11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

SPECIFIC ASPECTS OF NUMERICAL SIMULATION OF CIVIL ENGINEERING STRUCTURES WITH CROSS SECTION SHAPE CLOSE TO RECTANGULAR

Irina N. Afanasyeva[1], Anton R. Usmanov[2], Alexandr M. Belostotskiy[3], Sergey I. Dubinsky

[1,2] Postgraduate, Department of Applied Mathematics and Computer Science,
Moscow State University of Civil Engineering (MGSU),
129337 Moscow, Yaroslavskoe shosse 26, Russia, niccm@mgsu.ru
[3] Professor, Department of Applied Mathematics and Computer Science,
Moscow State University of Civil Engineering (MGSU),
129337 Moscow, Yaroslavskoe shosse 26, Russia, niccm@mgsu.ru
[4] Associate Professor, Department of Applied Mathematics and Computer Science,
Moscow State University of Civil Engineering (MGSU),
129337 Moscow, Yaroslavskoe shosse 26, Russia, niccm@mgsu.ru

Key Words: *Civil Engineering, CFD, Numerical Simulation, Turbulence models.*

Present paper is devoted to the numerical simulation of the aerodynamics of structures, which shape is similar to a rectangular cylinder (e.g., bridges, high-rise buildings etc.). For considered structures wind loads are interesting as local wind loads on facades or as a dynamic loads leading to resonance effects like galloping and flutter. From this point of view it would be more interest and important to solve problem using Fluid Structure Interface (FSI) methods, but it is just a next step, because first of all it is necessary to obtain authentic transient wind loads on building and structures using Computational Fluid Dynamic (CFD) technologies.

All methods of fluid dynamics numerical simulation solve the Navier-Stocks equations. For the turbulence flow there are additional equations named as turbulence models: RANS (Reynolds Averaged Navier-Stoks), LES (Large Eddy Simulation) и DES (Detached Eddy Simulation) [1]. To use this approaches by right way and obtained good results there are requirements according to numerical schemes, additional parameters and especially to mesh qualities. RANS turbulence models have proven itself in the definition of the mean velocity and pressure fields and leading load frequency and less demanding on the mesh quality as well. This fact is important for creating numerical grid for buildings and civil structures of complex form and allows to solve this kind of problem in a short timeframe. But to accurate estimation of dynamic loads and three-dimensional effects of flow around the obstacle it is necessary to use LES group of turbulence models.

The purpose of the research is to assess the applicability of such turbulence models as LES (Large Eddy Simulation) and DES (Detached Eddy Simulation) demands for high mesh qualities for civil structures and buildings of complex three-dimensional shape. Therefore, there is need for investigation of the grid resolution in the boundary layer (near the walls of obstacles), in the transverse direction to the cross section of structure and shape and quality of mesh in the domain. This kind of research will give understanding of "the cost" of the obtaining authentic results using LES approach, the limit of the mesh simplification near and around the object for DES approach and the value of an error using RANS turbulence models.

The multivariate verification investigations is carried out at the world-known benchmark BARC [2], that is devoted to 1:5 prism aerodynamics research. The used control parameters are the mean and standard deviation of the pressure coefficient (C_p) along the prism side, friction coefficient (C_f), the aerodynamic forces (drag, lift and torque moment), Strouhal number (*Sh*) and coherence function of surface pressure at specific points. The comparison data was taken from wind tunnel experiment carried out by Bronkhorst [3].

The significant dependence numerical results on grid resolution in the boundary layer (especially for LES), on the aspect ratio of the elements edges (in three directions) and on the size of domain in the transverse direction to the cross section of structure was found out. The obtained results showed, that LES turbulence model is fairly heavy to application at real buildings and structures – the mesh must be very fine (about 3 000 000 cells) to obtain consistent results while the DES model requirement is 10 times less than LES model. Therefore DES model is far more suitable for the application to the civil problems – including the bridge and high-rise buildings aerodynamics investigation.

Specific aspects, rules and recommendations for the numerical simulation of civil engineering structures with cross sectional shape close to a rectangular were developed.

This technique was applied on the simplified bridge model (a rectangular with aspect ratio 1:10 in bridge true scale and with imposed appropriate boundary conditions) and on the real complex bridge model. Obtained numerical results compared with experimental data from wind tunnel. The purpose was to estimate practicability of two-dimensional/quasi-two-dimensional simulation versus to three-dimensional simulation of the bridge deck aerodynamics and to clarify the developed recommendations for more complex structures than a rectangular cylinder.

REFERENCES

[1] ANSYS, Inc. (2013), "ANSYS 14.5 Help".

[2] L. Bruno, D.Fransos, N. Coste, and A. Bosco, 3D flow around a rectangular cylinder: a computational study, *BBAA* VI *Colloquium on: Bluff Bodies Aerodynamics & Applications*, Proceedings, Milano, Italy, July, 20-24 2008.

[3] A.J. Bronkhorst, C.P. Geurts, C.A.van Bentum, Unsteady pressure measurements on a 5:1 rectangular cylinder. *13th International Conference on Wind Engineering*, Proceedings, July 10-15, 2011.