake vortex.

In order to verify that the ceiling of the channel has no effect on the flow field at the top side of the cube, the cube is designed in a channel with h=8H with the same Reynolds number. Because this geometry is so large, the symmetric condition is assumed and flow is studied with steady Reynolds stress transport turbulence model. The flow field, depicted in figure 4, shows that the mean flow separates from the leading edge and it does not reattach on the top side of the cube, which confirms that the previous numerical solution is independent of the channel boundary conditions.



(a) Velocity contours



(b) Streamlines

Figure 4. Velocity features at the symmetry plane z/H=0 in the channel with the height of 8 times greater than the cube

The study of the wall mounted cube as a 3D simple geometry is conducted as a background to verify the accuracy of the unsteady RSTM in modeling more complicated geometries, like a model of the Center of Excellence building. The unsteady RSTM is implemented to capture the flow field around a model of the CoE building with the 1/116 ratio of its actual size. The CoE building was constructed in Syracuse, NY.

The schematic of the building and the planes which the mean flow is studied on are shown in Figure 5.

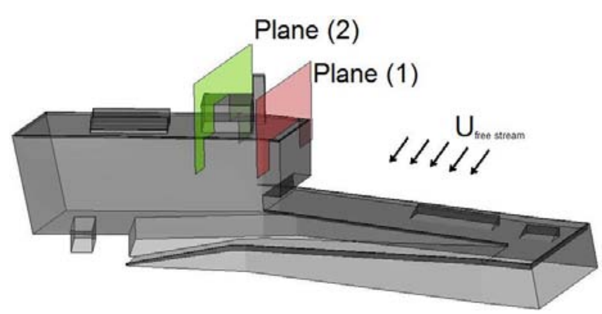
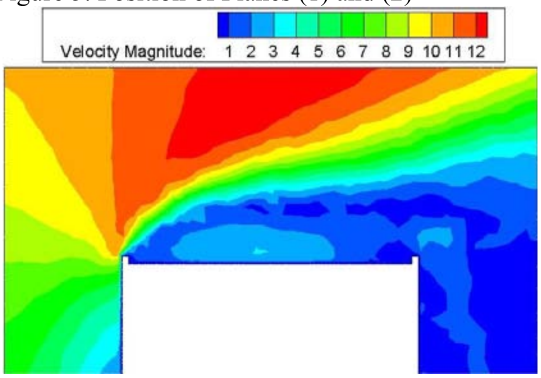
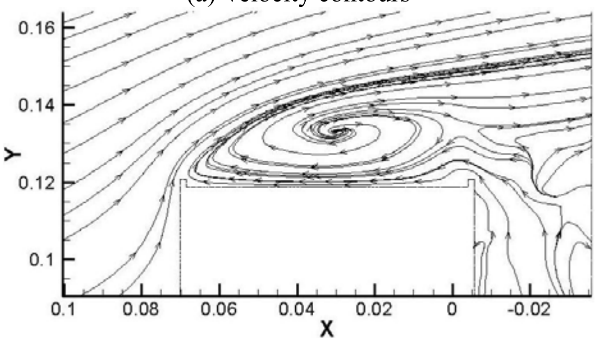


Figure 5. Position of Planes (1) and (2)



(a) Velocity contours



(b) Streamlines

Figure 6. Velocity at plane (1)

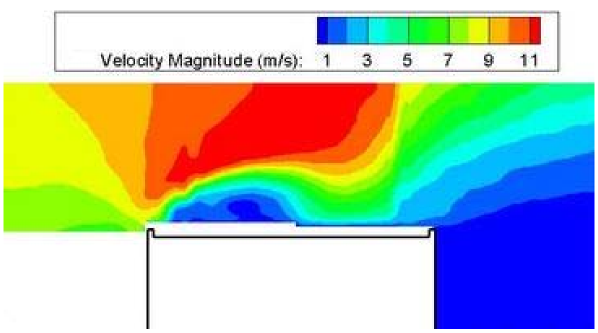
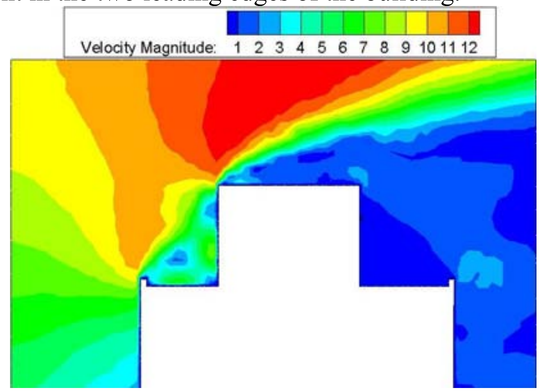


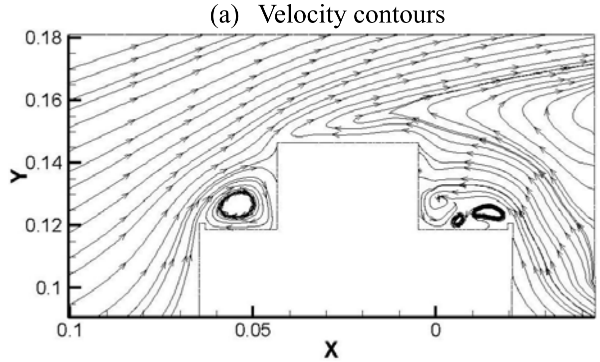
Figure 7. Velocity magnitude from PIV experiment at plane (1)

The wind speed is considered as 10m/s in direction of north to south of the building. The averaged maximum height of the model is 0.32m and it is located in the fully developed region of the flow in the wind tunnel.

Figures 6 and 8 show the RSTM simulation of the flow field on the planes (1) and (2) respectively. As it is illustrated, wind flow separates at the leading edge of the building and the recirculation region appears at the top and back sides of the obstacle. The geometry is more complex in figure 6, and two separations are present in the two leading edges of the building.



(a) Velocity contours



(b) Streamlines

Figure 8. Velocity at plane (2)

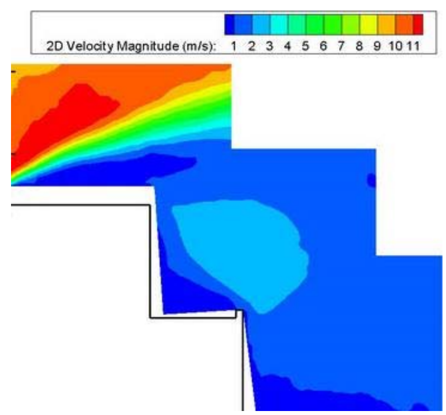


Figure 9. Velocity magnitude from PIV experiment at plane (2)

Moreover, the streamlines show two vortex regions, one at the upstream part and the other one at the back of the building.

Figures 7 and 9 are the PIV measurements of the velocity at wind speed of 10m/s. The comparison of the PIV results with the RSTM numerical simulation shows that they are generally in reasonable agreement in terms of the mean flow field.

CONCLUSIONS

The unsteady Reynolds Stress transport, as well as the LES turbulence model are implemented to study the flow field around a wall mounted cube in a turbulent channel flow, which showed that the numerical simulation could predict the separation and reattachments reasonably and the RSTM model is in good agreement with the LES model in a simple case of the cube. Therefore, the unsteady RSTM turbulence model is implemented in simulating the wind flow around an actual model of a research building as a complicated 3D geometry and the mean features of the velocity field were in good agreement with the PIV measurements.

REFERENCES

[1] Hussein H. J., Martinuzzi R. J., 1996, Energy balance for turbulent flow around a suface mounted cube placed in a channel, Journal of phys. Fluid, 8 (3), 764-780.

[2] Martinuzzi R. J., Tropea C., 1993, The flow around surface mounted prismatic obstacles placed in a fully developed channel flow. Trans. ASME, J. Fluid Eng. 115 (1), 85-91.

[3] Iaccarino G., Ooi A., Durbin P. A., Behnia M., 2003, Reynolds Averaged simulation of unsteady separated flow. Heat and Fluid Flow, 24, 147-156.

[4] Nakamura H., Igarashi T., Takayuki T., 2001, local heat transfer around wall mounted cube in the turbulent boundary layer. Int. J. Heat Mass Transfer, 44, 3385-3395.

[5] Shah K. B., 1998, Large Eddy Simulation of the Flow Past a Cubic Obstacle. Ph.D. Thesis

[6] Rodi W., 2002, Closure strategies for turbulent and transitional lows in: Launder B. E.,Sandham N. D., (Eds.), Large Eddy Simulations of the Flow Past Bluff Bodies. Cambridge University Press, pp. 361-391.

[7] Rodi W., Ferziger J. H., Breuer M., Pourquie M., 1997. Status of large eddy Simulation: result of a workshop. J. Fluids Eng. 119, 248-262.

[8] Tominaga Y., Murakami S., Mochida A., 1997. CFD Prediction of Gaseous Diffusion around a Cubic Model Using a Dynamic Mixed SGS Model Based on Composite Grid Technique. J. Wind Engineering and Industrial Aerodynamics. 67&68, 827-841.

[9] Brzoska M., A., Stock D., Lamd B., 1997. Determination of Plum Capture by the Building Wake. J. Wind Engineering and Industrial Aerodynamics. 67&68. 909-922.

[10] Li Y., Stathopoulos T., 1997. Numerical Evaluation of Wind-Induced Dispersion of Pollutants around a Building. J. Wind Engineering and Industrial Aerodynamics. 67&68. 757-766.

[11] Cheng H., Hayden P., Robins A., G., Castro I., P., 2007. Flow over Array cubes of Different Packing Densities. J. Wind Engineering and Industrial Aerodynamics. 95. 715-740.